

SEEXCLUDE

Superelement Matrix and Load Assembly Exclusion

Specifies the superelement identification numbers for which all matrices and loads will not be assembled into the downstream superelement.

Format:

$$\text{SEEXCLUDE} = \begin{Bmatrix} \text{ALL} \\ n \\ i \end{Bmatrix}$$

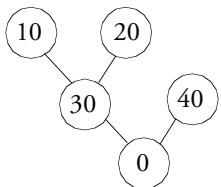
Examples:

`SEEXCLUDE=ALL`
`SEEXCLUDE=18`

Describer	Meaning
ALL	All upstream superelements will be excluded.
n	Set identification number of a previously appearing SET command. Only those superelements with identification numbers that appear on this SET command will be excluded (Integer > 0).
i	Identification number of a single superelement for which matrices will be excluded (Integer > 0).

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
3. This command is not meaningful when applied to the residual structure.
4. For a further discussion of this command, see the *MSC Nastran Handbook for Superelement Analysis*.
5. If the SEEXCLUDE command is specified in a restart of SOLs 101 through 200, then PARAM,SERST,MANUAL must be specified. Also, the SEKR command must be specified for the superelement immediately downstream from the excluded superelement. For example, if superelement 10 is excluded in the following superelement tree:



then the user must specify the following commands in the Case Control Section:

`SEKR = 30`
`PARAM, SERST, MANUAL`

SEFINAL

Final Superelement for Assembly

Specifies the superelement identification number for the final superelement to be assembled.

Format:

$$\text{SEFINAL} = \left\{ \begin{array}{c} n \\ i \end{array} \right\}$$

Example:

SEFINAL=14

Descriptor	Meaning
n	Set identification of a previously appearing SET command. Each superelement identification number appearing on the SET command must belong to a disjoint model (Integer > 0).
i	Identification number of the final superelement to be assembled (Integer > 0).

Remarks:

1. If this command is not present, the program selects the order of the superelements for assembly operations.
2. This command, if present, must be located before the first SUBCASE command.
3. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
4. This command can be used on restarts to ensure that minor modeling changes do not also change the processing order. For this usage, inspect the SEMAP table to determine which superelements were final superelements on the prior run.
5. See the *MSC Nastran Handbook for Superelement Analysis* for a further discussion of this command.

SEKREDUCE

Superelement Stiffness Matrix Assembly and Reduction

Specifies the superelement identification numbers for which stiffness matrices are assembled and reduced.

Format:

$$\text{SEKREDUCE} = \left\{ \begin{array}{l} \text{ALL} \\ n \\ i \end{array} \right\}$$

Examples:

SEKREDUCE=ALL

SEKREDUCE=9

Descriptor	Meaning
ALL	Assembles and reduces matrices for all superelements.
n	Set identification number of a previously appearing SET command. Matrices will only be assembled for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which the stiffness matrix will be assembled and reduced (Integer > 0).

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. SEKREDUCE is an alternate form, and is entirely equivalent to the obsolete command SEMASSEMBLE.
5. SEALL=ALL is the default, but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SELGENERATE

Superelement Load Generation

Specifies the superelement identification numbers for which static loads will be generated.

Format:

$$\text{SELGENERATE} = \left\{ \begin{array}{l} \text{ALL} \\ n \\ i \end{array} \right\}$$

Examples:

`SELGENERATE=ALL`
`SELGENERATE=18`

Descriptor	Meaning
ALL	Generates static loads for all superelements.
n	Set identification number of a previously appearing SET command. Static load matrices will only be generated for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which load matrices will be generated (Integer > 0).

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. SEALL=ALL is the default, but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SELREDUCE**Superelement Load Assembly and Reduction**

Specifies the superelement identification numbers for which the static load matrices will be assembled and reduced.

Format:

$$\text{SELREDUCE} = \left\{ \begin{array}{l} \text{ALL} \\ n \\ i \end{array} \right\}$$

Examples:

`SELREDUCE=ALL`
`SELREDUCE=9`

Describer	Meaning
ALL	Assembles and reduces matrices for all superelements.
n	Set identification number of a previously appearing SET command. Matrices will be assembled only for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which the load matrices will be assembled and reduced (Integer > 0).

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only be appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. This command is used on restarts to selectively assemble and reduce load matrices.
5. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
6. In superelement static analysis, SELREDUCE is equivalent to SELASSEMBLE.
7. In dynamic analysis, SELASSEMBLE combines the functions of SELREDUCE and SEMREDUCE.
8. SEALL=ALL is the default, but can be overridden by specifying and of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SEMGENERATE**Superelement Matrix Generation**

Specifies the superelement identification numbers for which stiffness, mass, and damping matrices will be generated.

Format:

$$\text{SEMGENERATE} = \left\{ \begin{array}{l} \text{ALL} \\ n \\ i \end{array} \right\}$$

Examples:

SEMGENERATE=ALL
SEMGENERATE=7

Descriptor	Meaning
ALL	Generates structural matrices for all superelements.
n	Set identification number of a previously appearing SET command. Structural matrices will only be generated for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which structural matrices will be generated (Integer > 0).

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. SEALL=ALL is the default, in the structured SOLs 101 through 200. This default can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SEMREDUCE**Superelement Mass and Damping Assembly and Reduction**

Specifies the superelement identification numbers for which the mass and damping matrices will be assembled and reduced. In buckling analysis, the differential stiffness matrices will be assembled and reduced.

Format:

$$\text{SEMREDUCE} = \left\{ \begin{array}{l} \text{ALL} \\ n \\ i \end{array} \right\}$$

Examples:

SEMREDUCE=ALL

SEMREDUCE=9

Describer	Meaning
ALL	Assembles and reduces matrices for all superelements.
n	Set identification number of a previously appearing SET command. Matrices will only be assembled for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which the load matrices or the mass and damping matrices will be assembled and reduced (Integer > 0).

Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only appear as a member of a set.
3. This command is used on restart to selectively assemble and reduce mass and damping matrices. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. In dynamic analysis, SELASSEMBLE combines the functions of SELREDUCE and SEMREDUCE.
6. This command has no function in static analysis.
7. SEALL=ALL is the default but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

SERESP**Superelement Response Sensitivity**

Specifies the superelement identification numbers for which the design sensitivity matrices will be generated.

Format:

$$SERESP = \left\{ \begin{array}{l} ALL \\ n \\ i \end{array} \right\}$$

Examples:

SERESP=ALL
SERESP=18

Descriptor	Meaning
ALL	Requests design sensitivity matrix generation for all superelements. This is the default value if SERESP is missing.
n	Set identification number of a previously appearing SET command. Design sensitivity matrices will be generated for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which the design sensitivity matrix will be generated.

Remarks:

1. This command, if present, then it must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see the [MSC Nastran Reference Manual](#).
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. If both the SEDV and SERESP commands are not present, then the design variable processing and design sensitivity matrix generation will be performed for all superelements.

SET

Set Definition, General Form

Sets are used to define the following lists:

1. Identification numbers (point, element, or superelement) for processing and output requests.
2. Frequencies for which output will be printed in frequency response problems, or times for transient response, using the OFREQ and OTIME commands, respectively.
3. Surface or volume identification numbers to be used in GPSTRESS or STRFIELD commands.
4. DRESP1 entries that are used in the spanning of subcases.
5. Grid point number and component type code to be used by the MCFRACTION command.

Formats:

SET n = {*i*₁[,*i*₂, *i*₃, THRU *i*₄, EXCEPT *i*₅, *i*₆, *i*₇, *i*₈, THRU *i*₉]}

SET n = {*r*₁[,*r*₂, *r*₃, *r*₄]}

SET = ALL

SET n = {*i*₁/*c*₁[,*i*₂/*c*₂, *i*₃/*c*₃, *i*₄/*c*₄]}

SET n = {*l*₁, [*l*₂, *l*₃]}

Examples:

```
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
```

```
SET 101=1.0, 2.0, 3.0
SET 105=1.009, 10.2, 13.4, 14.0, 15.0
SET 1001=101/T1, 501/T3, 991/R3
SET 2001=M1,M2
```

Descriptor	Meaning
n	Set identification number. Any set may be redefined by reassigning its identification number. SETs specified under a SUBCASE command are recognized for that SUBCASE only (Integer > 0).
<i>i</i> ₁ , <i>c</i> ₁	Grid point identification numbers and component codes. The <i>c</i> values must be of T1, T2, T3, R1, or R3.
<i>l</i> ₁ , <i>l</i> ₂ etc.	Identification names of literals used for matrix or group selection.
<i>i</i> ₁ , <i>i</i> ₂ etc.	Identification numbers. If no such identification number exists, the request is ignored (Integer \geq 0).
<i>i</i> ₃ THRU <i>i</i> ₄	Identification numbers (<i>i</i> ₄ > <i>i</i> ₃) (Integer > 0).

Descriptor	Meaning
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. An EXCEPT list may not include a THRU list or ALL.
$r_1, r_2, \text{etc.}$	Frequencies or times for output. The nearest solution frequency or time will be output. EXCEPT and THRU cannot be used. If an OFREQ or OTIME command references the set then the values must be listed in ascending sequences, $r_1 < r_2 < r_3 < r_4 \dots \text{etc.}$, otherwise some output may be missing. If an OFREQ or OTIME command is not present, all frequencies or times will be output (Real > 0.0).
ALL	All members of the set will be processed.

Remarks:

1. A SET command may be more than one physical command. A comma at the end of a physical command signifies a continuation command. Commas may not end a set. THRU may not be used for continuation. Place a number after the THRU.
2. Set identification numbers following EXCEPT within the range of the THRU must be in ascending order.
3. In SET 88 above, the numbers 77, 78, etc., are included in the set because they are outside the prior THRU range.
4. SET commands using the grid point/component code format cannot contain THRU. SETs using this format should be selected only by the MCFRACTION Case Control command.
5. SET commands using literals apply only to direct matrix input such as K2PP etc. or FLSPROUT panel grouping.

SETP**Process Set Definition**

Process sets are used to define lists of SET identifications to be processed individually for data recovery:

Formats:

SETP n = {i₁[i₂, i₃THRU i₄EXCEPT i₅, i₆, i₇, i₈THRU i₉]}

Examples:

SETP 77=5, 6

SETP 88=5, 6, 7, 8, 9, 10 THRU 55

Descriptor	Meaning
n	SETP identification number. Any SETP may be redefined by reassigning its identification number. SETPs specified under a SUBCASE command are recognized for that SUBCASE only (Integer > 0).
i ₁ , i ₂ , ..., i _n	SET identification numbers i ₁ , i ₂ , etc. If no such identification number exists, the request is ignored (Integer > 0).
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. An EXCEPT list may not include a THRU list or ALL.

Remarks:

1. A SETP command may be more than one physical command. A comma at the end of a physical command signifies a continuation command. Commas may not end a set. THRU may not be used for continuation—place a number after the THRU.
2. Set identification numbers following EXCEPT,k within the range of the THRU, must be in ascending order.
3. SETP usage is limited to the EDE, EKE and ESE Case Control commands.

SETS DEFINITION

Case Control Processing Delimiter

Delimits the various type of commands under grid point stress and/or p-version element set definitions. This command is synonymous with OUTPUT(POST).

Format:

SETS DEFINITION

Example:

SETS DEFINITION

Remark:

1. Either SETS DEFINTIIION or OUTPUT(POST) may be specified, but not both.

SKIP

Case Control Processing Delimiter

Activates or deactivates the execution of subsequent commands in Case Control (including plot commands).

Format:

SKIP { *ON* }
 { *OFF* }

Example:

SKIPOFF

Remarks:

1. SKIPON and SKIPOFF commands may appear as many times as needed in the Case Control Section.
2. Commands that are skipped will be printed.
3. SKIPON ignores subsequent commands until either a SKIPOFF or BEGIN BULK command is encountered. This allows the user to omit requests without deleting them from the data. In the following example, plot commands will be skipped.

```
TITLE=EXAMPLE
SPC=5
LOAD=6
SKIPON$SKIP PLOT REQUEST
OUTPUT (PLOT)
SET 1 INCLUDE ALL
FIND
PLOT
BEGIN BULK
```

SMETHOD

Iterative Solver Method Selection

Selects iterative solver method and parameters.

Format:

$$\text{SMETHOD} \left\{ \begin{array}{l} \text{ELEMENT} \\ n \\ \text{MATRIX} \end{array} \right\}$$

Example:

```
SMETHOD = ELEMENT $ selects element-based iterative solver defaults.
SMETHOD = MATRIX $ selects matrix based iterative solver defaults.
SMETHOD = 1000 $ specifies ID of ITER Bulk Data entry to select
iterative.
```

Descriptor	Meaning
ELEMENT	Selects the element-based iterative solver with default control values.
MATRIX	Selects the matrix-based iterative solver with default control values.
n	Sets identification of an ITER Bulk Data entry (Integer > 0).

Remarks:

1. The matrix-based iterative solver is available in SOLs 101, 106, 108, 111, 153, and 400 and allows use of all features.
2. The element-based iterative solver is only available in SOLs 101, 200 and 400. SMETHOD must be placed above all SUBCASEs in this case. It is intended primarily for very large solid element models and does not handle p-elements. See the ITER Bulk Data entry for a list of restrictions in addition to details on setting the convergence parameter epsilon.
3. The element-based iterative solver can be used with SMP by setting smp=number on the command line. Please refer to [smp, 11](#) of this manual for correct usage.
4. For SOL 600, the iterative solver is activated using the MARCSOLV PARAM.
5. GPGPU devices are not supported for iterative methods.

SOLUTION**Solution ID Selection**

Selects the solution ID for a 3rd step external superelement data recovery restart in SOL 400.

Format:

SOLUTION = n

Example:

SOLUTION=10

Descriptor	Meaning
n	Solution identification number.

Remarks:

1. SOLUTION command is used only in SOL 400 for 3rd step external superelement data recovery restart.
2. See Remark 8. under EXTDRIN Case Control command's description for a further explanation and example.

SPC Single Point Constraint Set Selection

Selects a single point constraint set to be applied.

Format:

SPC = n

Example:

SPC=10

Descriptor	Meaning
n	Set identification number of a single-point constraint that appears on an SPC, SPC1, SPC2 (SOL 700), FRFSPC1 (in FRF Based Assembly or FBA process) or SPCADD Bulk Data entry (Integer > 0).

Remarks:

1. In cyclic symmetry analysis, this command must appear above the first SUBCASE command.
2. Multiple boundary conditions are only supported in SOLs 101, 103, 105, 145, and 200. Multiple boundary conditions are not allowed for upstream superelements. The BC command must be specified to define multiple boundary conditions for the residual structure in SOLs 103, 105, 145, and 200.

SPCFORCES

Single Point Forces of Constraint Output Request

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

Format:

$$\text{SPCFORCES} \left[\begin{array}{c} \left[\begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[\begin{array}{c} \text{PRINT}, \text{PUNCH} \\ \text{PLOT} \end{array} \right], \left[\begin{array}{c} \text{REAL or NOZPRINT} \\ \text{PHASE} \end{array} \right] \end{array} \right] \left[\begin{array}{c} \left[\begin{array}{c} \text{PSDF}, \text{ATOC}, \text{CRMS} \\ \text{or RALL} \end{array} \right], \\ \left[\begin{array}{c} \text{RPRINT} \\ \text{NORPRINT} \end{array} \right], \text{RPUNCH}, [\text{CID}] \end{array} \right] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Examples:

```
SPCFORCES = 5
SPCFORCES(SORT2, PUNCH, PRINT, IMAG) = ALL
SPCFORCES(PHASE) = NONE
SPCFORCES(SORT2, PRINT, PSDF, CRMS, RPUNCH)=20
SPCFORCES(PRINT, RALL, NORPRINT)=ALL
```

Descriptor	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.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

Descriptor	Meaning
NOZPRINT	Print only nonzero SPC forces appearing in SORT2 output. This keyword does not affect SORT1 output.
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. The request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
CRMS	Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
PRUNCH	Writes random analysis results in the punch file.
CID	Request to print output coordinate system ID in printed output file (.f06).
ALL	Single point forces of constraint for all points will be output. See Remarks 1. and 4.
NONE	Single point forces of constraint for no points will be output.
n	Set identification of a previously appearing SET command. Only single point constraint forces for points with identification numbers that appear on this SET command will be output (Integer > 0).

Remarks:

1. See Remark 1 under [DISPLACEMENT \(Case\), 298](#) for a discussion of SORT1 and SORT2. In the SORT1 format, only nonzero values will be output.
2. In a statics problem, a request for SORT2 causes loads at all points (zero and nonzero) to be output.
3. SPCFORCES=NONE overrides an overall output request.
4. In SORT1 format, SPCFORCES recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. However, if a scalar point is not consecutively numbered, it will begin a new sextet on a new line of output. If a sextet can be formed and all values are zero, then the line will not be printed. If a sextet cannot be formed, then zero values may be output.
5. SPCFORCES results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
6. In SOLs 129 and 159, SPCFORCES results do not include the effects of mass and damping elements.

7. In all solution sequences except SOLs 129 and 159, SPCFORCES results do include the effects of mass and damping, except damping selected by the SDAMPING Case Control command. PARAM,DYNNSPCF,OLD may be specified to obtain SPCFORCES results, which do not include mass and damping effects.
8. In inertia relief analysis, the SPCFORCES output is interpreted differently for SOLs 1, 101, and 200:
 - a. In SOL 1, the SPCFORCE output reflects the effects due to the applied loads only, and not the inertial loads.
 - b. In SOLs 101 and 200, the SPCFORCE output includes both the effects due to inertial loads and applied loads.
9. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
10. Note that the CID keyword affects only grid point related output such as DISP, VELO, ACCE, OLOAD, SPCF and MPCF. In addition, the CID keyword needs to appear only once in a grid-related output command, anywhere in the Case Control Section, to turn on the printing algorithm.

SPLINOUT

Export of Spline Matrix

Requests output of the spline matrix for external use.

Format:

$$SPLINOUT \left[\left[\begin{array}{l} BOTH \\ DISP \\ FORCE \end{array} \right], \left\{ \begin{array}{l} (OP2 = unit) \\ DMIPCH \end{array} \right\} \right]$$

Examples:

Descriptor	Meaning
BOTH	Requests output of the force and displacement splines (Default).
DISP	Requests output of only the displacement splines.
FORCE	Requests output of only the force splines.
OP2	Requests output to an .op2 file
unit	Unit the .op2 file is assigned to
DMIPCH	Requests output to a .pch file.

Remarks:

1. Matrices are output in external sort.
2. If displacement and force splines are identical, only displacement splines are output.
3. Option FACTORS is only allowed together with option OP2.
4. If $OP2 = unit$ is specified, a table relating the matrix columns to structural degrees of freedom, and the matrix rows to aerodynamic degrees of freedom, will be written to the .op2 file.
5. If $OP2 = unit$ is specified, an appropriate ASSIGN OP2 statement must be present in the File Management Section for this unit.
6. If DMIPCH is specified, DMI entries are written to the.pch file.

STATSUB

Static Solution Selection for Differential Stiffness

Selects the static solution to use in forming the differential stiffness for static analysis, buckling analysis, normal modes, complex eigenvalue, frequency response and transient response analysis.

Format:

$$\text{STATSUB} \left(\begin{matrix} \text{BUCKLING} \\ \text{PRELOAD} \end{matrix} \right) = n$$

Examples:

`STATSUB=23`

`STAT=4`

`STATSUB (PREL)=7`

Descriptor	Meaning
BUCKLING	Subcase ID number corresponding to static subcase of buckling or varying load (Default in buckling analysis).
PRELOAD	Subcase ID number corresponding to static subcase of preload or constant load (Default in dynamic analysis).
n	Subcase identification number of a prior subcase specified for static analysis (Integer > 0).

Remarks:

1. STATSUB may be used in SOLs 101, 103, 105, 107 through 112, 115, 116, 200 and SOL 400 (ANALYSIS = BUCKLING only in SOL 200 and SOL 400).
2. STATSUB must be specified in the same subcase that contains the METHOD selection for buckling or normal modes, CMETHOD for complex eigenvalue analysis, TSTEP for transient response, and FREQ for frequency response.
3. In SOL 105, if it is intended that results from the first static subcase are used to compute the differential stiffness, then the STATSUB command is not required. That is, the default for STATSUB is the first static subcase identification. In SOLs 101, 103 and 107 through 112, 115, and 116, STATSUB must reference a separate static subcase.
4. In dynamic analysis, only one STATSUB command may be specified in each dynamic subcase. In buckling analysis with a preload, both STATSUB(BUCKLING) and STATSUB(PRELOAD) must be specified in each buckling subcase. STATSUB(PRELOAD) is not supported in SOL 200 or SOL 400. Buckling Analysis with a preload is not supported in SOL 200 and SOL 400.
5. In dynamic analysis, any subcase that does not contain a CMETHOD command in SOLs 107 and 110, a FREQUENCY command in SOLs 108 and 111, and a TSTEP command in SOLs 109 and 112, will be treated as a static subcase.
6. SOL 200 and SOL 400 support linear buckling analysis only, but do not support post-buckling (nonlinear buckling analysis)

7. In versions prior to 2018, if it was desired to have a dynamic subcase **with** the effects of preload and another subcase **without** the effects of preload, then a static subcase with null load was required and its ID was referenced by STATSUB in the dynamic subcase **without** the effects of preload. In version 2018, STATSUB in the dynamic subcase **without** the effects of preload is no longer required. If the STATSUB is not removed and still references the static subcase with null load, then User Fatal Message 9244 will be issued.

STEP**Step Delimiter**

Delimits and identifies a nonlinear analysis step for SOL 400.

Format:

STEP=n

Examples:

STEP=10

Descriptor	Meaning
n	Step identification number (Integer > 0).

Remarks:

1. The STEP command can only be used in nonlinear solution sequence SOL 400 (NONLIN).
2. The STEP command is to be used below the SUBCASE Case Control command. If no SUBCASE is specified, MSC Nastran creates a default SUBCASE 1.
3. The STEP identification number n in a SUBCASE must be in increasing order, and less than 9999999.
4. The following example illustrates a typical application of SUBCASE and STEP:

```
SUBCASE 1
    STEP 1
        LOAD = 1
    STEP 2
        LOAD = 2
SUBCASE 2
    STEP 10
        LOAD = 10
    STEP 20
        LOAD = 20
```

5. The solutions of all SUBCASEs are independent of each other. However, the solution of any STEP is a continuation of the solution of the previous STEP.

STOCHASTICS

Randomization of Model Parameters

Request randomization of all or selected subsets of model parameters.

Format:

$$STOCHASTICS = \left\{ \begin{array}{l} ALL \\ n \end{array} \right\}$$

Examples:

STOCHASTICS=10

Descriptor	Meaning
ALL	All real values of C-entries, M-entries, P-entries, loading entries, and SPCD entries are to be randomized.
n	Set identification number of a STOCHAS Bulk Data entry (Integer > 0).

Remarks:

1. Only one STOCHASTICS command may appear in the Case Control Section and should appear above all SUBCASE commands.
2. The STOCHASTICS = n command may be used to request randomizing a set of analysis model parameters with user specified statistics. (See Remark 1 of the STOCHAS Bulk Data entry.)
3. The default (STOCHASTICS = all) randomizes all scalar analysis model parameters that are real values on the C-entries, M-Entries, P-entries, all loading entries, and SPCD entries with default coefficients of variance (0.05) and multipliers of standard deviations (m=3.).
4. This command will only invoke a single Nastran randomization run. Separate runs can be submitted to achieve different randomizations.

STRAIN

Element Strain Output Request

Requests the form and type of strain output.

Format:

$$\text{STRAIN} \left[\begin{array}{c} \left[\begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[\begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] \left[\begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[\begin{array}{c} \text{VONMISES} \\ \text{MAXS or SHEAR} \end{array} \right], \left[\begin{array}{c} \text{STRCUR} \\ \text{FIBER} \end{array} \right], \right. \right. \\ \left. \left. \begin{array}{c} \left[\begin{array}{c} \text{CENTER} \\ \text{CORNER or BILIN} \\ \text{SGAGE} \\ \text{CUBIC} \end{array} \right], \left[\begin{array}{c} \text{PSDF, ATOC, CRMS} \\ \text{or RALL} \end{array} \right] \left[\begin{array}{c} \text{RPRINT} \\ \text{NORPRINT} \end{array} \right], \left[\begin{array}{c} \text{RPUNCH} \\ \text{RPUNCH} \end{array} \right] \end{array} \right) = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\} \right]$$

Examples:

STRAIN=5
 STRAIN(CORNER)=ALL
 STRAIN(PRINT, PHASE)=15
 STRAIN(PLOT)=ALL
 STRAIN(PRINT, PSDF, CRMS, RPUNCH)=20
 STRAIN(PRINT, RALL, NORPRINT)=ALL

Descriptor	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each element.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

Descriptor	Meaning
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. The request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. The request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
CRMS	Requests the cumulative root mean square function to be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
VONMISES	von Mises strains are output.
MAXS or SHEAR	Maximum shear strains are output.
STRCUR	Strain at the reference plane and curvatures are output for plate elements.
FIBER	Strain at locations Z1, Z2 are computed for plate elements.
CENTER	Output CQUAD4 element strains at the center only.
CORNER or BILIN	Output CQUAD4 element strains at the center and grid points. Using strain gage approach with bilinear extrapolation.
SGAGE	Output CQUAD4 element strains at center and grid points using strain gage approach.
CUBIC	Output CQUAD4 element strains at center and grid points using cubic bending correction.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
ALL	Strain for all elements will be output.
n	Set identification of a previously appearing SET command. Only strain for elements with identification numbers that appear on this SET command will be output (Integer > 0).
NONE	No element strain will be output.

Remarks:

1. Fully accurate nonlinear strains for nonlinear elements are requested by the STRESS command and appear in the nonlinear stress output. In SOLs 106 and 129, if LGDISP > 0 the center, and if requested for CQUAD4 the corner, strains are computed from the displacements alone and are only approximate. If LGDISP = -1 and material nonlinear then no STRAIN output will be created. In SOL 400, if enhanced material options are used, the computed strains are accurate and in this case a CQUAD4 corner request is ignored.
2. In SOLs 106 and 129, the STRAIN request pertains only to linear elements and only if the parameter LGDISP is -1, which is the default. Nonlinear strains for nonlinear elements are requested by the STRESS command and appear in the nonlinear stress output.
3. STRAIN=NONE overrides an overall output request.
4. Definitions of stress, strain, curvature, and output locations are given in the [Structural Elements](#) in the *MSC Nastran Reference Guide*.
5. If the STRCUR option is selected, the values of Z1 will be set to 0.0. and Z2 will be set to -1.0 on the output.
6. The VONMISES, MAXS, and SHEAR options are ignored in the complex eigenvalue and frequency response solution sequences.
7. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase, and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. (In superelement analysis, the first subcase refers to the first subcase of each superelement. Therefore, it is recommended that these options be specified above all subcases.) Consequently, options specified in subcases other than the first subcase will be ignored. See also Remark 8 under the [FORCE \(Case\), 382](#) Case Control command for further discussion.
8. See Remark 1 under the [DISPLACEMENT \(Case\), 298](#) Case Control command for a discussion of SORT1 and SORT2.
9. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
10. Random response for complex strain in composites plates or shells, and layered solid composites and layered solid shell composites will be available for each ply.

STRESS**Element Stress Output Request**

Requests the form and type of element stress output. Note: ELSTRESS is an equivalent command.

Format:

$$\text{STRESS} \left[\begin{array}{c} \left[\begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[\begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[\begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[\begin{array}{c} \text{VONMISES} \\ \text{MAXS or SHEAR} \end{array} \right], \right. \\ \left. \begin{array}{c} \left[\begin{array}{c} \text{CENTER} \\ \text{CUBIC} \\ \text{SGAGE} \\ \text{CORNER or BILIN} \end{array} \right], \left[\begin{array}{c} \text{PSDF, ATOC, CRMS} \\ \text{or RALL} \end{array} \right] \left[\begin{array}{c} \text{RPRINT} \\ \text{NORPRINT} \end{array} \right], \left[\begin{array}{c} \text{RPUNCH} \\ \text{RPUNCH} \end{array} \right] \end{array} \right) = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Examples:

```
STRESS=5
STRESS (CORNER)=ALL
STRESS (SORT1,PRINT,PUNCH,PHASE)=15
STRESS (PLOT)=ALL
STRESS (PRINT, PSDF, CRMS, RPUNCH)=20
STRESS (PRINT, RALL, NORPRINT)=ALL
```

Descriptor	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each element type.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG

Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.

Descriptor	Meaning
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 11.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 11.
CRMS	Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 11.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 11.
VONMISES	Requests von Mises stresses.
MAXS or SHEAR	Requests maximum shear in the plane for shell elements and octahedral stress for solid elements.
CENTER	Requests CQUAD4, CQUADR and CTRIAR element stresses at the center only. The default for CQUAD4 is CENTER. The default for CQUADR and CTRIAR is CORNER.
CUBIC	Requests CQUAD4 element stresses at the center and grid points using strain gage approach with cubic bending correction.
SGAGE	Requests CQUAD4 element stresses at center and grid points using strain gage approach.
CORNER or BILIN	Requests CQUAD4, CQUADR. and CTRIAR element stresses at center and grid points using bilinear extrapolation.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
ALL	Stresses for all elements will be output.
n	Set identification of a previously appearing SET command. Only stresses for elements with identification numbers that appear on this SET command will be output (Integer > 0).
NONE	No element stress will be output.

Remarks:

1. ALL should not be used in a transient problem due to excessive output.

2. See Remark 1 under the [DISPLACEMENT \(Case\), 298](#) Case Control command description for a discussion of SORT1 and SORT2.
3. ELSTRESS is an alternate form and is equivalent to STRESS.
4. STRESS=NONE overrides an overall output request.
5. Prior to MSC Nastran 2014, in nonlinear analysis, the nonlinear stresses will still be printed unless NLSTRESS(PLOT) is specified.
6. The VONMISES option is ignored for ply stresses.
7. The VONMISES, MAXS, and SHEAR options are ignored in the complex eigenvalue and frequency response solution sequences.
8. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase, and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. (In superelement analysis, the first subcase refers to the first subcase of each superelement. Therefore, it is recommended that these options be specified above all subcases.) Consequently, options specified in subcases other than the first subcase will be ignored. See also Remark 8 under the [FORCE \(Case\), 382](#) Case Control command for further discussion.
9. For composite ply output, the grid point option for CQUAD4 elements will be reset to the default option (CENTER).
10. MAXS for shell elements is not an equivalent stress.
11. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
12. Random response for complex stress in composites plates or shells, and layered solid composites and layered solid shell composites will be available for each ply.

STRFIELD**Grid Point Stress Output Request**

Requests the computation of grid point stresses for graphical postprocessing and mesh stress discontinuities.

Format:

$$\text{STRFIELD} = \left\{ \begin{array}{l} \text{ALL} \\ n \end{array} \right\}$$

Examples:

STRFIELD=ALL
STRFIELD=21

Descriptor	Meaning
ALL	Grid point stress requests for all surfaces and volumes defined in the OUTPUT(POST) Section will be saved for postprocessing.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command, and in the OUTPUT(POST) Section, will be included in the grid point stress output request for postprocessing (Integer > 0).

Remarks:

1. The STRFIELD command is required for the graphical display of grid point stresses in postprocessors that use the .xdb file (PARAM,POST,0), or when the GPSDCON or ELSDCON commands are specified, and does not provide printed output. The GPSTRESS command can be used to obtain printed output.
2. Only grid points connected to elements used to define the surface or volume are output. See the SURFACE and VOLUME Case Control commands.
3. Element stress output (STRESS) must be requested for elements referenced on requested SURFACE and VOLUME Case Control commands.
4. In nonlinear static and transient analysis, grid point stresses are computed only if parameter LGDISP is -1, which is the default. Also, in nonlinear transient analysis, grid point stresses are computed only for elements with linear material properties.

SUBCASE

Subcase Delimiter

Delimits and identifies a subcase.

Format:

SUBCASE=n

Example:

SUBCASE=101

Descriptor	Meaning
n	Subcase identification number (9999999 > Integer > 0).

Remarks:

1. The subcase identification number, n, must be greater than all previous subcase identification numbers.
2. Plot requests and RANDPS requests refer to n.
3. See the MODES Case Control command for use of this command in normal modes analysis.
4. If a comment follows n, then the first few characters of the comment will appear in the subcase label in the upper right-hand corner of the output.
5. Note that in nonlinear statics (SOL106/129), SUBCASE's are not stand-alone solutions like in other solution sequences. They act as a load-progression, and the ending conditions of one SUBCASE become the initial conditions of the next SUBCASE.

SUBCOM**Combination Subcase Delimiter**

Delimits and identifies a combination subcase.

Format:

SUBCOM = n

Example:

SUBCOM = 125

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

Remarks:

1. The subcase identification number, n, must be greater than all previous subcase identification numbers.
2. A SUBSEQ command must follow this command.
3. SUBCOM may only be used in SOL 101 (statics) or SOL 144 (static aeroelasticity) and in SOL 200 with ANALYSIS=STATICS or ANALYSIS=SAERO.
4. Output requests above the subcase level will be used.
5. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the SUBCOM with a TEMP(LOAD) command, or the element deformations with a DEFORM command.
6. SUBCOMs may be specified in superelement analysis with the following recommendations:
 - a. For each superelement, specify its SUBCASE(s) consecutively, directly followed by its SUBCOM(s).
 - b. Specify a SUPER command with a new load sequence number under each SUBCOM command.

The following example demonstrates a model with one superelement and one load combination:

```

SUBCASE 101
SUPER=1,1
LOAD=100
SUBCASE 102
SUPER=1,2
LOAD=200
SUBCOM 110
LABEL=COMBINE SUBCASES 101 AND 102
SUPER=1,3
SUBSEQ=1.,1.
SUBCASE 1001
SUBCASE 1002
SUBCOM 1010
LABEL=COMBINE SUBCASES 1001 AND 1002
SUBSEQ=1.,1.
  
```

7. For static aeroelasticity, only the displacement/element responses are combined. Trim, stability derivative and monitor point results are not combined.
8. SUBCOMs are not allowed in external superelement creation runs and are ignored in external superelement data recovery in assembly runs.

SUBSEQ**Subcase Sequence Coefficients**

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

Format:

SUBSEQ=R1 [, R2, R3, ..., Rn]

Example:

SUBSEQ=1.0, -1 .0, 0.0, 2.0

Descriptor	Meaning
Ri	Coefficients of the previously occurring subcases. See Remark 4. (Real).

Remarks:

1. The SUBSEQ command can only appear after a SUBCOM command.
2. SUBSEQ may be only used in SOL 101 (statics) or SOL 144 (static aeroelasticity) and in SOL 200 with ANALYSIS=STATICS or ANALYSIS=SAERO.
3. This command list is limited to a maximum of 200 numbers.
4. R1 to Rn refer to the immediately preceding subcases. In other words, Rn is applied to the most recently appearing subcase, R(n - 1) is applied to the second most recently appearing subcase, and so on. The embedded comments (\$) describe the following example:

```
DISPL = ALL
SUBCASE 1
SUBCASE 2
SUBCOM 3
SUBSEQ = 1.0, -1.0 $ SUBCASE 1 - SUBCASE 2
SUBCASE 11
SUBCASE 12
SUBCOM 13
SUBSEQ = 0.0, 0.0, 1.0, -1 .0 $ SUBCASE 11 - SUBCASE 12
or
SUBSEQ = 1.0, - 1.0 $ EQUIVALENT TO PRECEDING COMMAND. USE ONLY
ONE.
```

SUBSEQ1

Subcase Factors for Combination

Gives the factors for linear combination of a specific group of SUBCASEs.

Format:

SUBSEQ1= s0, s1, sub1, s2, sub2, [sn, subn]

Example:

SUBSEQ1= 1.0, 1.0, 101, -1.0, 102

Descriptor	Meaning
S0	Factor for all SUBCASEs involved. (No default; Real<>0.).
Sn	Factor applicable to SUBn only (Real; No default).
SUBn	SUBCASE ID (No default; Integer>0).

Remarks:

1. The SUBSEQ1 command can only appear after a SUBCOM command.
2. SUBSEQ1 may only be used in SOL 101 (Statics) or SOL 144 (Static Aeroelasticity) and in SOL 200 with ANALYSIS=STATIC or ANALYSIS=SAERO.
3. SUBSEQ1 and SUBSEQ are mutually exclusive and can't both appear under a SUBCOM.
4. S0, S1 and SUB1are required input for SUBSEQ1. S2,SUB2 to Sn,SUBn pair are optional.

SUBSTEP**SUBSTEP Delimiter**

Delimits and identifies a nonlinear analysis SUBSTEP for COUPLED analysis in SOL 400.

Format:

SUBSTEP=n

Examples:

STEP=50

NLSTEP=10

SUBSTEP=1

ANALYSIS=HSTAT

SUBSTEP=2

ANALYSIS=NLSSTAT

Descriptor	Meaning
n	Substep identification number. (Integer > 0)

Remarks:

1. The SUBSTEP command can only be used in nonlinear solution sequence SOL 400 (NONLIN).
 2. The SUBSTEP command can only be used in a STEP command.
 3. When used in a STEP command two or more SUBSTEP commands must occur.
 4. Each SUBSTEP must contain a *unique* ANALYSIS-type statement. Currently there is a limitation of only two SUBSTEPS per STEP with the following options:
 - ANALYSIS=HSTAT for the first SUBSTEP and ANALYSIS=NLSSTAT for the second SUBSTEP.
 - ANALYSIS=HTRAN for the first SUBSTEP and ANALYSIS=NLTTRAN for the second SUBSTEP
 - ANALYSIS=HTRAN for the first SUBSTEP and ANALYSIS=NLSSTAT for the second SUBSTEP
 - ANALYSIS=HSTAT for the first SUBSTEP and ANALYSIS=NLTTRAN for the second SUBSTEP
 5. Within a STEP the SUBSTEP identification number n must be in increasing order and not greater than 9999999.
 6. The following example illustrates a typical application of SUBCASE, STEP, and SUBSTEPS
- Subcase 100

```
STEP 10
STRESS= ALL
NLSTRESS=ALL
SET 1456 = list
NLSTEP=84
SubSTEP 1
```

```

ANALYSIS=HSTAT
THERMAL=ALL
FLUX=ALL
SPC=35
LOAD=11
SubSTEP 2
ANALYSIS=NLSTAT
SPC=2
LOAD=110
DISP(PLOT)=1456

```

7. The solutions of all SUBCASEs are independent of each other. The solution of any STEP is a continuation of the solution of the previous STEP. *The solutions of the SUBSTEPS occur simultaneously within a STEP.*
8. In coupled analysis, TEMP(LOAD)=m will be ignored as the temperature loading from the thermal substep is automatically transferred to the mechanical substep. Note that in this multi-physics coupled framework, both the heat transfer and structural physics are executed on the same mesh with the same time steps.
9. In coupled analysis, TEMP(INIT)=n will be honored. It should be noted that when a thermo-mechanical static analysis is defined, TEMP(INIT) defines the initial temperature for both the thermal and mechanical substeps. Note that when a transient thermal-static mechanical analysis is define, IC=n should be used to define the initial temperature for the thermal substep and TEMP(INIT)=n (the same ID n) should be used to define the initial temperature for the structural substep. Then an initial thermal strain is defined as:

$$\varepsilon T = A(T) \cdot (T - TREF) - A(T_o) \cdot (T_o - TREF)$$

where T_o comes from TEMP(INIT) and TREF comes from the material entry.

10. The LOADSET Case Control Command *is not allowed* with this command. All dynamic loading must be applied through the use of the DLOAD Case Control command.
11. For coupled analysis, i.e., when two or more sub-steps are defined, a *single* Bulk Data entry NLSTEP must be included in the STEP command containing the SUBSTEP commands. It must occur above or in the STEP command and above the first SUBSTEP command.

The loads and constraints can be independently defined for each physics under the relevant SUBSTEP. The commands that are normally used for single physics are applicable. For, e.g., for a HTRAN-NLSTAT coupled analysis, DLOAD should be used n the HTRAN substep while LOAD should be used in the NLSTAT substep.

12. Any case control command will apply to all SUBSTEPS when and only when it is above all the SUBSTEPS of the STEP.

SUBTITLE

Output Subtitle

Defines a subtitle that will appear on the second heading line of each page of printer output.

Format:

SUBTITLE=subtitle

Example:

SUBTITLE=PROBLEM NO. 5-1A

Descriptor	Meaning
subtitle	Any character string.

Remarks:

1. SUBTITLE appearing under a SUBCASE command will appear in the output for that subcase only.
2. SUBTITLE appearing before all SUBCASE commands will appear in the output for all subcases except those in Remark 1.
3. If no SUBTITLE command is present, the subtitle line will be blank.
4. The subtitle also appears on plotter output.

SUPER**Superelement Subcase Assignment**

Assigns a subcase(s) to a superelement or set of superelements.

Format:

$$SUPER = \left\{ \begin{array}{c} ALL \\ \left\{ \begin{array}{c} n \\ i \end{array} \right\} [, l] \end{array} \right\}$$

Examples:

SUPER=17, 3

SUPER=15

SUPER=ALL

Descriptor	Meaning
i	Superelement identification number (Integer > 0).
ALL	The subcase is assigned to all superelements and all loading conditions (Default).
n	Set identification number of a previously appearing SET command. The subcase is assigned to all superelements with identification numbers that appear on this SET command (Integer > 0).
l	Load sequence number (Integer > 0; Default=1).

Remarks:

1. All subcases with requests for specific superelement(s) must contain the SUPER command. If no SUPER command is specified in the Case Control Section, then all subcases will be assigned to all superelements; i.e., SUPER=ALL is the default.
2. All subcases associated with superelements must precede those for the residual structure except when SUPER=ALL or SUPER=n and the selected set includes the residual structure.
3. The load sequence number is only used in static analysis and frequency response analysis when there are multiple loading conditions. Also, the residual structure must have a subcase specified for each unique load condition. This is required because the number of residual structure subcases is used to determine the number of load conditions for all superelements.
4. The load sequence number is associated with the order of the subcases for the residual structure; i.e., the third loading condition is associated with the third subcase for the residual structure.
5. Subcases are required for superelements when there is a load, constraint, or output request.
6. If a set is referenced by n, then the SET identification number must be unique with respect to any superelement identification numbers. In addition, the same sets must be used for all loading conditions.

7. If the ALL option is used, it must be used for all loading conditions.
8. If there are no superelements in the model then the SUPER command will be ignored except in 3-step external superelement data recovery restarts in SOL 400. See Remarks 9 and 10 under the EXTDRIN Case Control command description.

SUPPORT1

Fictitious Support Set Selection

Selects the fictitious support set (SUPPORT1 or SUPPORT6 entries only) to be applied to the model.

Format:

SUPPORT1=n

Examples:

SUPPORT1=15

SUP0=4

Descriptor	Meaning
n	Set identification of fictitious support set defined on the SUPPORT1 or SUPPORT6 Bulk Data entries (Integer > 0).

Remarks:

1. SUPPORT1 or SUPPORT6 Bulk Data entries will not be used unless selected in the Case Control Section by the SUPPORT1 command.
2. SUPPORT entries will be applied in all subcases.
3. For SOL 600, Case Control command SUPPORT1 must reference a SUPPORT6 Bulk Data entry with ID = N.

SVECTOR**Solution Set Eigenvector Output Request**

Requests the form and type of solution set eigenvector output.

Format:

$$SVECTOR(\begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix}) = \left\{ \begin{array}{c} ALL \\ n \\ NONE \end{array} \right\}$$

Examples:

SVECTOR=ALL

SVECTOR (PUNCH) =NONE

Descriptor	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Displacements for all points (modes) will be output.
NONE	Displacements for no points (modes) will be output.
n	Set identification of a previously appearing SET command. Only displacements of points with identification numbers that appear on this SET command will be output (Integer > 0).

Remarks:

1. SVECTOR=NONE overrides an overall output request.
2. Output will be presented as a tabular listing of grid points for each eigenvector.

SVELOCITY**Solution Set Velocity Output Request**

Requests the form and type of solution set velocity output.

Format:

$$\text{SVELOCITY}\left[\left(\begin{matrix} \text{SORT1} \\ \text{SORT2} \end{matrix}\right), \text{PRINT}, \text{PUNCH}, \left[\begin{matrix} \text{REAL or IMAG} \\ \text{PHASE} \end{matrix}\right]\right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Examples:

SVELOCITY=5

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

Descriptor	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.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number).
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
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 with identification numbers that appear on this SET command will be output (Integer > 0).

Remarks:

1. Velocity output is only available for transient and frequency response problems.
2. The defaults for SORT1 and SORT2 depend on the type of analysis, and is discussed in Remark 1 under the [DISPLACEMENT \(Case\), 298](#) Case Control command. If SORT1 is selected for any of the commands SACC, SDIS, and SVEL, then the remaining commands will also be SORT1.
3. SVELOCITY=NONE overrides an overall output request.

SYM Symmetry Subcase Delimiter

Delimits and identifies a symmetry subcase.

Format:

SYM=n

Example:

SYM=123

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

Remarks:

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

SYMCOM

Symmetry Combination Subcase Delimiter

Delimits and identifies a symmetry combination subcase.

Format:

SYMCOM=n

Example:

SYMCOM=123

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

Remarks:

1. The subcase identification number n must be greater than all previous subcase identification numbers.
2. SYMCOM may only be used in statics problems.
3. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the SYMCOM by use of a TEMP(LOAD) command, or the element deformations by a DEFORM command.
4. An alternate command is the SUBCOM command.
5. SYMCOMs may be specified in superelement analysis with the following recommendations:
 - a. For each superelement, specify its SUBCASEs consecutively, directly followed by its SYMCOM(s).
 - b. Specify a SUPER command with a new load sequence number under each SYMCOM command.

The following example represents a model with one superelement and one load combination:

```
SUBCASE 101
SUPER=1,1
LOAD=100
SUBCASE 102
SUPER=1,2
LOAD=200
SYMCOM 110
LABEL=COMBINE SUBCASES 101 AND 102
SUPER=1,3
SYMSEQ=1.,1.
SUBCASE 1001
SUBCASE 1002
SYMCOM 1010
LABEL=COMBINE SUBCASES 1001 AND 1002
SYMSEQ=1.,1.
```

SYMSEQ**Symmetry Sequence Coefficients**

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

Format:

SYMSEQ=R1 [,R2,R3,..., Rn]

Example:

SYMSEQ=1.0, -2.0, 3.0, 4.0

Descriptor	Meaning
Ri	Coefficients of the previously occurring n SYM subcases. (Real)

Remarks:

1. SYMSEQ may only appear after a SYMCOM command.
2. The default value for the coefficients is 1.0 if no SYMSEQ command appears.
3. SYMSEQ may only be used in static analysis or inertia relief.
4. Ri is limited to a maximum of 200 numbers.

TACCELERATION

Acceleration Output Request for trim components, TRMC

Requests the form and type of acceleration output for TRMC.

Format:

$$TACCELERATION \left[\begin{bmatrix} SORT1 \\ SORT2 \end{bmatrix}, \begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix} \begin{bmatrix} REAL \text{ or } IMAG \\ PHASE \end{bmatrix} \right] = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

Example:

```
SET 20 = 3/103, 5/0, 12/ALL $
TACCELERATION=ALL
TACC (REAL, PUNCH, PRINT)=20
TACCE=20
TACCELE (SORT2, PRINT)=20
```

Descriptor	Meaning		
SORT1	Output will be presented as a tabular listing of grid points for each load and frequency.		
SORT2	Output will be presented as a tabular listing of frequency for each grid point.		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

- | | |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| PHASE | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees. |
| ALL | TRMC acceleration for all points of all TRMCs will be output. See Remarks 2. |

Descriptor	Meaning
NONE	No TRMC acceleration will be output.
n	Set identification of a previously appearing SET command. Only points with identification numbers that appear on this SET command will be output (Integer > 0). SET 20 in above examples section is to request SET 103(not present) for TRMC 3, none for TRMC 5 and ALL for TRMC 12.

Remarks:

1. Refrain from using ALL which may produce voluminous output.
2. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.
3. TDISP=NONE suppresses the generation of TRMC acceleration output.
4. TACCELERATION is supported only with PARAM,TRMBIM,PHYSICAL
5. TACCELERATION is available for PEM jobs of SOL 108, 111 and SOL 200 with ANALYSIS=DFREQ and MFREQ. Note that TACCELERATION is available for output only and cannot be utilized as design response.
6. For restart PEM job, TACCE request must remain the same as cold start PEM job.

TDISPLACEMENT

Displacement Output Request for trim components, TRMC

Requests the form and type of displacement output for TRMC.

Format:

$$\text{TDISPLACEMENT} \left[\begin{bmatrix} \text{SORT1} \\ \text{SORT2} \end{bmatrix}, \begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix} \begin{bmatrix} \text{REAL or IMAG} \\ \text{PHASE} \end{bmatrix} \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Example:

```
SET 20 = 3/103, 5/0, 12/ALL $
TDISPLACEMENT=ALL
TDISP=(REAL, PUNCH, PRINT)=20
TDISP=20
TDISPLACE(SORT2, PRINT)=20
```

Descriptor	Meaning		
SORT1	Output will be presented as a tabular listing of grid points for each load and frequency.		
SORT2	Output will be presented as a tabular listing of frequency for each grid point.		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

- | | |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| PHASE | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees. |
| ALL | TRMC displacement for all points of all TRMCs will be output. See Remarks 2. |

Descriptor	Meaning
NONE	No TRMC displacement will be output.
n	Set identification of a previously appearing SET command. Only points with identification numbers that appear on this SET command will be output (Integer > 0). SET 20 in above examples section is to request SET 103(not present) for TRMC 3, none for TRMC 5 and ALL for TRMC 12.

Remarks:

1. Refrain from using ALL which may produce voluminous output.
2. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.
3. TDISP=NONE suppresses the generation of TRMC acceleration output.
4. TDISPLACEMENT is supported only with PARAM,TRMBIM,PHYSICAL
5. TDISPLACEMENT is available for PEM jobs of SOL 108, 111 and SOL 200 with ANALYSIS=DFREQ and MFREQ. Note that TDISPLACEMENT is available for output only and cannot be utilized as design response.
6. For each grid in a TRMC, it has 3 or 6 DOFs for solid phase and 1 DOF for fluid phase. TDISPLAMENT output results for both solid phase DOFs and fluid phase DOF.
7. The 3 DOFs of solid phase takes T1, T2 and T3 locations of a regular 6 DOFs structural grid and set R1, R2 and R3 locations to 0.0. The 1 DOF of fluid phase is reported as acoustic pressure.
8. For restart PEM job, TDISP request must remain the same as cold start PEM job.

TEMPERATURE**Temperature Set Selection**

Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

Format:

$$\text{TEMPERATURE} \left[\begin{array}{c} \text{INITIAL} \\ \text{MATERIAL} \\ \text{LOAD} \\ \text{BOTH} \end{array} \right] \left[, \text{HSUBCASE} = i \left[, \text{HSTEP} = j \left[, \text{HTIME} = \left[\begin{array}{c} t \\ \text{ALL} \end{array} \right] \right] \right] \right], \text{VERIFY} = \left[\begin{array}{c} \text{GRID} \\ \text{ELEMENT} \\ \text{BOTH} \\ \text{NONE} \end{array} \right]$$

Examples:

`TEMPERATURE (LOAD) =15`

`TEMPERATURE (MATERIAL) =7`

`TEMPERATURE=7`

`TEMPERATURE (LOAD, HSUBCASE=20, HTIME=12.0) =39`

Descriptor	Meaning
MATERIAL	The selected temperature set will be used to determine temperature-dependent material properties indicated on MATTi Bulk Data entries. See Remarks 6., 7., and 8.
LOAD	The selected temperature set will be used to determine an equivalent static load and to update material properties in a nonlinear analysis. See Remarks 2., 5., 6., 7. and 14.
BOTH	Both MATERIAL and LOAD will use the same temperature set.
n	Set identification number of TEMP, TEMPD, TEMPP1,TEMPB3, TEMPRB, TEMPF, or TEMPAX Bulk Data entries (Integer > 0).
INITIAL	The selected temperature table will be used to determine initial temperature distribution in nonlinear static analysis. See Remarks 4., 6., 7., 8., 9., and 12.
HSUBCASE	Specifies a SUBCASE executed in the selected thermal job. See Remark 14.
i	Identification number of a SUBCASE executed in the selected thermal job. (Integer ≥ 0 , Default = 0 is the first SUBCASE) See Remark 14.
HSTEP	Specifies a STEP executed in the selected thermal job. See Remark 14.
j	Identification number of a STEP executed in the selected thermal job. (Integer ≥ 0 , Default = 0 is the first STEP) See Remark 14.
HTIME	Specifies the time of a time step executed in the selected nonlinear transient thermal job. See Remark 14.
t	Time of a time step executed in the selected nonlinear transient thermal job. (Real ≥ 0.0 , Default is the last time of the specified SUBCASE and/or STEP) See Remark 14.

Descriptor	Meaning
ALL	Selects all time steps executed in the selected nonlinear transient thermal job. See Remark 14.
VERIFY	Output temperature verification data output requested
GRID	Output grid temperature verification data
ELEMENT	Output element temperature verification data
BOTH	Output both grid and element temperature verification data
NONE	Do not output any verification data

Remarks:

1. In linear analysis, only one TEMP(MATE) may be made in any problem and should be specified above the subcase level. If there are multiple subcase and TEMP(MATE) is not above the first subcase, then it must appear in the last subcase or it will be ignored. See also Remarks 6. and 7.
2. The total load applied will be the sum of external (LOAD command), thermal (TEMP(LOAD) command), element deformation (DEFORM command), and constrained displacement (SPC command) loads.
3. Static, thermal, and element deformation loads should have unique set identification numbers.
4. INITIAL is used in steady state heat transfer analysis for conduction material properties, and provides starting values for iteration. In structural analysis, TEMP(INIT) is used to specify an initial temperature. It may appear above or in the first subcase. In SOL 400, it may appear above or in the first STEP.
5. In superelement data recovery restarts, TEMPERATURE(LOAD) requests must be respecified in the Case Control Section.
6. In linear static analysis, temperature strains are calculated by

$$\epsilon_T = A(T_o) \cdot (T - T_o)$$

where $A(T_o)$ is the thermal expansion coefficient defined on the MATi Bulk Data entries, T is the load temperature defined with TEMPERATURE(LOAD), and T_o is the initial temperature defined as follows. The following rules apply for TEMPERATURE(INITIAL), TEMPERATURE(MATERIAL), and TREF on the MATi entries:

- a. If TEMPERATURE(INITIAL) and TREF are specified, then the TEMPERATURE(INITIAL) set will be used as the initial temperature to calculate both the loads and the material properties.
- b. If TEMPERATURE(MATERIAL) and TREF are specified, then TREF will be used as the initial temperature in calculating the load and the TEMPERATURE(MATERIAL) set will be used for the calculation of material properties.
- c. If neither TEMPERATURE(INITIAL), TEMPERATURE(MATERIAL), nor TEMPERATURE(BOTH) is present, TREF will be used to calculate both the load and the material properties and will be obtained from the MATi entry. The MATTi is not used in this case.

7. In nonlinear static analysis, temperature strains are calculated with

$$\epsilon_T = A(T) \cdot (T - TREF) - A(T_o) \cdot (T_o - TREF)$$

where $A(T)$ is the thermal expansion coefficient defined on the MATi Bulk Data entries, T is the load temperature defined with TEMPERATURE(LOAD), and T_o is the initial temperature defined with TEMPERATURE(INITIAL). The following rules apply:

- a. The specification of TEMPERATURE(MATERIAL) or TEMPERATURE(BOTH) will cause a fatal error.
- b. If a subcase does not contain a TEMPERATURE(LOAD) request, then the thermal load set will default to the TEMPERATURE(INITIAL) set.
- c. TEMPERATURE(LOAD) will also cause the update of temperature-dependent material properties due to the temperatures selected in the thermal load set. Temperature-dependent material properties are specified with MATi, MATTi, MATS1, and/or TABLEST Bulk Data entries.
- d. If TREF and TEMPERATURE(INITIAL) are specified, then the TEMPERATURE(INITIAL) set will be used as the initial temperature to calculate both the loads and the material properties. Both are used in the definition of thermal strain.

For SOL 600, TREF and TEMP(INIT) must be consistent (the same values) or unexpected results may occur.

8. TEMPERATURE(MATERIAL) and TEMPERATURE(INITIAL) cannot be specified simultaneously in the same run.
9. TEMP(INIT) is not used with TEMPAX.
10. Temperature loads cause incorrect element forces and stresses in all the dynamic analysis except SOL 400 nonlinear transient analysis with the elements having nonlinear capability.

It should be noted that:

- a. For nonlinear dynamics analysis with SOL 400, TEMPERATURE/THERMAL load can't be applied by TEMP/DLOAD (Case Control) and TEMPD/TLOADi (Bulk Data), this load should be applied by TEMP(Case Control) and TTEMP(Bulk Data);
- b. For pure linear analysis with SOL 400, the TEMP/DLOAD (Case Control) and TEMPD/TLOADi (Bulk Data) has to be used to apply the temperature load.
11. In linear analysis, TEMPERATURE(MATERIAL) is not supported for hyperelastic elements (MATHP). TEMP(INIT) must be placed above the subcase level, and TEMP(LOAD) placed within the subcase.
12. For layered composites, neither the TREF specified on the material entries, nor TEMP(INIT) nor TEMP(MATE) are used to determine ply reference temperature. The TREF on the PCOMP or PCOMPG entries is used for all plies of the element. This is true for both linear and nonlinear analysis. If TEMP(INIT) is defined, the TREF on the PCOMP or PCOMPG entries is only used for determining the material properties and TEMP(INIT) is used for thermal strains. If TEMP(INIT) is not defined, the TREF on the PCOMP or PCOMPG entries is used for determining the material properties and thermal strains.

13. For SOL 600, in a thermal stress analysis where the temperatures were produced in a previous SOL 600 simulation, the use of MCHSTAT is preferred.
14. For TEMPERATURE(LOAD) requests in SOL 400, HSUBCASE, HSTEP, and HTIME are used to retrieve the temperature results from an existing thermal database. This feature allows user to select either steady state or transient thermal results for nonlinear structural analysis, with the flexibility of different time steps and dissimilar mesh sizes between thermal and structural runs. The following rules apply for using this capability:
 - HSUBCASE, HSTEP, and HTIME keywords must follow a LOAD keyword.
 - Although all three keywords have default values, at least one keyword must exist to apply this uncoupled multi-physics feature in analysis.
 - HTIME=ALL is used in nonlinear transient structural analysis to perform real time temperature interpolations. In this case, the nodal temperatures of nonlinear elements are updated at each time step. These temperatures are equal to the temperature results of the selected thermal database at current time.
 - The set IDs of TEMP(LOAD) must be different from the set IDs of TEMP(INIT).
 - To save temperature results of thermal analysis in MASTER nastran database, use the following command.

nastran thermal_job_name scratch=no

In addition, the user must specify the following Bulk Data entry:

PARAM,NLPACK,-1

for transient thermal models with number of time steps greater than the default NLPACK output time steps.

- The following File Management Statements are required in the current structural model to select the thermal database.

ASSIGN hrun='thermal_job_name.MASTER'

DBLOC DATABLK=(HEATDB) LOGI=hrun

- Contact body integrity is not maintained while doing the thermal mapping. While this would not be a restriction for bodies that are not physically in contact or in glued thermal contact, it could lead to undesirable temperature mapping for bodies that are in regular thermal contact with temperature gradients across the contact interface.

TERMIN

Conditions to Terminate in SOL 600

Selects a TERMIN Bulk Data entry which specifies criteria such that a SOL 600 analysis can be terminated, for example, if the displacement at a certain grid exceeds a specified value.

Format:

TERMIN=N

Example:

TERMIN=5

Descriptor	Meaning
N	ID of a matching TERMIN Bulk Data entry specifying the termination conditions for a particular analysis.

Remarks:

1. This entry may only be used within subcases (it may not be placed above the first subcase entry). If there are no subcase entries, it may be placed anywhere in the Case Control Section.
2. Most SOL 600 analyses do not require TERMIN entries.
3. If some subcases have TERMIN entries and others do not, only those that do will check for termination conditions.
4. TERMIN criteria may be different for different subcases.

TFL**Transfer Function Set Selection**

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

Format:

TFL=n

Example:

TFL=77

TFL = 1, 25, 77

Descriptor	Meaning
n	Set identification of a TF Bulk Data entry (Integer > 0).

Remarks:

1. Transfer functions will not be used unless selected in the Case Control Section.
2. Transfer functions are supported in dynamics problems only.
3. Transfer functions are described in the [MSC Nastran Dynamic Analysis User's Guide](#).
4. It is recommended that PARAM,AUTOSPC,NO be specified when using transfer functions. See [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide*.
5. The transfer functions are additive if multiple TF values are referenced on the TFL command.

THERMAL**Temperature Output Request**

Requests the form and type of temperature output.

Format:

$$\text{THERMAL} \left[\left(\begin{bmatrix} \text{SORT1} \\ \text{SORT2} \end{bmatrix}, \begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix} \right) \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Examples:

THERMAL=5

THER (PRINT, PUNCH)=ALL

Descriptor	Meaning
SORT1	Output is presented as a tabular listing of point temperatures for each load or time step.
SORT2	Output is presented as a tabular listing of loads or time steps for each.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL Temperatures for all points will be output.

NONE Temperatures for no points will be output.

n Set identification of a previously appearing SET command. Only temperatures of points with identification numbers that appear on this SET command will be output (Integer > 0).

Remarks:

1. The THERMAL output request is designed for use with the heat transfer option. The printed output will have temperature headings. The PUNCH option produces TEMP Bulk Data entries, and the SID on the entries will be the subcase number (=1 if no SUBCASES are specified).

2. SORT1 is the default in steady state heat transfer analysis. SORT2 is the default in transient heat transfer analysis.
3. In a transient heat transfer analysis, the SID on the punched TEMP Bulk Data entries equal the time step number.

TITLE Output Title

Defines a character string to appear on the first heading line of each page of MSC Nastran printer output.

Format:

TITLE=title

Example:

TITLE=RIGHT WING, LOAD CASE 3.

Descriptor	Meaning
title	Any character string.

Remarks:

1. If this command appears under a SUBCASE command, then the title appears in the output for that subcase only.
2. If this command appears before all SUBCASE commands, then the title is used in all subcases without a TITLE command.
3. If no TITLE command is present, then the title line will contain data and page numbers only.
4. The title also appears on plotter output.

TRIM**Aerodynamic Trim Variable Constraint Selection**

Selects trim variable constraints in static aeroelastic response.

Format:

TRIM = n

Example:

TRIM=1

Descriptor	Meaning
n	Set identification number of a TRIM Bulk Data entry (Integer > 0).

Remark:

1. Aerodynamic extra points (trim variables) not constrained by a TRIM Bulk Data entry will be free during the static aeroelastic response solution.

TRIMF

Trim Load Output Specification

Specifies options for the output of trim loads from a static aeroelastic analysis as FORCE/MOMENT Bulk Data entries.

Format:

TRIMF[([**UNIT** = *i*], [**LOADSET** = *n*], [**LARGE**], [**INERTIA**], [**APPLIED**], [**AIR**]

[**NOSUM**][**RIGID**],[**NOELASTIC**], [**QNORM**])= $\left\{ \begin{array}{l} \text{ALL} \\ n \end{array} \right\}$

Example:

TRIMF (LOADSET=10001, LARGE) =ALL
TRIMF (UNIT=59, INERTIA, NOSUM) =1

Descriptor	Meaning
UNIT	Fortran unit to which data are written. (Optional; Default = 7) (punch file).
LOADSET	Load set id for output bulk data entries. If the TRIMF specification results in multiple load sets, then the defined ID will be used for the first and each subsequent load set has an ID incremented by 1. (Optional; Default = 1)
LARGE	Write the output data in large field format (16 characters per field). The default is 8 characters per field.
INERTIA	Write out inertial loads as a separate load set. By default, the separate load set will not be written.
APPLIED	Write out applied loads as a separate load set. By default, the separate load set will not be written.
AIR	Write out aerodynamic loads as a separate load set. By default, the separate load set will not be written.
NOSUM	By default, the sum of the inertial, applied, and aerodynamic loads will be written as a separate load set. This option suppresses the writing of that set of loads.
RIGID	Write out rigid instances of the selected loads (Inertial, Applied, Air and/or Sum) as separate load sets. By default, the separate load set will not be written.
NOELASTIC	By default, the sum of the rigid and elastic increment loads will be written as a separate load set. This option suppresses the writing of that set of loads.
QNORM	Normalize the load by the dynamic pressure used in the trim analysis. By default, the loads are not normalized.
ALL	Loads for all points will be output
n	Set identification of a previously appearing SET command. Only loads on points with identification numbers that appear on this SET command will be output (Integer > 0)

Remark:

1. By default, the loads are written to the punch file (Fortran unit 7). If the user specifies an alternate Fortran unit number on the TRIMF entry, by default the loads will be written to a file name that is machine specific (i.e. 'fort.53' on many LINUX platforms). The user may connect the Fortran unit to a user-defined file name by using an ASSIGN entry in the FMS Section of the input file. For example:

```
ASSIGN USERFILE='load13.inc', STATUS=UNKNOWN, FORMATTED, UNIT=53
```

2. Up to eight loads sets are available: Rigid Inertial, Rigid Applied, Rigid Air, Rigid Sum and four more with the sum of the rigid and elastic increment. This table indicates how the describers invoke each of these sets:

	INERTIAL	APPLIED	AIR	SUM
RIGID	O	O	O	O
ELASTIC	O	O	O	DEFAULT

where O = Optional. For a load to appear, both the row and column in the table above have to be set.

3. Care must be taken if LOADSET is specified in a run with multiple subcases. There are no checks that the load set IDs which are generated by one subcase are not also used for another subcase. For example, consider the following Case Control commands:

```
SUBCASE 1
  TRIM = 1
  TRIMF(RIGID) = ALL $
SUBCASE 2
  TRIM = 2
  TRIMF(LOADSET=2) = ALL
```

Subcase 1 will generate two load sets with set IDs 1 and 2. Subcase 2 will also output a load set ID 2.

4. The LOADSET option should not be specified above the subcase level when there are multiple subcases. If it is, each subcase will start numbering its load ID's from LOADSET.

TRIMGRP**Selection of Trim Component(s)**

Selects a set of trim components for analysis.

Format:

$$TRIMGRP = \begin{bmatrix} sid \\ ALL \\ NONE \end{bmatrix}$$

Example:

TRIMGRP=101

Descriptor	Meaning
sid	Set identification of a previously appearing SET command or TRIMID. (Integer>0).
ALL	All trim components present in the model are included in the analysis.
NONE	This is the default of TRIMGRP. No trim component is involved in the analysis.

Remark:

1. TRIMGRP is used to select a group of trim components for analysis, including the calculation of trim component boundary coupling matrices.:
2. PEM capability is available in SOL 108, SOL 111 and SOL 200 with analysis=mfreq.
3. SUBCASEs without TRIMGRP will have the equivalent effect of TRIMGRP=none even with the presence of 'BEGIN TRMC' in the deck.
4. To run PEM job with multiple processors, additional information is available in 'Running PEM jobs' section of [Using PEM Functions in MSC Nastran](#) in the *MSC Nastran Installation and Operations Guide*.
5. For restart PEM job, TRIMGRP must remain the same as cold start PEM job and no changes to any TRMC should be observed. Otherwise, the PEM job should be run as cold start.
6. Collapsed logic is introduced to improve performance of large PEM jobs satisfy following requirements:
 - a. Same TRIMGRP for all SUBCASEs and
 - b. same master frequencies for all TRMCs. The activation of collapsed logic is automatic if requirements are met. With collapsed logic activated, restart job is restricted to the same TRIMGRP as cold start.

TSTEP**Transient Time Step Set Selection**

Selects integration and output time steps for linear or nonlinear transient analysis.

Format:

TSTEP=n

Example:

TSTEP=731

Describer	Meaning
n	Set identification number of a TSTEP or Bulk Data entry (Integer > 0).

Remarks:

1. A TSTEP entry must be selected to execute a linear transient analysis (SOLs 109 or 112) and for a nonlinear transient analysis (SOLs 129 and 159).
2. A entry must be selected in each subcase to execute a nonlinear transient problem.
3. For the application of time-dependent loads in modal frequency response analysis (SOLs 111 and 146), the TSTEP entry must be selected by the TSTEP command. The time-dependent loads will be recomputed in frequency domain by a Fourier transform.
4. In one subcase or STEP for SOL 400, users should only specify one of TSTEP, TSTEPNL or NLSTEP.

TSTEPNL

Transient Time Step Set Selection for Nonlinear Analysis

See the description of the [TSTEP \(Case\), 629](#). (The TSTEP Case Control entry can be used to select a TSTEPNL Bulk Data entry, however, a Bulk Data TSTEP is completely different from a Bulk Data Entry TSTEPNL.)

TSTRU

Temperature Set ID for a Structures Run

Defines a temperature set ID for a structures run based on a heat transfer subcase.

Format:

TSTRU=n

Example:

TSTRU=999

Describer	Meaning
n	Set identification for use on TEMP(LOAD)=n or TEMP(INIT)=n

Remarks:

1. TSTRU should be placed in a heat transfer subcase.
2. If TSTRU does not explicitly appear in the heat transfer subcase, it is defaulted to TSTRU=heat transfer subcase ID.
3. In a structures run, a temperature set generated from a heat transfer run will override an existing temperature set with identical set ID defined with TEMP, TEMPD, TEMPF, TEMPP1, TEMPRB, or any combination.
4. TSTRU may be placed in the first subcase of a PARAM,HEATSTAT,YES run.
5. TSTRUs may be placed in each subcase of an APPHEAT run. The associated structural analysis then requires the following:

```
ASSIGN heat_run='heat transfer job name.MASTER'
DBLOC DATABLK=(UG,EST,BGPDTS,CASECCR/CASEHEAT) LOGICAL=heat_run
```

6. Heat transfer runs and structural runs must have the same mesh. P-elements should also have the same geometry description. P-order between runs can be different.
7. For nonlinear heat transfer SOL 106 or SOL 153, the INOUT field on the NLPARM Bulk Data entry must be blank or NO if the results of the run are to be transferred to a linear structures run.

```
PARAM,NLHTLS,-1
```

The above parameter should be placed in the nonlinear heat run. This will place UG heat transfer on the database.

TVELOCITY

Velocity Output Request for trim components, TRMC

Requests the form and type of velocity output for TRMC.

Format:

$$TVELOCITY \left[\begin{pmatrix} SORT1 \\ SORT2 \end{pmatrix}, \begin{pmatrix} PRINT, PUNCH \\ PLOT \end{pmatrix} \begin{pmatrix} REAL \text{ or } IMAG \\ PHASE \end{pmatrix} \right] = \left\{ \begin{array}{l} ALL \\ n \\ NONE \end{array} \right\}$$

Example:

```
SET 20 = 3/103, 5/0, 12/ALL $
TVELOCITY=ALL
TVEL(REAL, PUNCH, PRINT)=20
TVELO=20
TVELOCI(SORT2, PRINT)=20
```

Descriptor	Meaning		
SORT1	Output will be presented as a tabular listing of grid points for each load and frequency.		
SORT2	Output will be presented as a tabular listing of frequency for each grid point.		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

- | | |
|--------------|--|
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| PHASE | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees. |
| ALL | TRMC velocity for all points of all TRMCs will be output. See Remarks 2. |

Descriptor	Meaning
NONE	No TRMC velocity will be output.
n	Set identification of a previously appearing SET command. Only points with identification numbers that appear on this SET command will be output (Integer > 0). SET 20 in above examples section is to request SET 103(not present) for TRMC 3, none for TRMC 5 and ALL for TRMC 12.

Remarks:

1. Refrain from using ALL which may produce voluminous output.
2. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.
3. TVELO=NONE suppresses the generation of TRMC velocity output.
4. TVELOCITY is supported only with PARAM,TRMBIM,PHYSICAL
5. TVELOCITY is available for PEM jobs of SOL 108, 111 and SOL 200 with ANALYSIS=DFREQ and MFREQ. Note that TVELOCITY is available for output only and can not be utilized as design response.
6. For restart PEM job, TVELO request must remain the same as cold start PEM job.

UNGLUE

Contact Body Unglue Selection

Selects the grids should use standard contact instead of glued contact in glued bodies in SOL 400.

Format:

UNGLUE=n

Example:

UNGLUE=10

Descriptor	Meaning
n	Set identification number of the UNGLUE Bulk Data entry (Integer > 0).

Remarks:

1. This command is used only in SOL 400 for 3D Contact analysis.
2. The default SID of UNGLUE Bulk Data entry is defined on BCONTACT Case Control command if applicable; however, the SID on UNGLUE Case Control command can overwrite it.

VCCT

Virtual Crack Closure Technique for SOLs 400/600 Analysis

Selects grid sets to be used for virtual crack closure analysis in SOL 600 and SOL 400.

Format:

VCCT=N

Example:

VCCT=0

VCCT=1

Descriptor	Meaning
N	ID of a matching Bulk Data VCCT entry specifying the crack.

Remarks:

1. This entry can only be used in SOLs 400/600.
2. Different sets of cracks can be selected for different subcases using this option.
3. For SOL 600, N=0 may be entered above any subcases; then, the Bulk Data entry VCCT with ID=0 will be used in the Marc model definition section. The fracture mechanics calculations will be performed for all subcases. Otherwise, if N > 0, the matching Bulk Data entry VCCT will be used in Marc's history definition section for the applicable subcase, and all subsequent subcases, until a new VCCT is activated.

VECTOR

Displacement Output Request

Requests the form and type of displacement vector output.

See the description of the [DISPLACEMENT \(Case\), 298](#).

VELOCITY**Velocity Output Request**

Requests the form and type of velocity vector output.

Format:

$$\begin{aligned}
 VELOCITY & \left[\left(\begin{matrix} SORT1 \\ SORT2 \end{matrix} \right), \left[\begin{matrix} PRINT, PUNCH \\ PLOT \end{matrix} \right], \left[\begin{matrix} REAL \text{ or } IMAG \\ PHASE \end{matrix} \right], \left[\begin{matrix} PSDF, ATOC, CRMS \\ \text{or RALL} \end{matrix} \right], \right. \\
 & \quad \left. \left[\begin{matrix} RPPRINT \\ NORPRINT, RPUNCH \end{matrix} \right], [CID] \right] \\
 & = \left\{ \begin{array}{c} ALL \\ n \\ NONE \end{array} \right\}
 \end{aligned}$$

Examples:

VELOCITY=5

VELOCITY(SORT2, PHASE, PUNCH)=ALL

VELOCITY(SORT2, PRINT, PSDF, CRMS, RPUNCH)=20

VELOCITY(PRINT, RALL, NORPRINT)=ALL

Descriptor	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.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

Descriptor	Meaning
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 5.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 5.
CRMS	Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 5.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 5.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
CID	Request to print output coordinate system ID in printed output (.f06) file.
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 with identification numbers that appear on this SET command will be output (Integer > 0).

Remarks:

1. Velocity output is only available for transient and frequency response problems.
2. See Remark 1 under [DISPLACEMENT \(Case\), 298](#) for a discussion of SORT1 and SORT2.
3. VELOCITY=NONE overrides an overall output request.
4. Velocity results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
5. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
6. Note that the CID keyword affects only grid point related output, such as DISP, VELO, ACCEL, OLOAD, SPCF and MPCF. In addition, the CID keyword needs to appear only once in a grid point-related output request anywhere in the Case Control Section to turn on the printing algorithm.

VINTENSITY**Vibration Intensity, VI, Output Request**

Request output of vibration intensity for structural elements in SOLs 108 and 111 only.

Format:

$$\text{VINTENSITY} \left(\begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix} \right) = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

Examples:

`VINTENSITY = ALL`

Descriptor	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Vibration Intensities will be computed for all supported structural elements.
n	Set identification of a previously defined set of structural elements. Vibration Intensities will be computed for the structural elements in this set only.
NONE	Vibration intensity will not be processed.

Remarks:

1. VINTENSITY = NONE overrides an overall request.
2. This Case Control command can be used in SOL 108 and SOL 111 only
3. Structural element types covered for VI computations are BAR, BEAM, QUAD4, TRIA3, QUADR, TRIAR, QUAD8, TRIA6, HEXA, PENTA and TETRA.
4. For BAR/BEAM element types, VI is computed in element axial direction using following equation

$$VI_x = \text{Real}[-(F_x v_x^* + V_1 v_y^* + V_2 v_z^* + T \omega_x^* - M_2 \omega_y^* + M_1 \omega_z^*)] / \text{Area},$$

Where F_x = axial force in element x direction,

V_1 = shear force in element y direction,

V_2 = shear force in element z direction,
 T = torsion about element x direction,
 M_2 = Bending moment in element y direction,
 M_1 = Bending moment in element z direction,
 v_i = Translational velocity about element i direction,
 ω_i = Rotational velocity about element i direction,

'*' superscript denotes complex conjugate of the complex value. VI_x is computed at both end of BAR/BEAM element and the average of ends VI is presented as element VI_x .

5. For 2D element types, such as QUAD4/TRIA3, VI is computed in element coordinate system using following equations

$$VI_x = \text{REAL}[-(V_x v_z - M_x \omega_y^* + M_{xy} \omega_x^* + F_x v_x^* + F_{xy} v_y^*) / \text{Area}_x, \\ VI_y = \text{REAL}[-(V_y v_z + M_y \omega_x^* - M_{xy} \omega_y^* + F_y v_y^* + F_{yx} v_x^*) / \text{Area}_y,$$

Where V_x, V_y = transverse shear forces,

M_x, M_y = Bending moments,

M_{xy} = twisting moment,

F_x, F_y = Membrane forces,

F_{xy}, F_{yx} = Membrane shear,

$\text{Area}_x, \text{Area}_y$ = Area in the corresponding element axes,

v_i = Translational velocity about element i direction,

ω_i = Rotational velocity about element i direction,

'*' superscript denotes complex conjugate of the complex value. VI_i, VI_i are computed in element coordinate system using average corner velocities and the element forces at the element center.

6. For 3D element types, VI is computed in basic coordinate system using following equations

$$VI_x = \text{REAL}[-(\sigma_x v_x + \tau_{xy} v_y + \tau_{xz} v_z)]$$

$$VI_y = \text{REAL}[-(\sigma_y v_y + \tau_{yx} v_x + \tau_{yz} v_z)]$$

$$VI_z = \text{REAL}[-(\sigma_z v_z + \tau_{zx} v_x + \tau_{zy} v_y)]$$

Where VI_x, VI_y, VI_z = VI in basic coordinate system,

$\sigma_x, \sigma_y, \sigma_z$ = Normal stresses,

$\tau_{xy}, \tau_{yz}, \tau_{xz}$ = Shear stresses,

v_i = Translational velocity in basic i direction,

7. VI for all element types is presented in basic coordinate system in print and/or punch output.

VUGRID**View Geometry Output for p-Version Analysis**

Requests output of view grid and view element entries used in p-version element data recovery.

Format:

$$VUGRID \left[\begin{array}{c} PRINT, PUNCH \\ PLOT \end{array} \right] = \left\{ \begin{array}{c} ALL \\ n \end{array} \right\}$$

Example:

VUGRID (PRINT)=n

Descriptor	Meaning
ALL	All view element and grid entries will be output.
n	Set identification of a previously appearing SET command. Only those p-version elements with identification numbers that appear on this SET command will be output (Integer > 0).
PRINT or (blank)	Printer File (.f06)
PUNCH	Punch File (.pch)
PLOT	Plot File (.op2/.h5)

* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

Remarks:

1. VUGRID is processed only when an analysis with p-version elements is requested.
2. Only one VUGRID command per analysis is allowed.
3. The VUGRID command is used only for output control, and does not in anyway affect the p-version analysis.
4. See parameters VUHEXA, VUTETRA, and VUPENTA in [Parameters, 783](#), for renaming element entries.
5. See parameters VUELJUMP and VUGJUMP in [Parameters, 783](#), for numbering of view grid and view element entries.

WEIGHTCHECK**Rigid Body Mass Reduction Check**

At each stage of the mass matrix reduction, compute rigid body mass and compare with the rigid body mass in the g-set.

Format:

$$\text{WEIGHTCHECK} \left[\begin{array}{l} \left(\begin{array}{l} \text{PRINT} \\ \text{NOPRINT} \end{array} \right) \text{PUNCH}, \text{SET} = \left\{ \begin{array}{l} \text{G}, \text{N}, \text{N} + \text{AUTOSPC}, \text{F}, \text{A}, \text{V} \\ \text{ALL} \end{array} \right\} \\ \text{GRID} = \text{gid}, \text{CGI} = \left[\begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right], \left[\begin{array}{l} \text{WEIGHT} \\ \text{MASS} \end{array} \right] \end{array} \right] = \left\{ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right\}$$

Examples:

WEIGHTCHECK=YES

WEIGHTCHECK(GRID=12, SET=(G, N, A), MASS)=YES

Descriptor	Meaning
PRINT	Write output to the print file (Default).
NOPRINT	Do not write output to the print file.
PUNCH	Write output to the punch file.
SET	Selects degree of freedom set(s) (Default SET=G).
gid	Reference grid point for the calculation of rigid body motion. The default is the origin of the basic coordinate system.
CGI	For $\text{SET} \neq \text{G}$, CGI = YES requests output of center of gravity and mass moments of inertia (Default: CGI = NO).
WEIGHT/MASS	Selects output in units of weight or mass (Default = WEIGHT).

Remarks:

1. WEIGHTCHECK must be specified above the subcase level.
2. For SET=N, N+AUTOSPC, F, or A, the WEIGHTCHECK command also outputs a percentage loss or gain in the reduced rigid body mass matrix (e.g., MAA) as compared to the g-set rigid body mass matrix (e.g., MGG). G must also be requested to obtain this comparison; e.g., WEIGHTCHECK(SET=(G,A))=YES.
3. SET=N+AUTOSPC uses the mass matrix for the n-set with the rows corresponding to degrees of freedom constrained by the PARAM, AUTOSPC operation zeroed out. If AUTOSPC was not performed, then this check is redundant with respect to SET=N.
4. WEIGHTCHECK is available in all SOLs. However for the residual structure in SOLs 101, 105, 114, and 116, because no mass reduction is performed, only WEIGHTCHECK(SET=J) is available. The 'J' set does not include upstream superelements.

5. If Lagrange multipliers are present via RIGID=LAGRAN or LGELIM then for degree-of-freedom sets N, N+AUTOSPC, F, and A the check will be performed on degree-of-freedom sets NL, NL+AUTOSPC, FL, and AL. The output will also be labeled accordingly.

WETSENS

Sensitivity Wetted Grids for SOL 108/111

Select SOLUTION frequencies and RESPONSE DOFs for the generation of sensitivity for wetted grids.

Format:

$$WETSENS \left(\begin{bmatrix} PRINT, PUNCH \\ PLOT \end{bmatrix}, \begin{bmatrix} REAL \text{ or } IMAG \\ PHASE \end{bmatrix}, [THRESH = P], RESPONSE = r, \left[SOLUTION = \begin{cases} ALL \\ self \end{cases} \right] \right),$$

$$[WETTED, SQWETT] = \begin{cases} ALL \\ n \\ NONE \end{cases}$$

Example:

```
SET 81 = 100.0, 120.0
SET 91 = 11240/T3, 4001/T1
SET 95 = 9000000 THRU 9000050
$
WETSENS (RESPONSE=91,solution=81,WETTED) = 95
```

Descriptor	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

THRESH	The magnitude of element sensitivity less than p will be suppressed in all output files: print, punch, plot, .op2, and .xdb. (Default = 0.0).
RESPONSE	Adjoint load response will be computed for unit load applied at grid point components in SET r.
SOLUTION	Frequency responses at these forcing frequencies, defined in setf, will be used for element sensitivity computation. (Default=all forcing frequencies)
WETTED	Sensitivity for wetted grids will be computed and output.

Descriptor	Meaning
SQWETT	Squared sensitivity for wetted grids will be computed and output.
ALL	Sensitivities for all elements will be calculated.
n	Set identification number. Sensitivity for all elements specified on the SET n command will be calculated. The SET n command must be specified in the same subcase as the ELSSENS command, or above all subcases (Integer > 0). The IDs in set n must be GID (grid ID).
NONE	Elemental sensitivity will not be output.

Remarks:

1. Set r for RESPONSE on WETSENS is default to set r on ELSENS. If no ELSENS in the deck, set r for WETSENS must be provided.
2. The equations for various options of WETSENS

$$\text{WETSENS(WETTED)} = [U_{\text{setf}}]^t [\text{AGG}] [U_r]$$

$$\text{WETSENS(SQWETT)} = [U_{\text{setf}}]^t [\text{AGG}] [U_r] + [U_{\text{setf}}]^{*t} [\text{AGG}]^* [U_r]^*$$

where $[U_{\text{setf}}]$ is the displacement of SOLUTION

$[U_r]$ is the displacement of RESPONSE

$[\text{AGG}]$ is Fluid/Structure Coupling matrix

superscript * means complex conjugate of the term.

Case Control Applicability Tables

The following tables describe the applicability of Case Control commands to Solution Sequences:

Table 5-3 and Table 5-4	SOLs (101 through 200) -- Subcase Definition, Superelement Control, and Auxiliary Model Control
Table 5-5 and Table 5-6	SOLs (101 through 200) -- Data Selection
Table 5-7 and Table 5-8	SOLs (101 through 200) -- Output Selection

Table 5-3 Case Control Commands in SOLs 101 Through 112 -- Subcase Definition, Superelement Control, and Auxiliary Model Control

Command Name	Solution Number									
	101	103	105	106	107	108	109	110	111	112
ADACT	X	X								
ANALYSIS										
AUXCAS										
AUXMODEL										
BEGIN BULK	X	X	X	X	X	X	X	X	X	X
CAMPBLL					X					
EXTDRIN	X	X			X	X	X	X	X	X
EXTDROUT	X	X			X	X	X	X	X	X
EXTSEOUT	X	X			X	X	X	X	X	X
MASTER	X	X	X	X	X	X	X	X	X	X
MCFRACTION								X	X	X
MODES		X								
OUTPUT(blank)	X	X	X	X	X	X	X	X	X	X
OUTPUT(PLOT)	X	X	X	X	X	X	X	X	X	X
OUTPUT (POST) or SETS DEFINITION	X	X	X	X	X	X	X	X	X	X
OUTPUT (XYPLOT)	X	X	X	X	X	X	X	X	X	X
REPCASE	X	X								
SEALL	X	X	X	X	X	X	X	X	X	X
SEDR	X	X	X	X	X	X	X	X	X	X
SEDV										
SEEXCLUD	X	X	X	X	X	X	X	X	X	X
SEFINAL	X	X	X	X	X	X	X	X	X	X
SEKRR	X	X	X	X	X	X	X	X	X	X
SELG	X		X	X		X	X		X	X
SELR	X		X	X		X	X		X	X
SEMR	X	X	X	X	X	X	X	X	X	X
SEMR		X	X		X	X	X	X	X	X
SERE										

Table 5-3 Case Control Commands in SOLs 101 Through 112 -- Subcase Definition, Superelement Control, and Auxiliary Model Control

Command Name	Solution Number									
	101	103	105	106	107	108	109	110	111	112
STOCHASTICS	X	X								
SUBCASE	X	X	X	X	X	X	X	X	X	X
SUBCOM	X									
SUBSEQ	X									
SUPER	X	X	X	X	X	X	X	X	X	X
SYM	X									
SYMCOM	X									
SYMSEQ	X									

Table 5-4 Case Control Commands in SOLs 114 Through 400 -- Subcase Definition, Superelement Control, and Auxiliary Model Control

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
ADACT												
ANALYSIS									X		X	
AUXCAS											X	
AUXMODEL											X	
BEGIN BULK	X	X	X	X	X	X	X	X	X	X	X	
EXTDRIN												X
EXTDROUT												X
EXTSEOUT												X
MASTER	X	X	X	X	X	X	X	X	X	X	X	
MODES		X	X									X
OUTPUT(blank)	X	X	X	X	X	X	X	X	X	X	X	
OUTPUT(PLOT)	X	X	X	X	X	X	X	X	X	X	X	
OUTPUT (POST) or SETS DEFINITION	X	X	X	X	X	X	X	X	X	X	X	
OUTPUT(XYPLOT)	X	X	X	X	X	X	X	X	X	X	X	
REPCASE												X
SEALL	X	X	X	X	X	X	X	X	X	X	X	

Table 5-4 Case Control Commands in SOLs 114 Through 400 -- Subcase Definition, Superelement Control, and Auxiliary Model Control

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
SEDR	X	X	X	X	X	X	X	X	X	X	X	
SEDV												X
SEEXCLUD	X	X	X	X	X	X	X	X	X	X	X	
SEFINAL	X	X	X	X	X	X	X	X	X	X	X	
SEKRR	X	X	X	X	X	X	X	X	X	X	X	
SELG	X		X	X	X	X		X	X	X	X	
SELRL	X		X	X	X	X		X	X	X	X	
SEMR	X	X	X	X	X	X	X	X	X	X	X	
SEMR		X	X	X	X	X	X	X		X	X	
SERE												X
STOCHASTICS												X X
SUBCASE	X	X	X	X	X	X	X	X	X	X	X	
SUBCOM												X
SUBSEQ												X
SUPER	X	X	X	X	X	X	X	X	X	X	X	
SYM												X
SYMCOM												X
SYMSEQ												X

Table 5-5 Case Control Commands in SOLs 101 Through 112 -- Data Selection

Command Name	Structured Solution Number										
	101	103	105	106	107	108	109	110	111	112	
ADAPT	X	X									
AUTOSPC	X	X	X		X	X	X	X	X	X	
AXISYMME	X	X	X		X	X	X	X	X	X	
B2GG	X	X	X	X	X	X	X	X	X	X	
B2PP					X	X	X	X	X	X	
BC		X									
CLOAD				X							
CMETHOD					X				X		

Table 5-5 Case Control Commands in SOLs 101 Through 112 -- Data Selection (continued)

Command Name	Structured Solution Number									
	101	103	105	106	107	108	109	110	111	112
DEFORM	X		X							
DESGLB										
DESOBJ										
DESSUB										
DLOAD					X	X	X	X	X	X
DSYM										
FMETHOD										
FREQUENC						X			X	
GUST										
HARMONIC	X	X	X		X	X	X	X	X	X
IC							X			
K2GG	X	X	X	X	X	X	X	X	X	X
K2PP					X	X	X	X	X	X
LOAD	X		X	X		X	X		X	X
LOADSET	X		X	X		X	X		X	X
M2GG	X	X	X	X	X	X	X	X	X	X
M2PP					X	X	X	X	X	X
METHOD		X		X	X	X	X	X	X	X
MFLUID		X			X	X	X	X	X	X
MODTRAK										
MPC	X	X	X	X	X	X	X	X	X	X
NLPARM					X					
NONLINEA							X			X
OMODES		X	X					X*	X*	X*
P2G	X		X	X		X	X		X	X
RANDOM						X			X	
RESVEC		X		X	X	X	X	X	X	X
SDAMPING								X	X	X

Table 5-5 Case Control Commands in SOLs 101 Through 112 -- Data Selection (continued)

Command Name	Structured Solution Number									
	101	103	105	106	107	108	109	110	111	112
SDENSITY	X	X	X							
SMETHOD	X			X		X			X	
SPC	X	X	X	X	X	X	X	X	X	X
STATSUB*	X	X	X		X	X	X	X	X	X
SUPPORT1	X	X	X	X	X	X	X	X	X	X
TEMPER(INIT)				X						
TEMPER(LOAD)	X		X	X		X	X		X	X
TEMPER(MATE)	X	X	X	X	X	X	X	X	X	X
TFL					X	X	X	X	X	X
TRIM										
TRIMGRP									X	
TSTEP							X			X
WEIGHTCHECK	X	X	X	X	X	X	X	X	X	X

*If STATSUB is specified, then the Case Control commands that select static loads become applicable to the solution sequence supporting STATSUB.

Table 5-6 Case Control Commands in SOLs 114 Through 400 -- Data Selection

Command Name	Solution Number										
	114	115	116	118	129	144	145	146	153	159	200
ADAPT											
AUTOSPC	X	X	X	X		X	X	X		X	X
AXISYMME											X
B2GG	X	X	X	X	X	X	X	X	X	X	X
B2PP				X	X		X	X		X	X
BC											
CLOAD									X		
CMETHOD							X				
DEFORM	X		X			X					X
DESGLB										X	
DESMOD											X

Table 5-6 Case Control Commands in SOLs 114 Through 400 -- Data Selection (continued)

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
DESOBJ											X	
DESSUB											X	
DESVAR											X	
DLOAD				X	X		X	X		X	X	
DSYM	X	X	X	X								
FMETHOD						X					X	
FREQUENC				X				X			X	
GUST							X	X				
HARMONIC	X	X	X	X							X	
IC											X	
K2GG	X	X	X	X	X	X	X	X	X	X	X	
K2PP				X	X		X	X		X	X	
LOAD	X		X	X	X	X		X	X		X	
LOADSET	X		X	X	X	X		X	X	X	X	
M2GG	X	X	X	X	X	X	X	X	X	X	X	
M2PP				X	X		X	X		X	X	
METHOD		X	X	X	X	X	X	X	X	X	X	
MFLUID		X	X	X	X	X	X	X			X	
MODTRAK												
MPC	X	X	X	X	X	X	X	X	X	X	X	
NLBUCK												X
NLPARM									X			
NONLINEA					X				X			
OMODES		X										X
P2G	X		X	X	X	X		X	X		X	
RANDOM				X				X				
RESVEC		X		X	X	X	X	X	X	X	X	
SDAMPING							X	X				X
SMETHOD												
SPC	X	X	X	X	X	X	X	X	X	X	X	
STATSUB				X							X	

Table 5-6 Case Control Commands in SOLs 114 Through 400 -- Data Selection (continued)

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
SUPPORT1	X	X	X	X	X	X	X	X	X	X	X	
TEMPER(INIT)												
TEMPER(LOAD)	X			X		X		X				X
TEMPER(MATE)	X	X	X	X	X	X	X	X	X	X	X	
TFL				X	X		X	X		X	X	
TRIM						X						X
TRIMGRP												X
TSTEP					X			X		X	X	
WEIGHTCHECK	X	X	X	X	X	X	X	X				X

Table 5-7 Case Control Commands in SOLs 101 Through 112 -- Output Selection

Command Name	Solution Number									
	101	103	105	106	107	108	109	110	111	112
ACCELERA		X				X	X		X	X
ACFPMRESULT						X				X
ACPOWER						X				X
AEROF										
APPRESSURE										
BOUTPUT				X						
CMSENRGY		X		X				X	X	X
DATAREC	X	X				X	X		X	X
DISPLACE	X	X	X	X	X	X	X	X	X	X
DSAPRT										
ECHO	X	X	X	X	X	X	X	X	X	X
EDE		X			X	X	X	X*	X	X
EKE		X			X	X	X	X*	X	X
ELSDCON	X		X							
ELSUM	X	X	X	X	X	X	X	X	X	X
ENTHALPY										
ERP						X			X	

Table 5-7 Case Control Commands in SOLs 101 Through 112 -- Output Selection (continued)

Command Name	Solution Number									
	101	103	105	106	107	108	109	110	111	112
ESE	X	X	X	X	X	X	X	X*	X	X
FLUX	X									
FORCE	X	X	X	X	X	X	X	X	X	X
GPFORCE	X	X	X	X			X ⁺			X ⁺
GPKE		X						X*	X*	X*
GPSDCON	X		X							
GPSTRAIN	X	X		X			X			X
GPSTRESS	X	X		X			X			X
GROUNDCHECK	X	X	X	X	X	X	X	X	X	X
HARMONIC	X	X	X		X	X	X	X	X	X
HDOT										
HOUTPUT										
INTENSITY						X				X
LABEL	X	X	X	X	X	X	X	X	X	X
LINE	X	X	X	X	X	X	X	X	X	X
MAXLINES	X	X	X				X			X
MAXMIN (old form)	X	X	X				X			X
MAXMIN (DEF)	X	X	X	X			X			X
MPCFORCE	X	X	X		X	X	X	X	X	X
MPRES		X			X	X	X	X	X	X
NLLOAD							X			X
NOUTPUT										
OFREQUEN						X				X
OLOAD	X		X	X		X	X		X	X
OTIME							X			X
OUTRCV	X	X				X	X			X
PAGE	X	X	X	X	X	X	X	X	X	X
PARTN	X	X	X	X	X	X	X	X	X	X
PLOTID	X	X	X	X	X	X	X	X	X	X
POST	X									
PRESSURE		X			X	X	X			

Table 5-7 Case Control Commands in SOLs 101 Through 112 -- Output Selection (continued)

Command Name	Solution Number									
	101	103	105	106	107	108	109	110	111	112
SACCELER						X	X		X	X

Table 5-7 Case Control Commands in SOLs 101 Through 112 -- Output Selection (continued)

Command Name	Solution Number									
	101	103	105	106	107	108	109	110	111	112
SDISPLAC		X	X		X	X	X		X	X
SET	X	X	X	X	X	X	X	X	X	X
SKIP	X	X	X	X	X	X	X	X	X	X
SPCFORCE	X	X	X	X	X	X	X	X	X	X
STRAIN	X	X	X	X	X	X	X	X	X	X
STRESS	X	X	X	X	X	X	X	X	X	X
STRFIELD	X	X	X				X			X
SUBTITLE	X	X	X	X	X	X	X	X	X	X
SURFACE	X	X		X			X			X
SVECTOR		X	X					X	X	X
SVELOCITY						X	X		X	X
THERMAL	X									
TITLE	X	X	X	X	X	X	X	X	X	X
VECTOR	X	X	X	X	X	X	X	X	X	X
VELOCITY						X	X		X	X
VOLUME	X	X		X			X			X
VUGRID	X	X				X	X		X	X

^{*}For modal part of solution.[†]Forces limited to stiffness contributions.

Table 5-8 Case Control Commands in SOLs 114 Through 400 --Output Selection

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
ACCELERA			X	X			X			X		
ACFPMRESULT												
ACPOWER												
AEROF						X	X	X			X	
APPRESSURE					X						X	
BOUTPUT				X					X	X		
CMSENRGY						X	X				X	
DATAREC											X	
DISPLACE	X	X	X	X	X	X	X	X	X	X	X	
DSAPRT												X
ECHO	X	X	X	X	X	X	X	X	X	X	X	
EDE		X		X								X
EKE		X		X								X
ELSDCON						X						X
ELSUM	X	X	X	X	X	X	X	X				X
ENTHALPY											X	
ERP											X	X
ESE	X	X	X	X		X						X
FLUX	X									X	X	X
FORCE	X	X	X	X	X	X	X	X	X	X	X	
GPFORCE	X	X	X	X								X
GPKE		X										X
GPSDCON	X					X			X			X
GPSTRAIN	X	X	X		X			X				X
GPSTRESS	X	X	X		X	X					X	X
GROUNDCHEC K	X	X	X	X	X	X	X	X				X
HARMONY	X	X	X	X								X
HDOT										X		
HOUTPUT	X	X	X	X								

Table 5-8 Case Control Commands in SOLs 114 Through 400 --Output Selection (continued)

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
INTENSITY												
LABEL	X	X	X	X	X	X	X	X	X	X	X	
LINE	X	X	X	X	X	X	X	X	X	X	X	
MAXLINES	X	X	X	X	X	X	X	X	X	X	X	
MAXMIN(DEF)												X
MPCFORCE	X	X	X	X		X	X	X			X	
MPRES			X			X	X	X			X	
NLLOAD											X	
NLOPRM												X
NLSTEP												X
NOUTPUT	X	X	X	X								
OFREQUEN					X				X		X	
OLOAD	X		X	X	X	X			X	X	X	X
OTIME	X		X	X	X	X			X	X	X	X
OUTRCV												
PAGE	X	X	X	X	X	X	X	X	X	X	X	
PARTN	X	X	X	X	X	X	X	X	X	X	X	
PLOTID	X	X	X	X	X	X	X	X	X	X	X	
PRESSURE												X
SACCELER					X	X			X			X
SDISPLAC		X			X	X		X	X		X	X
SET	X	X	X	X	X	X	X	X	X	X	X	
SKIP	X	X	X	X	X	X	X	X	X	X	X	
SPCFORCE	X	X	X	X	X	X	X	X	X	X	X	
STRAIN	X	X	X	X	X	X	X	X	X	X	X	
STRESS	X	X	X	X	X	X	X	X	X	X	X	
STRFIELD	X	X	X			X						X
SUBSET												X
SUBTITLE	X	X	X	X	X	X	X	X	X	X	X	

Table 5-8 Case Control Commands in SOLs 114 Through 400 --Output Selection (continued)

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
SURFACE	X	X	X								X	
SVECTOR		X	X	X			X		X		X	
SVELOCITY				X	X			X			X	
TEMPERATURE												
THERMAL	X									X	X	X
TITLE	X	X	X	X	X	X	X	X	X	X	X	
TRIMF						X						X
VECTOR	X	X	X	X	X	X	X	X	X	X	X	
VELOCITY				X	X			X			X	
VOLUME	X	X	X								X	
VUGRID												

X-Y PLOT Commands

The X-Y output request packet of the Case Control Section includes all commands between either OUTPUT(XYPLOT) or OUTPUT(XYOUT), and either BEGIN BULK or OUTPUT(PLOT). The remainder of this section describes the X-Y output commands.

A single set of plotted X-Y pairs is known as a curve. Curves are the entities to be plotted. The surface (paper, microfilm frame, etc.) on which one or more curves is plotted is known as a frame. Curves may be plotted on a whole frame, an upper-half frame, or a lower-half frame. Grid lines, tic marks, axes, axis labeling, and other graphic control options may be chosen by the user. The program will select defaults for parameters not selected by the user.

Only two commands are required for an X-Y output request.

1. Only one of OUTPUT(XYPLOT) or OUTPUT(XYOUT) at the beginning of the X-Y output command packet.
2. At least one of the commands XYPILOT, XYPEAK, XYPRINT, XYPUNCH, XYPAPLOT.

The commands OUTPUT(XYPLOT) and OUTPUT(XYOUT) are equivalent. If the X-Y output is to be printed and/or punched, a PLOTTER command is not required.

If only the required commands are used, the graphic control options will all assume default values. Curves using all default parameters have the following general characteristics.

1. Tic marks are drawn on all edges of the frame. Five spaces are provided on each edge of the frame.
2. All tic marks are labeled with their values.
3. Linear scales are used.
4. Scales are selected such that all points fall within the frame.
5. The plotter points are connected with straight lines.
6. The plotted points are not identified with symbols.

The above characteristics may be modified by inserting any of the parameter definition commands in the next section, ahead of the XY____ command(s). The use of a parameter definition command sets the value of that parameter for all following command operation commands unless the CLEAR command is inserted. If grid lines are requested, they will be drawn at the locations of all tic marks that result from defaults or user request. The locations of tic marks (or grid lines) for logarithmic scales cannot be selected by the user. Values for logarithmic spacing are selected by the program. The values for the number of tic marks (or grid lines) per cycle depend on the number of logarithmic cycles required for the range of the plotted values.

The definition and rules for the X-Y output commands follow. The form of X-Y output commands differ in many instances from that of similar commands used in the OUTPUT(PLOT) section.

X-Y Output Command Summary

	Commands Applied To All Curves
PLOTTER	Selects format of plot file for interpretation by plotter postprocessor.
CAMERA	Selects plotter media.

Commands Applied To All Curves	
PENSIZE	Selects pen number.
DENSITY	Selects the line density for microfilm plotters only.
XPAPER	Defines the size of the paper in x-direction.
YPAPER	Defines the size of the paper in y-direction.
XMIN	Specifies the minimum value on the x-axis.
XMAX	Specifies the maximum value on the x-axis.
XLOG	Selects logarithmic or linear x-axis.
YAXIS	Controls the plotting of the y-axis on all curves.
XINTERCEPT	Specifies the location of the x-axis on the y-axis.
UPPER TICS	Specifies how to draw tic marks on upper edge.
LOWER TICS	Specifies how to draw tic marks on lower edge.
CURVELINESYMBOL	Selects lines and/or symbols to be drawn through the x-y points.
XDIVISIONS	Specifies spacing of tic marks on the x-axis for all curves.
XVALUE PRINT SKIP	Specifies how often to print the x-values alongside the x-axis tic marks.
CLEAR	Resets X-Y Plot commands to their default value.
XTITLE	Defines a character string that will appear along the x-axis.
TCURVE	Defines a character string that will appear at the top of the plot frame.
LONG	Controls amount of curve's summary printout.
CSCALE	Defines scale factor for characters in the plot frame.
Commands Applied to Whole Frame Curves Only	
YMIN	Specifies the minimum value on the y-axis.
YMAX	Specifies the maximum value on the y-axis.
XAXIS	Controls the plotting of the x-axis.
YINTERCEPT	Specifies the location of the y-axis on the x-axis.
YLOG	Selects logarithmic or linear y-axis.
LEFT TICS	Specifies how to draw tic marks on left edge.
RIGHT TICS	Specifies how to draw tic marks on right edge of the frame.
ALLEDGE TICS	Specifies how to draw tic marks on all edges of the frame.
YDIVISIONS	Specifies spacing of tic marks on the y-axis.
YVALUE PRINT SKIP	Specifies how often to print the y-values alongside the y-axis tic marks applies.

Commands Applied to Whole Frame Curves Only	
XGRID LINES	Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks.
YGRID LINES	Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks.
YTITLE	Defines a character string that will appear along the y-axis
Commands Applied to Upper Half Frame Curves Only	
YTMIN	Specifies the minimum value on the y-axis.
YTMAX	Specifies the maximum value on the y-axis.
YTAXIS	Controls the plotting of the y-axis.
YTINTERCEPT	Specifies the location of the y-axis on the x-axis.
YTLOG	Selects logarithmic or linear y-axis.
TLEFT TICS	Specifies how to draw tic marks on the left edge.
TRIGHT TICS	Specifies how to draw tic marks on the right edges.
TALL EDGE TICS	Specifies how to draw tic marks on all edges.
YTDIVISIONS	Specifies spacing of tic marks on the y-axis.
YTVALUE PRINT	Specifies how often to print the y-values alongside the y-axis tic marks.
SKIP	
XTGRID LINES	Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks.
YTGRID LINES	Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks.
YTITLE	Defines a character string that will appear along the y-axis.
Commands Applied to Lower Half Frame Curves Only	
YBMIN	Specifies the minimum value on the y-axis.
YBMAX	Specifies the maximum value on the y-axis.
XBAXIS	Controls the plotting of the x-axis.
YBINTERCEPT	Specifies the location of the y-axis on the x-axis.
YBLOG	Selects logarithmic or linear y-axis.
BLEFT TICS	Specifies how to draw tic marks on left edge.
BRIGHT TICS	Specifies how to draw tic marks on right edge.
BALL EDGE TICS	Specifies how to draw tic marks on all edges.
YBDIVISIONS	Specifies spacing of tic marks on the y-axis.

Commands Applied to Lower Half Frame Curves Only	
YBVALUE PRINT	Specifies how often to print the y-values alongside the y-axis tic marks.
SKIP	
XBGRID LINES	Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks.
YBGRID LINES	Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks.
YBTITLE	Defines a character string that will appear along the y-axis.

X-Y Plot Generation Commands	
XYPAPLOT	Generate X-Y plots for a printer.
XYPEAK	Print only the summary for all curves.
XYPLOT	Generate X-Y plots for a plotter.
XYPRINT	Generate table of X-Y pairs for a printer.
XYPUNCH	Generate table of X-Y pairs for the PUNCH file

ALLEDGE TICS

Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on all edges of the frame.

Format:

ALLEDGE TICS tic

Example:

ALLEDGE -1

Descriptor	Meaning
tic	Specifies how to draw tic marks (Integer, Default= 0).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values (Default).
1	Draw tic marks and associated values.

Remarks:

1. ALLEDGE TICS applies to whole frame curves only.
2. To determine if on any given edge (a) tic marks will be drawn without values, (b) no tic marks or values will be drawn, or (c) tic marks with values will be drawn, the following sum must be computed by the user. Add the tic values of the edge in question to its associated ALLEDGE TICS, TALL EDGE TICS, or BALL EDGE TICS tic values. If the resulting value is less than zero, tic marks will be drawn without values. If the resulting value is zero, no tic marks or values will be drawn. If the resulting value is greater than zero, tic marks with values will be drawn. The user should be careful in the use of the ALLEDGE TICS, TALL EDGE TICS, or BALL EDGE TICS commands. For example, the use of only the ALLEDGE TICS = -1 command will result in no tic marks or values being drawn, since the default values for individual edges is +1. Tic values input may only be -1, 0, or 1.

BALL EDGE TICS**Controls Drawing of Tic Marks on Lower Half**

Specifies how to draw tic marks on lower half of frame.

Format:

BALL EDGE TICS tic

Example:

BALL EDGE TICS -1

Describer	Meaning
tic	Specifies how to draw tic marks (Integer, Default = 0).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values (Default).
1	Draw tic marks and associated values.

Remarks:

1. BALL EDGE TICS applies to lower frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 666](#).

BLEFT TICS

Controls Drawing of Tic Marks on Left Edge

Specifies how to draw tic marks on left edge of lower half of frame.

Format:

BLEFT TICS tic

Example:

BLEFT TICS -1

Descriptor	Meaning
tic	Specifies how to draw tic marks (Integer; Default= 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. BLEFT TICS applies to lower frame curves only.
2. See Remark 2 under [ALLEdge TICS, 666](#).
3. See related command [BRIGHT TICS, 669](#).

BRIGHT TICS

Controls Drawing of Tic Marks on Right Edge

Specifies how to draw tic marks on right edge of lower half of frame.

Format:

BRIGHT TICS tic

Example:

BRIGHT TICS -1

Descriptor	Meaning
tic	Specifies how to draw tic marks (Integer; Default = 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. BRIGHT TICS applies to lower frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 666](#).

CAMERA**Plotter Media Selection**

Selects plotter media.

Format:

CAMERA ctype

Example:

CAMERA 1

Descriptor	Meaning
ctype	Camera type (Integer 1, 2, or 3; Default= 2).
1	Film
2	Paper (Default)
3	Both

Remark:

1. If the CAMERA command is not specified then CAMERA 2 is assumed.

CLEAR Resets X-Y Plot Commands

Resets X-Y Plot commands to their default values.

Format

CLEAR

Remark:

1. All commands except XTITLE, YTITLE, YTTITLE, YBTITLE, and TCURVE will revert to their default values.

CSCALE

Character Scale Factor

Defines scale factor for characters in the plot frame. See the command [CSCALE, 750](#) in the OUTPUT(PLOT) Section.

CURVELINESYMBOL**Curve, Line and Symbol Selection**

Selects lines and/or symbols to be drawn through the x-y points.

Format:

CURVELINESYMBOL symtype

Example:

CURV 4

Descriptor	Meaning
symtype	Specifies the symbol drawn at the x-y points. If symtype is 0 then only lines will be drawn through the points with no symbol. If symtype is less than zero then only the symbol and not the lines will be drawn. If symtype is greater than zero then both the symbol and the lines will be drawn (-9 ≤ Integer ≤ 9; Default= 0).

symtype	Symbol
0	none
1	X
2	*
3	+
4	-
5	.
6	×
7	[]
8	<>
9	/ \

Remark:

1. If more than one curve is plotted per frame, then the symbol number is incremented by 1 for each curve.

DENSITY

Microfilm Plotter Line Density

Selects the line density for microfilm plotters only.

Format

DENSITY d

Example

DENS 3

Descriptor	Meaning
d	Specifies line density scale factor for microfilm plotters. A line density of d is d times heavier than a line density of 1 (Integer ≥ 0 ; Default = 1).

LEFT TICS**Controls Drawing of Tic Marks on Left Edge**

Specifies how to draw tic marks on left edge of whole frame curves.

Format:

LEFT TICS tic

Example:

LEFT -1

Descriptor	Meaning
tic	Specifies how to draw tic marks (Integer; Default = 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. LEFT TICS applies to whole frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 666](#).
3. See related command [RIGHT TICS, 680](#).

LONG

Summary Print Control

Controls amount of curve's summary printout.

Format:

LONG{ YES }
NO }

Describers	Meaning
YES	One page for each curve's summary (Default).
NO	Condensed curve summary.

Remark:

1. If LONG is not specified, then LONG=NO is assumed.

LOWER TICS

Controls Drawing of Tic Marks on Lower Edge

Specifies how to draw tic marks on lower edge.

Format:

LOWER TICS tic

Example:

LOWER -1

Describers	Meaning
tic	Specifies how to draw tic marks (Integer; Default= 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. LOWER TICS applies to all curves.
2. See Remark 2 under [ALLEDGE TICS, 666](#).

PENSIZE

Pen Selection

Selects pen number.

Format:

PENSIZE p

Example:

PENS 3

Descriptor	Meaning
p	Specifies pen number to be used to generate the plot (Integer > 0; Default = 1).

PLOTTER

X-Y Plot File Format

See the command [PLOTTER, 769](#) in the OUTPUT(PLOT) Section.

RIGHT TICS

Controls Drawing of Tic Marks on Right Edge

Specifies how to draw tic marks on right edge of the frame.

Format:

RIGHT TICS tic

Example:

RIGHT -1

Describers	Meaning
tic	Specifies how to draw tic marks (Integer; Default = 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. RIGHT TICS applies to whole frame curves only.
2. See Remark 2 under [ALLEdge TICS, 666](#).
3. See related command [LEFT TICS, 675](#).

SEPLOT**Superelement Plot Delimiter**

Assigns the subsequent PLOT or XYPLOT commands to one or more superelements.

Format:

SEPLOT seid1 [seid2 ...]

Examples:

SEPLOT 5

SEPLOT 0 3 7 200

Descriptor	Meaning
seidi	Superelement identification number (Integer ≥ 0).

Remarks:

1. See also related command SEUPPLOT.
2. Any PLOT or XYPLOT commands appearing above all SEPLOT (or SEUPPLOT) commands will apply in all SEPLOT (or SEUPPLOT) packets.
3. For multiple PLOT or XYPLOT commands, there should be a SEPLOT command with each PLOT. For the special case where the PLOTS or XYPLOTS refer to the same superelements and use the same FIND, a single SEPLOT followed by a single FIND may be placed above all commands.

SEUPPLOT**Superelement Plot Delimiter**

Assigns the subsequent PLOT or XYPILOT commands to a superelement and all of its upstream superelements.

Format:

SEUPPLOT seid

Example:

SEUPPLOT 7

Descriptor	Meaning
seid	Superelement identification number (Integer ≥ 0).

Remarks:

1. See also related command SEPLOT.
2. Any PLOT or XYPILOT commands appearing above all SEUPPLOT (or SEPLOT) commands will apply in all SEUPPLOT (or SEPLOT) packets.
3. For multiple PLOT or XYPILOT commands, there should be a SEUPPLOT command with each PLOT. For the special case where the PLOTS or XYPILOTs refer to the same superelements and use the same FIND, a single SEUPPLOT followed by a single FIND may be placed above all the commands.

TALL EDGE TICS

Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on all edges of the upper half of the frame.

Format:

TALL EDGE TICS tic

Example:

TALL -1

Describers	Meaning
tic	Specifies how to draw tic marks (Integer; Default = 0).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values (Default).
1	Draw tic marks and associated values.

Remarks:

1. TALL EDGE TICS applies to upper half frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 666](#).

TCURVE

Curve Title

Defines a character string that will appear at the top of the plot frame.

Format:

TCURVE ctitle

Example:

TCUR RIGHT WING -- LOAD CASE 3

Descriptor	Meaning
ctitle	Any character string (Character; Default = blank).

Remark:

1. TCURVE may not be continued to the next command line.

TLEFT TICS

Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on the left edge of the upper half of the frame.

Format:

TLEFT TICS tic

Example:

TLEFT -1

Describers	Meaning
tic	Specifies how to draw tic marks (Integer; Default= 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. TLEFT TICS applies to upper half frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 666](#).
3. See related command [TRIGHT TICS, 686](#).

TRIGHT TICS

Controls Drawing of Tic Marks on the Right Edge

Specifies how to draw tic marks on all edges of the upper half of the frame.

Format:

TRIGHT TICS tic

Example:

TRIGHT -1

Describers	Meaning
tic	Specifies how to draw tic marks (Integer; Default = 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. TRIGHT TICS applies to upper half frame curves only.
2. See Remark 2 under [ALLEdge TICS, 666](#).
3. See related command [TLEFT TICS, 685](#).

UPPER TICS

Controls Drawing Of Tic Marks On Upper Edge

Specifies how to draw tic marks on upper edge.

Format:

UPPER TICS tic

Example:

UPPER -1

Describers	Meaning
tic	Specifies how to draw tic marks (Integer; Default= 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. UPPER TICS applies to all curves.
2. See Remark 2 under [ALLEDGE TICS, 666](#).
3. See related command [LOWER TICS, 677](#).

XAXIS**X-Axis Plot Control**

Controls the plotting of the x-axis on whole frame curves only.

Format:

XAXIS{ YES }
NO

Describers	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis (Default).

Remarks:

1. XAXIS applies to whole frame curves only.
2. See related command [YAXIS, 712](#).

XBAXIS**X-Axis Plot Control**

Controls the plotting of the x-axis on lower half frame curves only.

Format:

XBAXIS{ YES }
 | NO |

Describers	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis (Default).

Remark:

1. XBAXIS applies to lower half frame curves only.

XBGRID LINES**Plot X-Axis Grid Lines**

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on lower half frame curves only.

Format:

XBGRID LINES { YES }
 { NO }

Describers	Meaning
YES	Plot the x-axis grid lines.
NO	Do not plot the x-axis grid lines (Default).

Remarks:

1. XBGRID applies to lower half frame curves only.
2. See related command [YBGRID LINES, 715](#).

XDIVISIONS**Tic Spacing on Y-Axis**

Specifies spacing of tic marks on the x-axis for all curves.

Format:

XDIVISIONS xd

Example:

XDIV 10

Descriptor	Meaning
xd	Number of spaces between tic marks on x-axis (Integer > 0; Default = 5).

Remarks:

1. XDIVISIONS applies to all curves and to the commands: UPPER TICS, LOWER TICS, and YINTERCEPT.
2. XDIVISIONS is ignored for a logarithmic x-axes.

XGRID LINES

Plot X-Axis Grid Lines

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on whole frame curves only.

Format:

XGRID LINES { YES }
NO

Describers	Meaning
YES	Plot the x-axis grid lines.
NO	Do not plot the x-axis grid lines (Default).

Remarks:

1. XGRID applies to whole frame curves only.
2. See related command [YGRID LINES, 723](#).

XINTERCEPT

Location of X-Axis on Y-Axis

Specifies the location of the x-axis on the y-axis.

Format:

XINTERCEPT xi

Example:

XINT 50.

Describer	Meaning
xi	Location of x-axis on the y-axis (Real; Default = 0.0).

XLOG**Logarithmic or Linear X-Axis**

Selects logarithmic or linear x-axis.

Format:

XLOG{ YES }
NO }

Describers	Meaning
YES	Plot a logarithmic x-axis.
NO	Plot a linear x-axis (Default).

Remarks:

1. XLOG applies to all curves.
2. The default value for tic division interval depends on the number of log cycles. The default values for tic divisions are given as follows but will range over whole cycles:

Number of Cycles	Intermediate Values
1, 2	2., 3., 4., 5., 6., 7., 8., 9.
3	2., 3., 5., 7., 9.,
4	2., 4., 6., 8.,
5	2., 5., 8.
6, 7	3., 6.
8, 9, 10	3.

XMAX**Maximum X-Axis Value**

Specifies the maximum value on the x-axis.

Format:

XMAX xmax

Example:

XMAX 100.

Describer	Meaning
xmax	Maximum value on the x-axis. (Real)

Remarks:

1. If XMAX is not specified, then the maximum value is set to the highest value of x.
2. See related commands [XMIN, 696](#), [YMIN, 726](#), and [YMAX, 725](#).

XMIN Minimum X-Axis Value

Specifies the minimum value on the x-axis.

Format:

XMIN xmin

Example:

XMIN 100.

Descriptor	Meaning
xmin	Minimum value on the x-axis (Real).

Remarks:

1. XMIN applies to all curves.
2. If XMIN is not specified, then the minimum value is set to the lowest value of x.
3. See related commands [XMAX, 695](#), [YMIN, 726](#), and [YMAX, 725](#).

XPAPER**Paper Size in X-Direction**

Defines the size of the paper in x-direction.

Format:

XPAPER xszie

Example:

XPAP 10.

Describer	Meaning
xsize	Size of paper in x-direction and in inches (Real; Default = 20.0).

Remarks:

1. The default paper size is 20 by 20 inches.
2. See related command [YPAPER, 727](#).

XTAXIS

X-Axis Plot Control

XYAXIS applies to upper half frame curves only.

Format:

XTAXIS{ YES }
NO

Describers	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis (Default).

XTGRID LINES

Plot X-Axis Grid Lines

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on upper half frame curves only.

Format:

XTGRID LINE { YES }
 { NO }

Describers	Meaning
YES	Plot the x-axis grid lines.
NO	Do not plot the x-axis grid lines (Default).

Remarks:

1. XTGRID applies to upper half frame curves only.
2. See related command [YTGRID LINES, 730](#)

XTITLE

X-Axis Title

Defines a character string that will appear along the x-axis.

Format:

XTITLE xtit

Example:

XTIT RIGHT WING CASE 3 - TIME

Descriptor	Meaning
xtit	Any character string (Character; Default = Blank).

Remarks:

1. XTITLE may not be continued to the next command line.
2. XTITLE applies to all curves.

XYPAPLOT

Generate X-Y Plots for a Printer

Generates X-Y plots for a printer.

See [XYPLLOT, 703](#) for format, describers, and additional remarks.

Remarks:

1. The x-axis moves vertically along the page and the y-axis moves horizontally along the page.
2. An asterisk (*) identifies the points associated with the first curve of a frame, and then for successive curves on a frame the points are designated by symbols O, A, B, C, D, E, F, G, and H.

XYPEAK

Print Curve Summary

Print only the summary for all curves. The summary output is titled:

"X Y - O U T P U T S U M M A R Y"

and is also printed under XYPILOT, XYPUNCH, XYPRINT, and XYPAPLOT. This output contains the maximum and minimum values of y for the range of x.

See [XYPILOT, 703](#) for format, describers, and additional remarks.

XYPLOT**Generate X-Y Plots for a Plotter**

Generates X-Y Plots for a plotter.

Format:

XYPLOT	yvtype ptype [i1, i2, i3,...] / id11 (itemu11 [, iteml11]), id12 (itemu12 [, iteml12]) , ... / id21 (itemu21 [, iteml21]), id22 (itemu22 [, iteml22]) , ... /
--------	--

Examples:

BEGIN BULK or OUTPUT(PLOT) commands are shown as a reminder to place X-Y output request packets properly in the Case Control Section; i.e., at the end of the Case Control Section, or just ahead of any structure plot requests.

Example 1:

```
OUTPUT (XYPLOT)
CSCALE = 1.8
XYPLOT SDISP/16(T1)
BEGIN BULK
```

This sequence causes a single whole frame to be plotted for the T1 displacement component of solution set point 16 using the default parameter values. If 16(T1) is not in the solution set, a warning message will be printed and no plot will be made. The plot will be generated for the NASTRAN plotter on file PLT, which must be available.

Example 2:

```
OUTPUT (XYOUT)
CSCALE = 1.8
XYPLOT, XYPRINT VELO RESPONSE 1,5 /3(R1,), 5(,R1)
OUTPUT (PLOT)
```

This sequence causes two frame plots (each consisting of an upper half frame and a lower half frame) to be plotted, one for subcase 1 and one for subcase 5, using the default parameter values. The velocity of the first rotational component of grid point 3 will be plotted on the upper half frame, and that of grid point 5 will be plotted on the lower half frame. Tabular printer output will also be generated for both curves.

Example 3:

```
OUTPUT (XYPLOT)
CSCALE = 1.8
YDIVISIONS = 20
XDIVISIONS = 10
SGRID LINES = YES
YGRID LINES = YES
XYPLOT DISP 2,5/10(T1),10(T3)
BEGIN BULK
```

This sequence causes two whole frame plots to be generated, one for subcase 2 and one for subcase 5. Each plot contains the T1 and T3 displacement component for grid point 10. The default parameters will be modified to include grid lines in both the x-direction and y-direction, with 10 spaces in the x-direction and 20 spaces in the y-direction. The plot will be generated for the NASTRAN plotter on file .plt.

Example 4

```
OUTPUT (XYPLOT)
CSCALE = 1.8
XAXIS = YES
YAXIS = YES
XPAPER = 40.
YPAPER = 20.
XYPLOT STRESS 3/ 15(2) / 21(7)
OUTPUT (PLOT)
```

This sequence causes two whole frame plots to be generated using the results from subcase 3. The first plot is the response of the axial stress for rod element number 15. The second plot is the response of the major principal stress at Z1 for CTRIA3 element number 21. The default parameters will be modified to include the x-axis and y-axis drawn through the origin. Each plot will be initially scaled to fit on 40 x 20 inch paper. The plots will be generated for the NASPLT postprocessor and NASTRAN file .plt2 which must be defined. NASPLT will redefine the plot to 14 x 7-inch paper (with default options).

Example 5

```
OUTPUT (XYPLOT)
CURVELINESYMBOL = -1
XYPLOT XYPA PLOT VG / 1(G,F) 2(G,F) 3(G,F) 4(G,F)
OUTPUT (PLOT)
```

This sequence is an example of plotting in a flutter analysis for which a split frame plot is made; the upper half is V-g and the lower half is V-f. Data from the first four loops will be plotted. Distinct symbols are used for data from each loop, and no lines are drawn between points (since the flutter analyst must sometimes exercise judgement about which points should be connected). The plots will also be printed in the normal output. These plots will not have all the features of the external plots, but can be very useful in getting a quick picture of the curves.

Example 5

```
XTITLE=EXCITATION FREQUENCY FROM 2.5 TO 250 HERTZ SC 200
YTITLE=FLUID MODE PF AT FLUID POINT 204 FOR NATURAL MODE 2
XYPLOT,XYPEAK FMPF(2) MODE 200/204
```

Example 6

```
YTITLE=EXCITATION FREQUENCY FROM 2.5 TO 250 HERTZ
YTITLE=PSD MPF FOR FLUID GRID 204 FOR NATURAL MODE 2
XYPLOT,XYPEAK FMPF(2) PSDF /204
```

Example 7

```
XYPLOT CSIG PSDF / 4 (2003)
```

This is an example of plotting Composite Stress value of stress quantity given by Item Code 3 for layer with ID 2 of element with ID 4.

This can also be equivalently written as (see Remark 9 for explanation)

`XYPLOT CSIG PSDF / 4 (-2,3)`

Describers	Meaning
yvtype	Type of y-value to be plotted: (Character).
ACCE	Acceleration in the physical set.
BOUT	Slideline contact output.
DISP	Displacement in the physical set.
ELFORCE	Element force.
ENTHALPY	Enthalpy in the physical set.
FLUX	Element heat flux.
FMPF (mode_id or frequency_id)	Fluid mode participation factors. See Remark 8.
GMPF (mode_id or frequency_id, panel_name, panel_grid_id)	Panel grid mode participation factors. See Remark 8.
HDOT	Rate of change of enthalpy in the physical set.
MPCF	Multipoint force of constraint.
LMPF	Load mode participation factors. See Remark 8.
NONLINEAR	Nonlinear applied load.
OLOAD	Applied load.
PMPF (mode_id or frequency_id, panel_name)	Panel mode participation factors. See Remark 8.
PRESSURE	Pressure of fluid-structure body.
SACCE	Acceleration in the solution set.
SDISP	Displacement in the solution set.
SMPF (mode_id or frequency_id)	Structural mode participation factors. See Remark 8.
SPCF	Single-point force of constraint.
STEMP	Temperature in the solution set.

Describers	Meaning
STRAIN	Element strain.
STRESS	Element stress.
SVELO	Velocity in the solution set.
TEMP	Temperature in the physical set.
VECTOR	Displacement in the physical set.
VELO	Velocity in the physical set.
VG	Flutter analysis.
CSIG	Composite Stress
CEPS	Composite Strain
CFAI	Composite Failure Index
CSRS	Composite Strength Ratio
ptype	Plot type defining the meaning of i1, i2, ... etc., idi, itemui, and itemli (Character, Default=“RESPONSE”).
AUTO	Autocorrelation function on whole frame curves only.
FREQ	Frequency—for given excitation frequency plot mode participation versus natural frequency—Oxx2E tables—point plot only. See Remark 8.
MODE	Mode - for given fluid mode plot mode participation versus excitation frequency - Oxx2M tables.
PSDF	Power spectral density function on whole frame curves only.
RESPONSE	Time or frequency in SORT2 format, or grid point identification numbers in SORT1 format (Default) .
SPECTRAL	Response spectrum on whole frame curves only.
i1, i2,...	Subcase identification numbers for ptype=RESPONSE. The list must be specified in ascending order. For ptype=SPECTRAL, the subcase refers to the RECNO in the DTI,SPSEL Bulk Data entry. The list is ignored for ptype=AUTO and PSDF (Integer ≥ 0 , Default is all subcases).

Describers	Meaning
idij	Element, grid, scalar, or extra point identification number for y-value for frame i. For yvtype=VG, idij refers to the loop count of a flutter analysis (Integer > 0).
itemuij, itemlij,	Item code for y-value. itemuij is for upper half or whole itemlij curves on frame i, and itemlij is for lower half curves only on frame i. If itemlij is not specified, then whole frame curves will be plotted with itemuij. itemlij is ignored for ptype=“AUTO”, “PSDF”, and “SPECTRAL” (Character or Integer > 0).
	For elements, the code represents a component of the element stress, strain, or force and is described in Table 7-1 and Table 7-6 of the Guide. For ptype=“AUTO” and “PSDF”, the complex stress or strain item codes need to be used. Since the output quantities are real, you can use either the real or the imaginary item code. Both will give the same result.

For grid points and pty=“RESPONSE”, the code is one of the mnemonics T1, T2, T3, R1, R2, R3, T1RM, T2RM, T3RM, R1RM, R2RM, R3RM, T1IP, T2IP, T3IP, R1IP, R2IP, or R3IP, where Ti stands for the i-th translational component, Ri stands for the i-th rotational component, RM means real or magnitude, and IP means imaginary or phase. For scalar or extra points, or heat transfer analysis, use T1, T1RM, or T1IP. The output format of results in PCH file is dictated by the case control request for DISP, STRESS. For example for SOL111:

```
DISP (PLOT, phase, SORT2) = 1
OUTPUT (XYOUT)
XYPUNCH DISP RESP /3 (T3RM)
```

This will punch results in PCH file in Mag/phase format. If the DISP command is missing results will shift to Real/Imaginary the default format for SOL111 displacement.

For grid points and ptype=“AUTO” or “PSDF”, the code is one of the mnemonics T1, T2, T3, R1, R2, R3. For scalar or extra points, use T1.

For yvtype=VG, itemui and/or itemli can be “F” for frequency or “G” for damping.

Remarks:

1. Multiple XYPILOT, XYPUNCH, XYPRINT, XYPEAK, and/or XYPAPILOT commands may be specified in the OUTPUT(XYPILOT) section.
2. Solution set requests are more efficient, because the time-consuming recovery of the dependent displacements can be avoided.
3. The item codes also appear in printed summaries as “CURVE ID” for grid points as well as element data.

4. The information after each slash (/) specifies the curve(s) that are to be plotted on the same frame. The describer idij identifies the grid point j or element j associated with the frame number i. All plot requests on one command are sorted on idij to improve the efficiency of the plotting process. Symbols are assigned in order by idij.
5. If any of the item codes, itemlij or itemuij, are not specified; e.g., (8) or (5), the corresponding half frame curve is not plotted. If both the comma (,) and itemlij not specified; e.g., (8), then whole frame curves will be plotted. Also, for any single frame, the specifications of “(itemuij,itemlij)” must be consistently half frame (upper and/or lower) or whole frame. For example on half frame curves, if iteml11 and the comma is not specified then either iteml12 or itemu12 must not be specified and on whole frame curves, the commas, iteml11, and iteml12 must not be specified. In other words, the curves on each plot frame must be all whole or half (upper and/or lower).
6. The XYPLOT command may be continued on the next line as long as “XYPLOT yvtype ptype [i1, i2, i3,...] /” is specified on the first line.
7. Specifying a nonexistent grid point may cause the program to exit in the XYTRAN module and missing plots to occur.
8. mode_id is used for natural frequency selection of Oxx2m participation versus excitation frequency output.
frequency_id is used for excitation frequency selection of Oxx2E participation versus natural frequency output. frequency_id is an integer value; e.g., (2) would represent the second frequency calculated.
9. For yvtype = CSIG, CEPS, CFAI and CSRS, we can have (itemuij, itemlij) or (itemuij) formats. For the (itemuij, itemlij) format, itemuij will be a negative integer and its absolute value indicates the Ply ID; itemlij indicates the composite stress item code. For the (itemuij) format, itemuij will be an integer I = PLYID*1000 + ITEM_CODE.
10. Printout in the f06 file coming from X-Y plot commands is not supported by the POST TOCASE command.

XYPRINT**Generate Table of X-Y Pairs for a Printer**

Generates tabular printer output of the X-Y pairs.

See [XYPLOT, 703](#) for format, describers, and additional remarks.

XYPUNCH

Generate Table of X-Y Pairs for the PUNCH File

Generates tabular punch output of the X-Y pairs. Same as XYPRINT except the output is written to the PUNCH file.

See [XYPLOT, 703](#) for format, describers, and additional remarks.

XVALUE PRINT SKIP**Print Values on X-Axis Tic Marks**

Specifies how often to print the x-values alongside the x-axis tic marks.

Format:

XVALUE PRINT SKIP xvps

Example:

XVAL 5

Descriptor	Meaning
xvps	Number of tic marks to be skipped between labeled tic marks with their corresponding values (Integer ≥ 0).

Remark:

1. XVALUE applies to all curves.

YAXIS

Y Axis Plot Control

Controls the plotting of the y-axis on all curves.

Format:

YAXIS{ YES }
NO }

Describers	Meaning
YES	Plot the y-axis.
NO	Do not plot the y-axis (Default).

YBDIVISIONS**Tic Spacing on Y-Axis**

Specifies spacing of tic marks on the y-axis for lower half frame curves only.

Format:

YBDIVISIONS ybd

Example:

YBDI 10

Descriptor	Meaning
ybd	Number of spaces between tic marks on y-axis (Integer > 0; Default = 5).

Remarks:

1. YBDIVISIONS applies to lower half frame curves only.
2. YBDIVISIONS is ignored for a logarithmic y-axis.

YBINTERCEPT

Location of Y Axis on X Axis

Specifies the location of the y-axis on the x-axis for lower half frame curves only.

Format:

YBINTERCEPT ybi

Example:

YBINT 50

Descriptor	Meaning
ybi	Location of y-axis on the x-axis (Real; Default = 0.0).

Remark:

1. YBINTERCEPT applies to lower half frame curves only.

YBGRID LINES

Plot Y Axis Grid Lines

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on lower half frame curves only.

Format:

YBGRID LINES { YES }
 { NO }

Describers	Meaning
YES	Plot the y-axis grid lines.
NO	Do not plot the y-axis grid lines (Default).

Remarks:

1. YBGRID applies to lower half frame curves only.
2. See related command [XBGRID LINES, 690](#)

YBLOG**Logarithmic or Linear Y Axis**

Selects logarithmic or linear y-axis for lower half frame curves only.

Format:

YBLOG{ YES }
 | NO |

Describers	Meaning
YES	Plot a logarithmic y-axis.
NO	Plot a linear y-axis (Default).

Remarks:

1. YBLOG applies to lower half frame curves only.
2. See Remark 2 under [XLOG, 694](#).

YBMAX**Maximum Y Axis Value**

Specifies the maximum value on the y-axis for lower half frame curves only.

Format:

YBMAX ymax

Example:

YBMAX 100

Describer	Meaning
ymax	Maximum value on the y-axis (Real).

Remarks:

1. YBMAX applies to lower half frame curves only.
2. If YBMAX is not specified, then the maximum value is set to the highest value of y.
3. See related command [YBMIN, 718](#).

YBMIN

Minimum Y Axis Value

Specifies the minimum value on the y-axis for lower half frame curves only.

Format:

YBMIN ymin

Example:

YBMIN 100

Descriptor	Meaning
ymin	Minimum value on the y-axis (Real).

Remarks:

1. YBMIN applies to lower half frame curves only.
2. If YBMIN is not specified then the minimum value is set to the lowest value of y.
3. See related command [YBMAX, 717](#).

YBTITLE

Y-Axis Title

Defines a character string that will appear along the y-axis for lower half frame curves only.

Format:

YBTITLE ytit

Example:

YBTIT RIGHT WING LOADS - CASE 3

Descriptor	Meaning
ytit	Any character string (Character; Default= Blank).

Remarks:

1. YBTITLE may not be continued to the next command line.
2. YBTITLE applies to lower half frame curves only.

YBVALUE PRINT SKIP

Print Values on Y Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks applies on lower half frame curves only.

Format:

YBVALUE PRINT SKIP yvps

Example:

YBVAL 5

Descriptor	Meaning
yvps	Number of tic marks to be skipped between labeled tic marks with their corresponding values (Integer ≥ 0).

Remark:

1. YBVALUE applies to lower half frame curves only.

YDIVISIONS**Tic Spacing on Y Axis**

Specifies spacing of tic marks on the y-axis for whole frame curves only.

Format:

YDIVISIONS *yd*

Example:

YDIV 10

Descriptor	Meaning
yd	Number of spaces between tic marks on y-axis (Integer > 0; Default = 5).

Remarks:

1. YDIVISIONS applies to whole frame curves only and to the commands: LEFT TICS, RIGHT TICS, and XINTERCEPT.
2. YDIVISIONS is ignored for a logarithmic y-axis.

YINTERCEPT

Location of Y Axis on X Axis

Specifies the location of the y-axis on the x-axis for whole frame curves only.

Format:

YINTERCEPT yi

Example:

YINT 50

Descriptor	Meaning
yi	Location of y-axis on the x-axis. (Real; Default = 0.0)

Remark:

1. YINTERCEPT applies to lower half frame curves only.

YGRID LINES

Plot Y Axis Grid Lines

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on whole frame curves only.

Format:

YGRID LINES { YES }
 { NO }

Describers	Meaning
YES	Plot the y-axis grid lines.
NO	Do not plot the y-axis grid lines (Default).

Remarks:

1. YGRID applies to whole frame curves only.
2. See related command [XGRID LINES, 692](#)

YLOG**Logarithmic or Linear Y Axis**

Selects logarithmic or linear y-axis for whole frame curves only.

Format:

YLOG{ YES }
NO }

Describers	Meaning
YES	Plot a logarithmic y-axis.
NO	Plot a linear y-axis (Default).

Remarks:

1. YLOG applies to whole frame curves only.
2. See Remark 2 under [XLOG, 694](#).

YMAX**Maximum Y Axis Value**

Specifies the maximum value on the y-axis.

Format:

YMAX ymax

Example:

YMAX 100

Describer	Meaning
ymax	Maximum value on the y-axis (Real).

Remarks:

1. If YMAX is not specified, then the maximum value is set to the highest value of y.
2. See related command [YMIN, 726](#).

YMIN Minimum Y Axis Value

Specifies the minimum value on the y-axis.

Format:

YMIN ymin

Example:

YMIN 100

Descriptor	Meaning
ymin	Minimum value on the y-axis (Real).

Remarks:

1. YMIN applies to all curves.
2. If YMIN is not specified, then the minimum value is set to the lowest value of y.
3. See related command [YMAX, 725](#).

YPAPER**Paper Size in Y-Direction**

Defines the size of the paper in y-direction.

Format:

YPAPER ysize

Example:

YPAP 10

Descriptor	Meaning
ysize	Size of paper in y-direction and in inches (Real; Default = 20.0).

Remarks:

1. The default paper size is 20 by 20 inches.
2. See related command [XPAPER, 697](#).

YTAXIS

Y Axis Plot Control

Controls the plotting of the y-axis on upper half frame curves only.

Format:

YTAXIS{ YES
 NO }

Describers	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis (Default).

Remark:

1. YTAXIS applies to upper half frame curves only.

YTDIVISIONS

The Spacing on Y Axis

Specifies spacing of tic marks on the y-axis for upper half frame curves only.

Format:

YTDIVISIONS ytd

Example:

YTDI 10

Descriptor	Meaning
ytd	Number of spaces between tic marks on y-axis (Integer > 0; Default = 5).

Remarks:

1. YTDIVISIONS applies to upper half frame curves only.
2. YTDIVISIONS is ignored for a logarithmic y-axis.

YTGRID LINES**Plot Y Axis Grid Lines**

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on upper half frame curves only.

Format:

TYGRID LINES { YES }
 NO

Describers	Meaning
YES	Plot the y-axis grid lines.
NO	Do not plot the y-axis grid lines (Default).

Remarks:

1. YTGRID applies to upper half frame curves only.
2. See related command [XTGRID LINES, 699](#)

YTINTERCEPT**Location of Y Axis on X Axis**

Specifies the location of the y-axis on the x-axis for upper half frame curves only.

Format:

YTINTERCEPT yti

Example:

YTINT 50

Describer	Meaning
yti	Location of y-axis on the x-axis (Real; Default= 0.0).

Remark:

1. YTINTERCEPT applies to upper half frame curves only.

YTITLE Y Axis Title

Defines a character string that will appear along the y-axis for whole frame curves only.

Format:

YTITLE ytit

Example:

YTIT RIGHT WING LOADS - CASE 3

Descriptor	Meaning
ytit	Any character string (Character; Default = Blank).

Remarks:

1. YTITLE may not be continued to the next command line.
2. YTITLE applies to whole frame curves only.

YTLOG**Logarithmic or Linear Y Axis**

Selects logarithmic or linear y-axis for upper half frame curves only.

Format:

YT(LOG){ YES }
NO }

Describers	Meaning
YES	Plot a logarithmic y-axis.
NO	Plot a linear y-axis (Default).

Remarks:

1. YTLOG applies to upper half frame curves only.
2. See Remark 2 under [XLOG, 694](#).

YTMAX

Maximum Y Axis Value

Specifies the maximum value on the y-axis for upper half frame curves only.

Format:

YTMAX ymax

Example:

YTMAX 100

Describer	Meaning
ymax	Maximum value on the y-axis (Real).

Remarks:

1. YTMAX applies to upper half frame curves only.
2. If YTMAX is not specified, then the maximum value is set to the highest value of y.
3. See related command [YTMIN, 735](#).

YTMIN**Minimum Y Axis Value**

Specifies the minimum value on the y-axis for upper half frame curves only.

Format:

YTMIN ymin

Example:

YTMIN 100

Describer	Meaning
ymin	Minimum value on the y-axis (Real).

Remarks:

1. YTMIN applies to upper half frame curves only.
2. If YTMIN is not specified then the minimum value is set to the lowest value of y.
3. See related command [YTMAX, 734](#).

YTTITLE

Y-Axis Title

Defines a character string that will appear along the y-axis for upper half frame curves only.

Format:

YTTITLE ytit

Example:

YTTIT RIGHT WING LOADS - CASE 3

Descriptor	Meaning
ytit	Any character string (Character; Default = Blank).

Remarks:

1. YTTITLE may not be continued to the next command line.
2. YTTITLE applies to upper half frame curves only.

YTVALUE PRINT SKIP

Print Values on Y Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks applies on upper half frame curves only.

Format:

YTVALUE PRINT SKIP yvps

Example:

YTVAL 5

Describer	Meaning
yvps	Number of tic marks to be skipped between labeled tic marks with their corresponding values (Integer ≥ 0).

Remark:

1. YTVALUE applies to upper half frame curves only.

YVALUE PRINT SKIP

Print Values on Y Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks applies on whole frame curves only.

Format:

YVALUE PRINT SKIP yvps

Example:

YVAL 5

Descriptor	Meaning
yvps	Number of tic marks to be skipped between labeled tic marks with their corresponding values (Integer ≥ 0).

Remark:

1. YVALUE applies to whole frame curves only.

OUTPUT(POST) Commands

SURFACE

Surface Definition

Defines a surface for the calculation of grid point stresses, strains, or mesh stress discontinuities.

Format:

```
SURFACE id SET sid, [FIBRE {ALL  
Z1  
Z2  
MID}],  
[SYSTEM {ELEMENT  
BASIC  
CORD cid}], [AXIS {X1  
X2  
X3}], [NORMAL [M]{R  
X1  
X2  
X3}],  
[TOPOLOGICAL]  
[GEOMETRIC] [TOLERANCE{THETA}], [BRANCH {[MESSAGE  
NOMESSAGE]}], [BREAK]  
[NOBREAK]]]
```

Example:

SURFACE 10 SET 9 NORMAL X3

Descriptor	Meaning
id	Surface identification number (Required).
SET	References a SET command that defines the elements in the surface (required). Either form of the SET command may be used.
sid	Set identification number.
FIBRE	Specifies the fiber location at which stresses will be calculated.
ALL	Requests output at all fiber locations; i.e., z=Z1, Z2, and MID.
Z1	Requests output z = Z1 only.
Z2	Requests output z = Z2 only.
MID	Requests output z = (Z1+Z2)/2 only.
SYSTEM	Specifies the coordinate system to be used as the output coordinate system.
ELEMENT	Specifies the element coordinate system for output.
CORD cid	Specifies the coordinate system defined on a CORDij Bulk Data entry for output.
BASIC	Specifies the basic coordinate system for output.
AXIS	Specifies the axis of the coordinate system to be used as the x output axis and the local x-axis when geometric interpolation is used.

Descriptor	Meaning
X1, X2, X3	Specifies the direction of the axis or the normal. X, Y, and Z may be substituted for X1, X2, and X3, respectively.
NORMAL	Specifies the reference direction for positive fiber and shear stress output, but has no effect when ELEMENT is specified.
M	Specifies the reverse of the direction given by R, X1, X2, or X3 and must be entered as MR, MX1, MX2, or MX3 with no space between the M and the following letter.
R	Specifies the radius vector from the origin of reference coordinate system to the grid point.
TOPOLOGICAL GEOMETRIC	Specifies the method to calculate the average grid point stress or strain. The default is TOPOLOGICAL.
theta	Specifies the interelement slope difference tolerance (in degrees) to detect stress discontinuity between elements (not used with TOPOLOGICAL) (Real; Default = 0.0).
BRANCH	Selects whether multiple element intersections (BREAK/NOBREAK) are to be treated as discontinuities, and if warning messages (MESSAGE/NOMESSAGE) are to be issued.
BREAK NOBREAK	Multiple element intersections are (or are not) to be treated as stress discontinuities.
MESSAGE NOMESSAGE	A warning message will (or will not) be issued when multiple element intersections are encountered.

Remarks:

1. SURFACE commands must be specified after OUTPUT(POST).
2. The surface identification number must be referenced on a SET command appearing after OUTPUT(POST). The SET identification number may then be referenced on GPSTRESS, GPSTRAIN, STRFIELD, ELSDCON, and GPSDCON commands. The sid on the surface entry must reference a SET defined after OUTPUT(POST).
3. The surface normal is also used in the definition of the local reference surface for geometric interpolation. Two options are available. In the first option, the radius vector (R) from the origin of the reference coordinate system to the grid point is used. In the second option, one axis (X1, X2, or X3) of the coordinate system is used. The direction can be reversed using the modification parameter M. The positive side of an element is defined as a side from which the NORMAL direction emerges, rather than the side determined by the connection specified on the element connection entries.
4. When the parameter ELEMENT is present, the element stresses or strains are used unmodified (defaults to output stresses in output system). The CORD keyword references a CORDij Bulk Data entry with coordinate system identification number cid.

5. When theta = 0, no testing is made. When theta is negative, grid point stresses will be calculated for each element connected to an exception point; otherwise, the best estimation of the grid point stress will be output.
6. BREAK is the default if theta is nonzero.
7. For all elements defined in SET 9 of the preceding example,
 - All fiber locations are output;
 - The basic output system is used;
 - The x-axis is x-axis of the basic system;
 - The surface normal direction point is z-axis of the basic system;
 - The topological interpolation method is used;
 - No tolerance test is made; and
 - No branch test is made.

The example illustrates a good choice for regular two-dimensional problems in the x-y plane.

VOLUME**Volume Definition**

Defines a volume for the calculation of grid point stresses, strains, or mesh stress discontinuities.

Format:

VOLUME id SET sid, [PRINCIPAL, DIRECT STRESS], **SYSTEM** { ELEMENT
CORD cid
BASIC }

Example:

VOLUME 21 SET 2

Descriptor	Meaning
id	Volume identification number.
sid	Set identification number of a SET command that defines the elements in the volume. Either form of the SET command may be used. The default is all elements.
PRINCIPAL	Requests principal stresses or strains, direction cosines, mean pressure, and von Mises equivalent stresses or strains to be calculated. If neither PRINCIPAL nor DIRECT is specified, then the default is to output both.
DIRECT	Requests direct stress or strains, mean pressure stress, and von Mises equivalent stress to be calculated. If neither PRINCIPAL nor DIRECT is specified, then the default is to output both.
SYSTEM	Used to specify the reference coordinate system used to define the output stress orientation coordinate system.
ELEMENT	Specifies the element coordinate system.
CORD cid	Specifies the coordinate system defined on a CORDij entry.
BASIC	Specifies the basic coordinate system.

Remarks:

1. VOLUME commands must be specified after OUTPUT(POST).
2. The volume identification number must be referenced on a SET command appearing after OUTPUT(POST). The SET identification number may then be referenced on GPSTRESS, GPSTRAIN, STRFIELD, ELSDCON, and GPSDCON Case Control commands.
3. If ELEMENT is specified, element stresses or strains are not transformed.
4. In the preceding example, for all elements in SET 2:
 - Both PRINCIPAL and DIRECT stress are output.
 - The BASIC output system is used.

OUTPUT(PLOT) Commands

The PLOT command requests the generation of undeformed, deformed, or contour plots. All other commands specify how the model will be plotted, type of projection, view angles, scales, etc. All commands have default actions if not specified by the user. The FIND command may be used to calculate an optimal SCALE, ORIGIN, and/or VANTAGE POINT to allow the construction of a plot in a user-specified region of the paper or film. All the commands used in the generation of the various plots will be printed out as part of the output, whether they are directly specified, defaulted or established using the FIND command. Initialization of commands to default values occurs only once. Subsequently, these values remain until altered by direct command input. The only exceptions are the view angles, scale factors, vantage points, and the origins. Whenever the plotter or the method of projection is changed, the view angles are reset to the default values, unless they are respecified by the user. In addition, the scale factors, vantage points, and the origin must be respecified by the user.

The commands are listed here in a logical sequence; however, they need not be so specified. Any order may be used, but if a command is specified more than once, the value or choice stated last will be used.

PLOTTER	Selects format of plot file for interpretation by plotter postprocessor.
ORTHOGRAPHIC, etc.	Selects orthographic projection.
PERSPECTIVE	Selects perspective projection.
STEREOSCOPIC	Selects stereoscopic projection.
AXES	Assigns axes of the basic coordinate system to the observer's coordinate system.
VIEW	Defines the angular relationship between the observer's coordinate system (r , s , and t axes specified on the AXES command) and the basic coordinate system.
MAXIMUM DEFORM	Defines the magnification of the maximum displacement.
SCALE	Defines reduction, as a scale factor, of the model's dimensions so that the model fits on a plot frame.
DISTORTION	Specifies the distortion scale factors of axes in the basic coordinate system.
CSCALE	Defines scale factor for characters in the plot frame.
ORIGIN	Defines the origin of the plot frame with respect to the origin of the (r , s , t) coordinate system defined on the AXES command.
VANTAGE POINT	Defines the location of the observer with respect to the model in the (r , s , t) coordinate system defined on the AXES command for perspective and stereoscopic projections only.
PROJECTION	Defines the separation, along the r -axis, between the observer and the projection plane if not already specified on the VANTAGE POINT command. Used by stereoscopic projections only.
OCULAR SEPARATION	Defines the separation of the left and right eye vantage points along the s -axis for stereoscopic projections.
CAMERA	Specifies microfilm plotter options.

PAPER SIZE	Defines the size and type of the paper.
PEN	Generates a message on the printed output to inform the plotter operator as to what size and color pen point to mount in the various pen holders.
PTITLE	Defines a character string that will appear at the top of the plot frame on the line below the sequence number.
SET	Defines a set of elements and/or grid points to be plotted.
FIND	Requests the plotter to optimally compute any of the parameters that can be specified on the SCALE, ORIGIN i, and/or VANTAGE POINT commands.
CONTOUR	Specifies contour plot options for stress, displacement, or temperature.
PLOT	Generates an undeformed plot or a deformed plot per subcase, mode number, frequency, or time step. A contour plot may also be requested with an undeformed or deformed plot.

AXES**Orientation of Observer's Coordinate System**

Assigns axes of the basic coordinate system to the observer's coordinate system.

Format:

AXES r s t [SYMMETRIC
ANTISYMMETRIC]

Example 1: View toward negative x-axis of model.

AXES MX, Y, MZ

Example 2: Mirror image of model.

AXES Y X Z

Describers	Meaning
r,s,t	Assigns axes of basic coordinate system to axes of observer's coordinate system (Default=X, Y, Z). X X-direction of basic coordinate system (Default for r). Y Y-direction of basic coordinate system (Default for s). Z Z-direction of basic coordinate system (Default for t). MX Negative X-direction of basic coordinate system. MY Negative Y-direction of basic coordinate system. MZ Negative Z-direction of basic coordinate system.
SYMMETRIC	Specifies a symmetric orientation of the view. See Remark 5. (Default).
ANTISYMMETRIC	Specifies an antisymmetric orientation of the view. See Remark 5.

Remarks:

1. If no AXES command is specified, then AXES X, Y, Z is the default.
2. The direction of view is in the negative r-direction; i.e., the projection plane is parallel to the s-t plane.
3. The VIEW command depends on the AXES command specification and defines the angular relationship between observer's coordinate system and the basic coordinate system.
4. The AXES command can be used to preposition the object in 90° increments in such a manner that only rotations less than 90° are required by the VIEW command to obtain the desired orientation. MX, MY, MZ can be used to define left-handed coordinate systems. Note that the default system is right-handed.
5. An undeformed or deformed plot of the symmetric portion of an object can be obtained by reversing the sign of the axis that is normal to the plane of symmetry. In the case of multiple planes of symmetry, the signs of all associated planes should be reversed. The ANTISYMMETRIC option should be specified when a symmetric model is loaded in an unsymmetric manner. This will cause

the deformations to be plotted antisymmetrically with respect to the specified plane or planes. Since the AXES command applies to all parts (SETs) of a single frame, symmetric and antisymmetric combinations cannot be made with this command (see the symmetry option on the [PLOT, 761](#) command in this section).

6. To avoid a mirror image, ensure that the r, s, and t axes obey the right-hand rule.

CAMERA**Microfilm Plotter Options**

Specifies microfilm plotter options.

Format:

CAMERA [PAPER
FILM BLANK FRAME n
BOTH]

Example:

CAMERA FILM

Describers	Meaning
FILM	Requests 35 mm or 16 mm film and positive or negative images.
PAPER	Requests positive prints.
BOTH	Requests positive prints and 35 mm or 16 mm film.

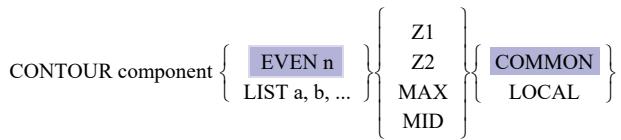
Remarks:

1. If the CAMERA command is not specified, then CAMERA PAPER BLANK FRAMES 0 is assumed.
2. If FILM or BOTH is specified, then these options must be communicated to the plotter operator through normal means of communications at the installation.
3. If FILM or BOTH are specified and if n is greater than 0 then n blank frames will be inserted between plots. The plotter must be operated in the manual mode in order to have blank frames inserted between positive prints. If blank frames are desired only on film, and not on paper, the plotter must be operated in the automatic mode.

CONTOUR**Contour Plot Options**

Specifies contour plot options for stress, displacement, or temperature.

Format:



Example:

CONTOUR MAGNIT LIST 2., 4., 6., 8., 10.

Descriptors	Meaning
component	Name of stress, displacement or temperature component (Character, Default=“MAJPRIN”).
MAJPRIN	Major principal stress. Not available for nonlinear elements (Default).
MINPRIN	Minor principal stress. Not available for nonlinear elements.
EQUIVAL	von Mises stress. When STRESS(MAXS) is requested in the Case Control Section, von Mises stress is used for plotting. For nonlinear analysis, Mohr-Coulomb or Drucker-Prager stress may also be plotted in conjunction with the MATS1 command.
XNORMAL	X component of normal stress.
YNORMAL	Y component of normal stress.
ZNORMAL	Z component of normal stress.
XYSHEAR	XY component of shear stress.
XZSHEAR	XZ component of shear stress.
YZSHEAR	YZ component of shear stress.
XDISP	T1 component of displacement in global coordinate system.
YDISP	T2 component of displacement in global coordinate system.
ZDISP	T3 component of displacement in global coordinate system.
MAGNIT	Magnitude of displacement or temperature.
EVEN n	Number of contours ($50 \geq \text{Integer} > 0$, Default is EVEN 10).
LIST a, b, ...	List of stresses, displacements or temperatures which define the contours (Real).
Z1	Stresses at Z1 from neutral plane (Default).

Describers	Meaning
Z2	Stresses at Z2 from neutral plane.
MAX	Maximum of stress at Z1 and Z2.
MID	Average of stress (membrane stress) at Z1 and Z2.
COMMON	Plot stress contours in basic coordinate system (Default).
LOCAL	Plot stress contours in element coordinate system. This is the coordinate system used in printed output.

Remarks:

1. The CONTOUR command should be specified immediately before its associated PLOT command.
2. A STRESS command must appear in the Case Control Section for all elements included in a CONTOUR request. If printed output is not desired, then STRESS(PLOT)=sid should be specified.
3. In linear analysis, stress contour plots are available for the following elements: CTRIA3, CQUAD4, CSHEAR, and CTRIA6. In nonlinear analysis, stress contour plots are available for CQUAD4 and CTRIA3 elements. The Bulk Data element connection entries for all elements must list the grid points in either clockwise or counterclockwise order. Mixing the order will result in meaningless or confusing plots.
4. When selecting contour options, note that
 - MAJPRIN, MINPRIN, EQUIVAL are the same in COMMON and LOCAL.
 - ZNORMAL, XZSHEAR, YZSHEAR, if selected in LOCAL, will be changed to COMMON.
 - CSHEAR elements only have the MAXSHEAR value.
5. The CTRIA6 element stress contour plots are different in that they must be selected as COMMON. Also, the following equivalences apply:

XNORMAL is radial
 YNORMAL is azimuthal
 ZNORMAL is axial
 XYSHEAR is shear
 XZSHEAR is maximum principal
 YZSHEAR is von Mises
 EQUIVAL is octahedra

CSCALE

Character Scale Factor

Defines scale factor for characters in the plot frame.

Format:

CSCALE cs

Example:

CSCA 2.0

Descriptor	Meaning
cs	Scale factor applied to characters in the plot frame (Default = .5).

Remarks:

1. CSCALE is used to control the spacing of characters when plots are made with the NASTRAN plotter and postprocessed with the MSC/NASPLOT routine. For example, if the SCALE FACTOR on the NASPLOT data command is 2.0, a value for cs of 0.5 will result in characters of default size (.07 inches) at the regular spacing. A value of 1.8 produces good spacing when using the postprocessing plotter programs NASTPLT, TEKPLT, and NEUPS. On the other hand, to double the size of both the plot and the characters, the SCALE FACTOR and the CSCALE FACTOR on the NASPLOT data command should both be set equal to 2.0.
2. The CSCALE command must immediately precede the PLOTTER command. If a second CSCALE command is specified, then a second PLOTTER command must also be specified.

DISTORTION

Distortion Scale Factors

Specifies the distortion scale factors of the axes in the basic coordinate system.

Format:

DISTORTION dx dy dz

Example:

DIST 0.5 1.0 1.0

Describers	Meaning
dx	Distortion scale factor of the basic coordinate system's x-axis (Default=1.0).
dy	Distortion scale factor of the basic coordinate system's y-axis (Default=1.0).
dz	Distortion scale factor of the basic coordinate system's z-axis (Default=1.0).

Remarks:

1. If no DISTORTION command is specified, then no distortion is applied.
2. If DISTORTION is specified, then all three values for dx, dy, and dz must be specified even though one or two will use the default.
3. The distortion factors are applied prior to any other scaling commands: SCALE, MAXIMUM DEFORMATION, CSCALE, etc.

FIND**Automatic Calculation of Scale, Origin, and Vantage Point**

Requests the plotter to optimally compute any of the parameters that can be specified on the SCALE, ORIGIN i, and/or VANTAGE POINT commands.

Format:

```
FIND [ SCALE ORIGIN oid VANTAGE POINT SET setid ,
      REGION { le be re te } ]
```

Example:

```
FIND SCALE ORIGIN 5 SET 2
```

Describers	Meaning
oid	Origin identification number (Integer>0).
setid	Set identification number etc. (Integer>0).
le	Fractional distance of left edge of plot region from the lower left corner of the image area (Real, Default=0.0).
be	Fractional distance of bottom edge of plot region from the lower left corner of the image area (Real, Default=0.0).
re	Fractional distance of right edge of plot region from the lower left corner of the image area (Real, Default=1.0).
te	Fractional distance of top edge of plot region from the lower left corner of the image area (Real, Default=1.0).

Remarks:

1. The FIND command is recommended over the specification of SCALE, ORIGIN, and VANTAGE POINT commands, and should be specified prior to its associated PLOT or CONTOUR command.
2. The FIND command requests the plotter to optimally compute any of the parameters that can be specified on the SCALE, ORIGIN, and/or VANTAGE POINT commands and based on the specification of the
 - PLOTTER command;
 - PROJECTION PLANE command;
 - SET and REGION specifications on the FIND command;
 - VIEW and/or AXES commands;
 - MAXIMUM DEFORMATION command; and
 - PAPER SIZE command. All of these commands must precede the associated FIND command.
3. The FIND command can be used to compute any or all of SCALE, ORIGIN, or VANTAGE POINT as long as they have not been previously specified.
4. If SET is not specified, then the first defined SET will be used.

5. If no options are specified on the FIND command, a SCALE and VANTAGE POINT are selected and an ORIGIN is located, using the first defined SET, so that the plotter object is located within the image area.
6. The plot region is defined as some fraction of the image area (image area = 0.0, 0.0, 1.0, 1.0 and first quadrant = 0.5, 0.5, 1.0, 1.0). The image area is located inside the margins on the paper.

MAXIMUM DEFORM

Length of Maximum Displacement

Defines the magnification of the maximum displacement. All other displacements are scaled accordingly.

Format:

MAXIMUM DEFORMATION d

Example:

Magnify the displacements such that the maximum displacement is equal to two units of length of the model.

MAXI DEFO 2.

Descriptor	Meaning
d	Specifies the length, in units of the model and not of the plot frame, to which the maximum displacement is scaled. (Default=5% of the largest dimension of the model represented by the elements in the SET specification on the PLOT command.)

Remarks:

1. If no MAXIMUM DEFORMATION command is specified, then the previously described default is assumed.
2. If you wish the FIND command to use the d value, a MAXIMUM DEFORMATION command should precede the FIND command.
3. If you wish the plot deformation scaling to be different than the FIND scaling, a different MAXIMUM DEFORMATION command can appear first before the PLOT command.
4. For nonlinear plotting, MAXIMUM DEFORMATION d and the MAXIMUM DEFORMATION field on the PLOT command should have the same value.

OCULAR SEPARATION

Separation of the Vantage Points

Defines the separation of the left and right eye vantage points along the s-axis for stereoscopic projections.

Format:

OCULAR SEPARATION os

Example:

OCULAR SEPARATION 2.0

Descriptor	Meaning
os	Separation, in inches, of the two vantage points along the s-axis (Real, Default=2.756 inches).

Remark:

1. The default value is the separation used in the standard stereoscopic cameras and viewers (70 mm). The default value is recommended.

ORIGIN

Origin of Plot Frame

Defines the origin of the plot frame with respect to the origin of the (r, s, t) coordinate system defined on the AXES command.

Format:

ORIGIN oid u v

Example:

ORIG 3 -1. -2.

Descriptors	Meaning
oid	Origin identification number which may be specified after the ORIGIN descriptor on the PLOT command (Integer > 0).
u	Horizontal displacement of plot frame origin from the rst origin (Real, Default = 0.0).
v	Vertical displacement of paper origin from the rst origin (Real, Default = 0.0).

Remarks:

1. In the transformation performed for any of the three projections, the origins of both the basic coordinate system and the observer's coordinate system are coincident. The ORIGIN command may be used to locate the plot frame origin (lower left hand corner) from the rst origin. The units are inches, and are not subject to the scaling of the plotted object.
2. The ORIGIN command is not recommended for general use. See the [FIND, 752](#) command to have the origin optimally located so as to place the plotted object in the center of the plot frame.
3. Ten origins may be specified at one time. However, any one can be redefined at any time. An eleventh origin is also provided if more than ten origins are erroneously defined; i.e., only the last of these surplus origins will be retained.
4. If a projection; e.g., ORTHOGRAPHIC, STEREOSCOPIC, or PERSPECTIVE, is changed in the plot packet, or the PLOTTER command is changed, then all previously defined origins are deleted.

ORTHOGRAPHIC, etc.**Type of Projection**

Selects type of projection.

Format:

[ORTHOGRAPHIC]
PERSPECTIVE
STEREOSCOPIC]

Describers	Meaning
ORTHOGRAPHIC	Selects orthographic projection (Default).
PERSPECTIVE	Selects perspective projection.
STEREOSCOPIC	Selects stereoscopic projection.

Remark:

1. If none of the preceding projections are specified, then ORTHOGRAPHIC is used.

PAPER SIZE

Paper Dimensions

Defines the size and type of the paper.

Format:

PAPER SIZE h X or BY v [TYPE ptype]

Example:

PAPER SIZE 10. BY 10.

Describers	Meaning
h	Horizontal size of paper in inches (Real, Default = 20.0).
v	Vertical size of paper in inches (Real, Default= 20.0).
ptype	Paper type (Character, Default = "VELLUM").

Remarks:

1. The default paper size for the PLOTTER NAST is 20 by 20 inches which is converted to a 7 by 7 inch plot frame by the NASPLOT postprocessor.
2. PAPER SIZE can be specified along with the NASPLOT postprocessor to create rectangular plots. For example, the command will result in a 14 by 7 inch plot frame if the default value of 1.0 is used for the SCALE FACTOR on the NASPLOT command. The SCALE FACTOR on the NASPLOT data command can be used to make larger plots having the shape defined with PAPER SIZE.
3. PAPER SIZE also affects the raster count for the NASTRAN plotter. The default raster count is 1000 for a paper size of 20 by 20. Doubling the paper size to 40 by 40 will double the raster count to 2000.

PEN Pen Color and Size Assignments

Generates a message on the printed output which may be used to inform the plotter operator as to what size and color pen point to mount in the various pen holders.

Format:

PEN pn [COLOR cname]

Example:

PEN 2 COLOR RED

Describers	Meaning
pn	Pen identification number (Integer > 0).
COLOR	Flag indicating the next word is a color name.
cname	Pen color (Character).

Remarks:

1. The actual number of pens available will depend on the plotter hardware configuration at each installation.
2. The PEN command does not control the pen used in generating the plot. See the PEN describer on the [PLOT, 761](#) command.
3. The PEN command is optional and is not appropriate for microfilm plotters.

PERSPECTIVE

Selects Perspective Projection

See the description of the [ORTHOGRAPHIC, etc., 757](#).

PLOT**Undeformed or Deformed Plot Request**

Generates an undeformed plot of the model or a deformed plot for a subcase, mode number or time step.

Format:

```
PLOT [analysis][dtype][CONTOUR][i1, i2 THRU i3, i4, etc.]
      [ { RANGE f1, f2 } ]
      [ { TIME t1, t2 } ] ,
```


$$\left[\begin{array}{c} \text{PHASE LAG } \phi \\ \text{MAGNITUDE} \end{array} \right] [\text{MAXIMUM DEFORMATION } d] ,$$

$$[\text{SET sid1}] [\text{ORIGIN oid1}] \left[\begin{array}{c} \text{SYMMETRY} \\ \text{ANTISYMMETRY} \end{array} \right]_w \left[\begin{array}{c} \text{PEN} \\ \text{DENSITY} \end{array} \right]_p [\text{SYMBOLS m[,n]]} ,$$

$$[\text{LABEL label}] \left[\begin{array}{c} \text{SHAPE} \\ \text{OUTLINE} \end{array} \right] [\text{VECTOR v}] , [\text{PRINT}] ,$$

$$[\text{SHRINK t, o}] [\text{NORMALS}] ,$$

$$[\text{SET sid2}] [\text{ORIGIN oid2}] \text{ etc.}$$

Examples:

See after Remarks Section.

Describers	Meaning
analysis	Type of analysis results (Character, default results in an undeformed plot or undeformed underlay for contour plots). STATIC Plot static deformations. MODAL Plot mode shapes or eigenvectors. CMODAL Plot complex mode shapes or eigenvectors. TRANSIENT Plot transient solutions. FREQUENCY Plot frequency solutions. SENOMOVE Plot undeformed superelements in their original position; i.e., ignore SELOC Bulk Data entry.
dtype	Specifies plot quantity (Character, Default = "DEFORMATION"). DEFORMATI ON Plot displacements or temperatures in the Z direction (Default). VELOCITY Plot velocities. ACCELERATI ON Plot accelerations.

Describers	Meaning
CONTOUR	Request for contour plot.
i1, i2,...	Subcase identification numbers. See SHAPE and VECTOR for use of "0" (underlay) command. See Remark 3. (Integer ≥ 0 , Default is all subcases).
RANGE f1 f2	Specifies range of natural frequencies, eigenvalues, excitation
TIME t1,t2	frequencies, time steps, or load factors. Used to minimize the amount of plotted data. See Remark 4. (Real).
PHASE LAG ϕ	Specified phase lag, in degrees, for plotting complex quantities. See Remark 5. (Real, Default = 0.0).
MAGNITUDE	Requests magnitude of complex quantities.
MAXIMUM DEFORMATION d	Specifies the magnification of the maximum displacement. See Remark 6. (Real).
SET sid	Set identification number which defines the set of elements or grid points to be plotted (Default is first SET command).
ORIGIN oid	Origin identification number (Default is first origin defined by the ORIGIN or FIND command).
SYMMETRY w ANTISYMMETRY w	Request plot of the symmetric portion of the symmetrically or antisymmetrically loaded model. This symmetric portion will be located in the space adjacent to the region originally defined by ORIGIN oid, and will appear as a reflection of the antisymmetrically deformed model about the plane whose normal is oriented parallel to the coordinate direction w. See Remark 7. (Default is no action).
PEN p	Specifies pen number that is used to generate the plot (Integer > 0).
DENSITY d	Specifies line density scale factor for film plotters which is d times heavier than a line density of 1 (Integer > 0).

Describers	Meaning
SYMBOLS m[,n]	All the grid points associated with the specified set will have symbol m overprinted with symbol n printed at its location. If n is not specified, only symbol m will be printed. See Remark 8.

m or n	Symbol
0	none
1	X
2	*
3	+
4	-
5	.
6	x
7	[]
8	<>
9	/ \

LABEL label	Specifies labeling options at elements and grid points:
GRID	All the grid points included in the specified set have their identification number printed to the right of the undeformed or deformed location (undeformed location in the case of superimposed plots) (Default).
ELEMENTS	All the elements included in the specified set are identified by the element identification number and type at the center of each element (undeformed location in the case of superimposed plots). See Remarks 11. and 12.
BOTH	Both GRID and ELEMENT options.
GSPC	Label those degrees-of-freedom that are constrained to zero through permanent single point constraints on GRID and GRDSET Bulk Data entries, or are constrained through SPC and SPC1 Bulk Data entries. The label consists of the grid point ID number and the constrained degrees-of-freedom.
EPID	Label elements with their respective property identification (PID) numbers. The label consists of the standard element labels and element PID.

Describers	Meaning
SHAPE	All the elements included in the specified set are shown by connecting the associated grid points in a predetermined manner. See Remark 9.
OUTLINE	Only the outline of all the elements in the specified set is shown. Elements not supported by contour plots are ignored. Outlines are always drawn with PEN 1. See Remark 9.
VECTOR v	A line will be plotted at the grid points of the set representing by length and direction the deformation of the point. See Remark 10. Possible values of v are: X, Y, or Z Requesting individual components. XY, XZ, or YZ Requesting two specified components. XYZ Requesting all three components. RXY, RXZ, or RYZ Requesting vector sum of two components. RYZ R Requesting total vector deformation. N Used with any of the preceding combinations to request no underlay shape be drawn.
PRINT	List of the average stresses at the interior grid points in the set will be printed for contour stress plots.
SHRINK t,o	t is the ratio of the reduction to the original dimensions of all two-dimensional elements except the CQUAD8 and CTRIA6 ($0.0 \leq t \leq 1.0$) (Real, Default = 0.1 which results in a 10% reduction). o is the ratio of the reduction in length to the original length for one-dimensional elements. There is no default value for o. t must be specified to shrink one-dimensional elements.
NORMALS	Plot vector normal to CHBDYP and CHBDYG elements.

Remarks:

1. If PLOT is specified with no describers, then a picture of the undeformed model will be prepared using the first defined set and the first defined origin.
2. Describers analysis through PHASE LAG must be specified in the order shown above.
3. The following should be noted when using subcase numbers for plotting eigenvectors.
 - a. If subcase numbers are specified, then the convention for displacement vectors is that the list of subcases must refer to subcase IDs whenever the number of modes recovered is equal to or less than the number of subcases defined. If the number of modes recovered is more than the subcases defined, the plot request for those modes associated with the subcases must refer to subcase IDs. After the mode associated with the last defined subcase, higher modes will be identified by incrementing the last defined subcase ID by one for each such higher mode.

- b. For the display of element quantities in contour plots, the automatic incrementing beyond the last defined subcase does not occur. All subcase numbers to be plotted must be defined. A MODES command in the Case Control Section may be used for this purpose.
 - c. In problems using cyclic symmetry solution sequences, the plot requests for segments of the model must refer to the coded subcase identification numbers (see [Theory](#) in the *MSC Nastran Reference Guide*). All eigenvectors recovered for the segment will be plotted. The RANGE option can be used to select a subset of all eigenvectors for plotting without use of coded subcase IDs.
 - d. RANGE does not require the use of subcase numbers.
4. RANGE specifies the range of values using requested subcases for which plots will be prepared. If only one number is input, it is used as the lower bound and the upper bound is set to the highest value computed. Unless otherwise noted, the default range is all values computed.
 - a. In real eigenvalue analysis, the values are natural frequencies, in units of cycles per unit time.
 - b. In buckling analysis, the values are eigenvalues.
 - c. In frequency response, the values are excitation frequencies in units of cycles per unit time.
 - d. In transient response, the values are in units of time.
 - e. In static nonlinear analysis (SOLs 106 and 153), the values are load factors. The default range is the highest load factor of each subcase.
 - f. In transient nonlinear analysis (SOLs 129 and 159), the values are in units of time. The default range is the last time step for each subcase.
 5. PHASE LAG ϕ is used in the equation:
$$u_R \cos \phi - u_I \sin \phi$$
where u_R and u_I are the real and imaginary parts of the response quantity, respectively. The printed output for magnitude and phase uses the convention of a phase lead.
 6. MAX DEFO is not recommended for general use. Each subcase is separately scaled according to its own maximum if this item is absent. If d is omitted, the set will be scaled to the maximum within the set being plotted.
 7. w specifies the basic coordinates X, Y, or Z, or any combination thereof. This option allows the plotting of symmetric and/or antisymmetric combinations, provided that an origin is selected for the portion of the model defined in the Bulk Data Section that allows sufficient room for the complete plot. This does not permit the combination of symmetric and antisymmetric subcases, as each plot must represent a single subcase. In the case of a double reflection, the figure will appear as one reflected about the plane whose normal is parallel to the first of the coordinates w, followed by a reflection about the plane whose normal is oriented parallel to the second of the coordinates w. This capability is primarily used in the plotting of models that are loaded in a symmetric or an antisymmetric manner. The plane of symmetry must be one of the basic coordinate planes.
 8. Grid points excluded from the set will not have a symbol. Grid points in an undeformed underlay will be identified with symbol 2.

9. In order to get a deformed shape, either SHAPE or OUTLINE must be present in the PLOT command. Both deformed and undeformed shapes or outlines may be specified. All the deformed shapes relating to the subcases listed may be underlaid on each of their plots by including "0" with the subcase string on the PLOT command. The undeformed plot will be drawn using PEN 1 or DENSITY 1 and symbol 2 (if SYMBOLS is specified).
10. All plots requesting the VECTOR option will have an underlay generated of the undeformed shape using the same sets, PEN 1 or DENSITY 1, and symbol 2 (if SYMBOLS is specified). If SHAPE and VECTOR are specified, the underlay will depend on whether "0" is used along with the subcases with DEFORMATION. It will be the deformed shape when not used, and will be both deformed and undeformed shapes when it is used. The part of the vector at the grid point will be the tail when the underlay is undeformed, and the head when it is deformed. If v="N" then no shape will be drawn, but other options such as SYMBOLS will still be valid.
11. Element type labels are: (Plot labels QH and TH indicate hyperelastic elements)

Element Type	Plot Label		Element Type	Plot Label
CAERO1	AE		PLOTEL	PL
CAXIF2	A2		CQUAD	QH
CAXIF3	A3		CQUAD4	Q4 or QH
CAXIF4	A4		CQUAD8	Q8 or QH
CBAR	BR		CQUADR	QR
CBEAM	BM		CQUADX	QH
CBEND	BD		CROD	RD
CBUSH	BU		CSHEAR	SH
CONEAX	CN		CSLOT3	S3
CONROD	CR		CSLOT4	S4
CDUMI	Di		CTETRA	TE
CFLUID2	F2		CTRIAX6	D1
CFLUID3	F3		CTRIA3	T3 or TH
CFLUID4	F4		CTRIA6	T6 or TH
CHBDYG	HB		CTRIAR	TR
CHBDYP	HB		CTRIAX	TH
CHEXA	HA		CTUBE	TU
CPENTA	HA		CVISC	VS

12. The heat transfer boundary condition elements CHBDYG and CHBDYP can be plotted for undeformed plots. There are several types of CHBDYi elements, as follows:

Type	No. of Primary Grid Points	Normals Available
POINT	1	yes
LINE	2	yes
AREA3	3	yes
AREA4	4	yes
REV	2	no
ELCYL	2	no
TUBE	2	yes
FTUBE	2	yes
AREA6	6	yes
AREA6	8	yes

The secondary grid points are used for ambient conditions and are ignored by the plotter. Type POINT must have a nonzero associated area (see AF on the associated PHBDY entry) and a defined normal direction (see V1, V2, V3 on the CHBDYP entry) to be plotted. It is plotted as a hexagon with approximately the correct area. Type LINE must have a nonzero width (see AF on the associated PHBDY entry) and a defined normal in order to plot.

13. To assign PLOT command to superelements it requires an SEUPPLOT or a SEPLOT command.

Examples:

1. Undeformed SHAPE using first defined SET, first defined ORIGIN and PEN 1 (or DENSITY 1).

PLOT

2. Undeformed SHAPE using SET 3, ORIGIN 4, PEN 2 (or DENSITY 2) with each grid point of the set having a + placed at its location, and its identification number printed adjacent to it.

PLOT SET 3 ORIGIN 4 PEN 2 SHAPE SYMBOLS 3 LABEL

3. Modal deformations as defined in subcase 5 using first defined SET, first defined ORIGIN, and PEN 1 (or DENSITY 1).

PLOT MODAL DEFORMATION 5 SHAPE

4. STATIC deformations as defined in subcases 3, 4, 5, and 8 deformed SHAPE; drawn with PEN 4, using first defined SET and ORIGIN, underlaid with undeformed SHAPE drawn with PEN 1. This command will cause four plots to be generated.

PLOT STATIC DEFORMATION 0, 3 THRU 5, 8 PEN 4, SHAPE

5. Deformations as defined in subcases 1, 2, 3, 4, and 5 undeformed underlay with PEN 1, consisting of SET 2 at ORIGIN 3, SET 2 at ORIGIN 4 (with a < placed at each grid point location), and SET 35 at ORIGIN 4. Deformed data as follows: SHAPE using SET 2 at ORIGIN 3 (PEN 3) and SET 35 at ORIGIN 4 (PEN 4); 3 VECTORS (X, Y and Z) drawn at each grid point of SET 2 at ORIGIN 4 (PEN 4) (less any excluded grid points), with o placed at the end of each vector.

```
PLOT STATIC DEFORMATION 0 THRU 5,
SET 2 ORIGIN 3 PEN 3 SHAPE,
SET 2 ORIGIN 4 PEN 4 VECTORS XYZ SYMBOLS 0,
SET 35 SHAPE
```

6. Static deformations as defined in subcases 3 and 4, both halves of a problem solved by symmetry using the X-Y principal plane as the plane of symmetry. SET 1 at ORIGIN 2 and SET 2 at ORIGIN 3, with the deformed shape plotted using DENSITY 3 and the undeformed model plotted using DENSITY 1. The deformations of the “opposite” half will be plotted to correspond to symmetric loading. This command will cause two plots to be generated.

```
PLOT STATIC DEFORMATIONS 0, 3, 4,
SET 1 ORIGIN 2 DENSITY 3 SHAPE,
SET 1 SYMMETRY Z SHAPE,
SET 2 ORIGIN 3 SHAPE,
SET 2 SYMMETRY Z SHAPE
```

7. Transient deformations as defined in subcase 1 for time = 0.1 to time = 0.2, using SET 1 at ORIGIN 1. The undeformed SHAPE using PEN or DENSITY 1 with an * at each grid point location will be drawn as an underlay for the resultant deformation vectors using PEN or DENSITY 2 with an < typed at the end of each vector drawn. In addition, a plotted value of 2.0 will be used for the single maximum deformation occurring on any of the plots produced. All other deformations on all other plots will be scaled relative to this single maximum deformation. This command will cause a plot to be generated for each output time step which lies between 0.1 and 0.2.

```
PLOT TRANSIENT DEFORMATION, TIME 0.1, 0.2,
MAXIMUM DEFORMATION 2.0, SET 1, ORIGIN 1, PEN 2, SYMBOLS 2,
VECTOR R
```

8. Contour plot of x-component of normal stress for elements in SET 2 in basic coordinate system at a distance Z1 from neutral plane with 10 contour lines, an outline of elements in SET 2, and using

```
ORIGIN 4.
CONTOUR XNORMAL
PLOT CONTOUR, SET 2, ORIGIN 4, OUTLINE
```

Contour plot of magnitude of displacements at grid points associated with elements in SET 5 with 5 contours having values of 2., 4., 6., 8., 10., and an outline of the elements in SET 5 using

```
ORIGIN 4.
CONTOUR MAGNIT, LIST 2., 4., 6., 8., 10.
PLOT CONTOUR, SET 5, OUTLINE
```

9. Plot the imaginary part of the complex eigenvector in SET 1.

```
PLOT CMODAL DEFORMATION PHASE LAG 90. SET 1 VECTOR R
```

PLOTTER

Plot File Format

Selects format of plot file for interpretation by plotter postprocessor.

Format:

PLOTTER { NAST
 SC }

Example:

PLOTTER NAST

Describers	Meaning
NAST	Specifies format suitable for Postscript plotters (Default).
SC	Specifies Stromberg-Carlson microfilm plotter format.

Remark:

1. If no PLOTTER command is specified, then PLOTTER NAST is the default.
2. If “PLOTTER NAST” is specified (or taken by default) and if the format of the PLOT file has been changed to formatted through the use of an ASSIGN statement, plot data will be generated in PostScript mode directly instead of being generated in binary mode. PostScript plot control information may be specified using the SYS= descriptor on the ASSIGN statement or by using the SYSFIELD= command-line keyword, specifying the control information using the PLOT (keyword=value,...) options. The valid keywords are the same as those that can be specified for the PLOTPS or MSCPLOTPS utility programs except that the “begin”, “debug”, “dump”, “end”, “format” and “output” keywords are not allowed.

PROJECTION

Separation Between Projection Plane and Observer

Defines the separation along the r-axis, and between the observer and the projection plane, if not already specified on the VANTAGE POINT command. Used by stereoscopic projections only.

Format:

PROJECTION PLANE SEPARATION do

Example:

PROJ PLAN SEPA 1.5

Descriptor	Meaning
do	Separation of the observer and the projection plane on the r-axis in model units. The VANTAGE POINT command may also specify the separation (Real, Default = 2.0).

Remarks:

1. The PROJECTION PLANE SEPARATION command is not recommended. The FIND command is recommended because it automatically calculates the optimum separation.
2. A theoretical description of projection plane separation is contained in [Plotting](#) in the *MSC Nastran Reference Guide*.

PTITLE**Plot Frame Title**

Defines a character string that will appear at the top of the plot frame on the line below the sequence number.

Format:

PTITLE ptitle

Example:

PTITLE RIGHT WING -- LOAD CASE 3

Descriptor	Meaning
ptitle	Any character string (Character, Default= blank).

Remarks:

1. PTITLE may not be continued to the next command line.
2. Up to four lines of title information will be printed in the lower left-hand corner of each plot. The text for the top three lines is taken from the TITLE, SUBTITLE, and LABEL commands in the Case Control Section. (See the [Case Control Commands, 191](#) for a description of the TITLE, SUBTITLE, and LABEL commands). The text for the bottom line may be of two forms depending on the type plot requested. One form contains the word UNDEFORMED SHAPE. The other form contains the type of plot (statics, modal, etc.) subcase number, load set or mode number, frequency or eigenvalue or time, and (for complex quantities) the phase lag or magnitude. The sequence number for each plot is printed in the upper corners of each frame. The sequence number is determined by the relative position of each PLOT execution command in the plot package. The information on the PTITLE command will be printed on the line below the sequence number. The date and (for deformed plots) the maximum deformation are also printed at the top of each frame.

SCALE**Scale Factor**

Defines reduction, as a scale factor, of model's dimensions so that model fits on a plot frame.

Format:

SCALE a [b]

Example:

SCALE 0.5

Describers	Meaning
a	Scale factor (Default=1.0).
b	Ratio of model size/real object size for stereoscopic projection only.

Remarks:

1. The SCALE command is not recommended. The FIND command is recommended because it automatically calculates the optimum scale factor.
2. For orthographic or perspective projections, a is the ratio of the plotted object, in inches, to the real object in the units of model; i.e., one inch of paper equals one unit of model.
3. For stereoscopic projections, the stereoscopic effect is enhanced by first reducing the real object to a smaller model according to b, and then applying a. The ratio of plotted/real object is then the product of a and b.
4. If the NASTRAN general purpose plotter is used in combination with the PLOTPS postprocessing routine, a scale factor can be computed as follows:

$$a = p \cdot \frac{20}{7} \cdot K$$

where:

p = ratio of plot size to object size. For instance, if the model is 100 inches long and the plot size is 7 inches, then

$$p = \frac{7}{100} = .007$$

$\frac{20}{7}$ = ratio of default PAPER SIZE to default PLOTPS frame size.

K = SCALE value on PLOTPS command (Default=1.0). See [Using the Utility Programs](#) in the *MSC Nastran Installation and Operations Guide*.

SEPLOT**Superelement Plot Delimiter**

Assigns the subsequent PLOT or XYPILOT commands to one or more superelements.

Format:

SEPLOT seid1 [seid2 ...]

Examples:

SEPLOT 5

SEPLOT 0 3 7 200

Descriptor	Meaning
seidi	Superelement identification number (Integer ≥ 0).

Remarks:

1. See also related command SEUPPLOT.
2. Any PLOT or XYPILOT commands appearing above all SEPLOT (or SEUPPLOT) commands will apply in all SEPLOT (or SEUPPLOT) packets.
3. For multiple PLOT or XYPILOT commands, there should be a SEPLOT command with each PLOT. For the special case where the PLOTS or XYPILOTS refer to the same superelements and use the same FIND, a single SEPLOT followed by a single FIND may be placed above all commands.

SET Set Definition Under OUTPUT(PLOT)

Defines a set of elements or grid point numbers to be plotted.

The SET command specifies sets of elements or grid points, corresponding to portions of the model, which may be referenced by PLOT and FIND commands. The SET command is required. Each set of elements defines, by implication, a set of grid points connected by those elements. The set may be modified by deleting some of its grid points. The elements are used for creating the plot itself and element labeling, while the grid points are used for labeling, symbol printing, and drawing deformation vectors.

Format:

$$\begin{aligned} \text{SET } n = & \left[\text{ALL} \left[\begin{array}{l} \text{ELEMENTS} \\ \text{GRID POINTS} \end{array} \right] \left[\text{EXCEPT} \left\{ \begin{array}{llll} \text{type1} & \text{type2} & \dots & \text{typej} \\ k1 & k2 & \dots & kj \text{ THRU } kk \text{ BY } incj \end{array} \right\} \right] \right], \\ & \left[\begin{array}{ll} \text{INCLUDE} & \text{ELEMENTS} \\ \text{EXCLUDE} & \text{GRID POINTS} \end{array} \right] \left\{ \begin{array}{llll} \text{type1} & \text{type2} & \dots & \text{typej} \\ k1 & k2 & \dots & kj \text{ THRU } kk \text{ BY } incj \end{array} \right\}, \\ & \left[\text{EXCEPT} \left\{ \begin{array}{llll} \text{type1} & \text{typem} & \dots & \text{typen} \\ k1 & km & \dots & kn \text{ THRU } ko \text{ BY } incn \end{array} \right\} \right] \end{aligned}$$

Examples:

1. SET 1 consists of elements 1, 5, 10, 11, 13, 14, 15, 20, 22, 24, and 26.

```
SET 1=INCLUDE 1, 5, 10 THRU 15 EXCEPT 12, INCLUDE 20 THRU 26 BY 2
```

2. SET 2 consists of all CTRIA3 and CQUAD4 elements except element 21.

```
SET 2=QUAD4 TRIA3 EXCEPT 21
```

3. SET 10 includes all CTRIAR elements plus elements 70 through 80.

```
SET 10 TRIAR INCLUDE ELEMENTS 70 THRU 80
```

4. SET 15 includes all elements from 15 to 20 and 26 to 100.

```
SET 15=15 THRU 100 EXCEPT 21 THRU 25
```

5. SET 2 includes all elements except CTETRA elements.

```
SET 2=ALL EXCEPT TETRA
```

Descriptor	Meaning
n	Sets identification number (0<Integer<999999).
ALL	Selects all elements or grid points. See typei.
ELEMENTS	Specifies that all identification numbers refer to elements.
GRID POINTS	Specifies that all identification numbers refer to grid points.

Describer	Meaning
INCLUDE	Includes specified element or grid point identification numbers or elements in the set.
EXCLUDE	Excludes specified element or grid point identification numbers or element types in the set.
EXCEPT	Modifies a prior ALL, INCLUDE, or EXCLUDE specification.
typei	Element types. The allowed element types are (Character):

Element Type	typei on SET Command		Element Type	typei on SET Command
CAXIF2	AXIF2		CQUAD	QUAD
CAXIF3	AXIF3		CQUAD4	QUAD4
CAXIF4	AXIF4		CQUAD8	QUAD8
CBAR	BAR		CQUADR	QUADR
CBEAM	BEAM		CQUADX	QUADX
CBEND	BEND		CROD	ROD
CBUSH	BUSH		CSHEAR	SHEAR
CONEAX	CONE		CSLOT3	SLOT3
CONROD	CONROD		CSLOT4	SLOT4
CDUMi	DUMi		CTETRA	TETRA
CFLUID2	FLUID2		CTRIAX6	TRIA6
CFLUID3	FLUID3		CTRIA3	TRIA3
CFLUID4	FLUID4		CTRIA6	TRIA6
CHBDYG	HBDY		CTRIAR	TRIAR
CHBDYP	HBDY		CTRIAX	TRIA6
CHEXA	HEXA		CTUBE	TUBE
CPENTA	PENTA		CVISC	VISC
PLOTEL	PLOTEL			

THRU	Specifies a range of identification numbers.
BY	Specifies an increment for a THRU specification.
inci	Increment for THRU range (Integer > 0).

Remarks:

1. This form of the SET command can only be specified after an OUTPUT(PLOT) delimiter.
2. The INCLUDE, EXCLUDE, and EXCEPT specifications may be specified more than once in the same set. See previous examples.
3. Commas or spaces may be used as separators.
4. Not all of the identification numbers in a THRU range have to correspond to elements or grid points. For example, elements 2, 4, 7, and 9 may be selected with 2 THRU 9, even if elements 3, 5, 6, and 8 do not exist. This is called an open set. Note that large open sets can cause higher computational costs.

SEUPPLOT**Superelement Plot Delimiter**

Assigns the subsequent PLOT or XYPLOT commands to a superelement and all of its upstream superelements.

Format:

SEUPPLOT seid

Example:

SEUPPLOT 7

Describer	Meaning
seid	Superelement identification number (Integer ≥ 0).

Remarks:

1. See also related command SEPLOT.
2. Any PLOT or XYPILOT commands appearing above all SEUPPLOT (or SEPLOT) commands will apply in all SEUPPLOT (or SEPLOT) packets.
3. For multiple PLOT or XYPILOT commands, there should be a SEUPPLOT command with each PLOT. For the special case where the PLOTS or XYPILOTS refer to the same superelements and use the same FIND, a single SEUPPLOT followed by a single FIND may be placed above all the commands.

STEREOSCOPIC

Selects Stereoscopic Projection

See the description of the [ORTHOGRAPHIC, etc., 757](#).

VANTAGE POINT

Location of the Observer

Defines the location of the observer with respect to the model in the (r, s, t) coordinate system defined on the AXES command for perspective and stereoscopic projections only.

Format:

VANTAGE POINT ro so to do sor

Example:

VANT 100.

Describers	Meaning
ro	Location of the observer on the r-axis in model units (Real).
so	Location of the observer and left eye of the observer on the s-axis, in model units, for perspective and stereoscopic projections, respectively (Real).
to	Location of the observer on the t-axis in model units (Real).
do	Separation of the observer and the projection plane on the r-axis in model units. The PROJECTION PLANE SEPARATION command may also specify the separation (Real).
sor	Location of the of the observer's right eye for stereoscopic projections in model units (Real).

Remarks:

1. VANTAGE POINT or the FIND command must be specified if the PERSPECTIVE or STEREOSCOPIC command is also specified.
2. The VANTAGE POINT command is not recommended. The FIND command is recommended because it automatically calculates the optimum vantage point.
3. A theoretical description of the vantage point is contained in [Plotting](#) in the *MSC Nastran Reference Guide*.

VIEW

Angular Relationship of Observer's Coordinate System

Defines the angular relationship between observer's coordinate system (r, s, and t axes specified on the AXES command) and basic coordinate system.

Format:

VIEW gamma beta alpha

Example 1:

View the model from the r-axis.

VIEW 0. 0. 0.

Example 2:

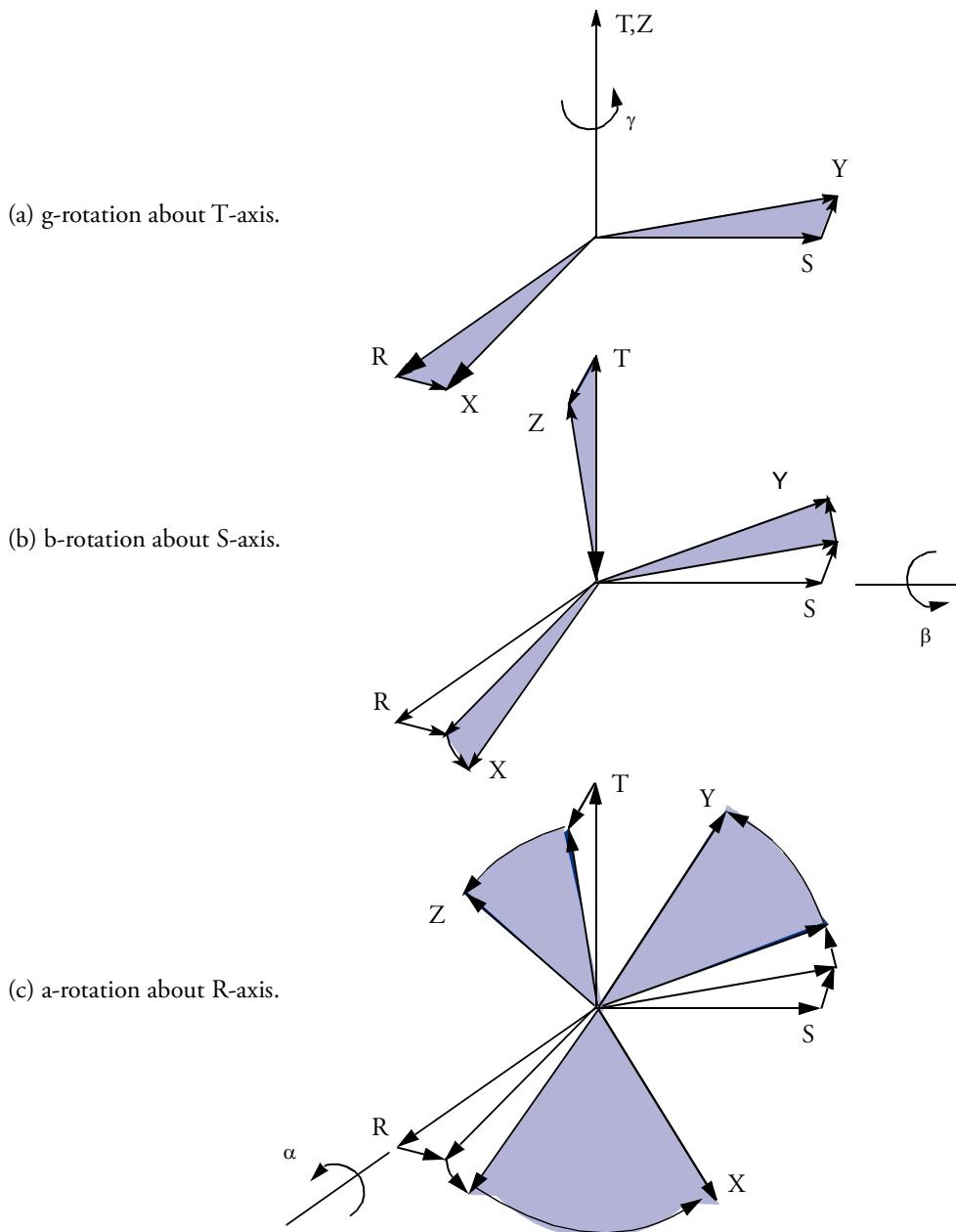
View the model midway between the r- and s-axes.

VIEW 45. 0. 0.

Describers	Meaning
gamma	Angle of rotation, in degrees, of t axis specified on AXES command (Default = 34.27).
beta	Angle of rotation, in degrees, of s axis specified on AXES command (Default = 23.17 if ORTHOGRAPHIC or STEREOSCOPIC command is specified and 0.0 if PERSPECTIVE command is specified).
alpha	Angle of rotation, in degrees, of r axis specified on AXES command (Default = 0.0).

Remarks:

1. If no VIEW command is specified, then VIEW 34.27 23.17 0.0 is assumed for orthographic and stereoscopic projections; and VIEW 34.27 0.0 0.0 is assumed for perspective projections. The default values produce a plot in which unit vectors on the axes of the basic coordinate system have equal lengths.
2. The angles are rotated in sequence: gamma rotates the t-axes, followed by beta which rotates the s-axes, followed by alpha which rotates the r-axes.



3. The VIEW command specifies the position of the model with respect to the s-t plane. Gamma and beta represent the angles of turn and tilt. Alpha is normally not used since it does not affect the orientation of the s-t plane, but only its orientation on the plot frame.

6

Parameters

- Parameter Descriptions 784
- Parameter Applicability Tables 1012

Parameter Descriptions

Parameters are used extensively in the solution sequences for input of scalar values and for requesting special features. Parameters values are specified on PARAM Bulk Data entries or PARAM Case Control commands. The PARAM Bulk Data entry is described in the [Bulk Data Entries, 1117](#). The PARAM Case Control command is described in the [PARAM \(Case\), 529](#). PARAMs may also be used in .rc and .ini files as described in [Specifying Parameters, 3350](#). in the *MSC Nastran Installation and Operations Guide*. A complete alphabetical list of PARAMeter names and their functions is given in this section.

If the Bulk Data involves the use of part superelements or external superelements, the following points should be noted regarding the use of the PARAM Bulk Data entry:

1. PARAM entries specified in the Main Bulk Data portion of the input data apply *only to the residual and not to the part superelements or external superelements*.
2. PARAM entries specified in the BEGIN SUPER portion of the Bulk Data for a part superelement or an external superelement apply *only to that superelement*.
3. The most convenient way of ensuring that PARAM entries apply not only to the residual, but also to all part superelements and external superelements is to specify such PARAM entries in Case Control, not in the Main Bulk Data. This is particularly relevant for such PARAMs as POST.

ACEXTMTD

ACEXTMTD is used to select a different solver for solving exterior acoustic model. Valid options are ITER for selecting iterative solver and KRYLOV for selecting Krylov solver.

ACEXTSET

ACEXTSET is used to select ITER bulk data entry for PARAM,ACEXTMTD,ITER.

ACOUT

Default = PEAK

ACOUT specifies the type of output to be used with the FORCE Case Control command in coupled fluid-structural analysis. ACOUT=RMS requests root-mean-square output.

To obtain sound pressure level in units of dB and dBA given by the FORCE command, a peak reference pressure must be specified with PARAM, PREFDB. The dB level is defined as:

$$\text{dB} = 20 \cdot \log\left(\frac{P}{\text{PREFDB}}\right)$$

Instead of ACCELERATION, suffix INT can be utilized to generate INTENSITY. See also the Case Control command [FLSTCNT \(Case\), 379](#).

ACOWEAK

Default = NO

ACOWEAK controls the execution of weakly coupled acoustic formulation logic. PARAM,ACOWEAK,YES will activate weakly coupled acoustic formulation which will solve exterior acoustic model along with CACINFx separately using load generated from structural response and coupling matrix. ACOWEAK,YES is available only in SOL 111 and SOL 200 with ANALYSIS=MFREQ.

In the cases where loading is on exterior acoustic only, PARAM,ACOWEAK,YESR will solve for exterior acoustic model first and generates loading at coupling nodes for structural and interior acoustic model. In addition, PARAM, ACOWEAK, YESR can be used to force NASTRAN to solve exterior acoustic model first if loading is on both structural and exterior acoustic models.

Note that the detailed knowledge on the loading is not required to use ACOWEAK, and with PARAM, ACOWEAK, yes, the Automated logic will determine which part of FSI to solve first.

The ACOWEAK solution technique supports trim components via TRIMGRP or ACTRIM between structure and interior cavity only. If there are trim components situated between the structure and exterior cavity, fully coupled solution techniques should be utilized.

With ACOWEAK solution scheme, the exterior acoustic portion of the model is solved in physical coordinates. Therefore, acoustic structural modal participation factor will not be possible since acoustic field eigenvalues and/or eigenvectors are missing or incomplete.

ACSYM

Default = YES

By default, the dynamic equations for coupled fluid-structure analysis in frequency response are symmetrized for efficiency. PARAM,ACSYM,NO requests the pre-Version 69 formulation which involves no symmetrization and will require more CPU time. See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide*. See also the Case Control command [FLSTCNT \(Case\), 379](#).

If the iterative solver is selected (see the Case Control command, [SMETHOD \(Case\), 579](#)) then the external work diagnostic will be different between ACSYM=YES and ACSYM=NO.

ACTDMP

Default=0

This parameter can be used to specify DMP processors for Actran. By default(ACTDMP=0), it will be set to DMP of Nastran job submittal.

ACTSMP

Default=0

This parameter can be used to specify SMP processors for Actran. By default(ACTSMP=0), it will be set to SMP of Nastran job submittal.

ACTMEM

Default=0

This parameter can be used to specify TOTAL amount memory for all processors for Actran via DMP or PARAM,ACTDMP in 4 bytes mega-words. By default (ACTMEM=0), Actran get the memory assigned to master node only. An example, for 'memorymax=64gb mem=max', use PARAM,ACTMEM,16000 which get ACTRAN same amount of TOTAL memory as NASTRAN.

ADJMETH

Default = 0

This parameter selects the processing method used in a triple matrix product in module DSADJ. The default is usually preferred, but ADJMETH=1 can be used when disk space is critical. ADJMETH=2 only holds the active solution vectors.

ADMEXTU

Default = 0

ADMEXTU is a file unit number used by the SOL 111/112 restart run along with ADMPOST=1. This file unit no is assigned to external super element op2 file. See the Case Control command [ADAMSMNF* \(Case\)](#), footnote and Remark 20.

ADMPOST

Default = 0

This parameter allows the user to bring an ADAMS results file into Nastran for modal data recovery in SOL 111 and SOL 112. A value of 1 brings in the ADAMS results without consideration of rigid body motion in the display of the results. A value of 2 brings in the ADAMS results with consideration of rigid body motion in the display of the results. See the Case Control command [ADAMSMNF* \(Case\)](#), 221 footnote and Remark 19. for more details on restrictions and required file assignments.

ADPCON

Default = 1.0

Initial penalty values used in contact analysis are calculated automatically by the program and are given by $k \cdot SFAC \cdot |ADPCON|$ where k is a number selected for each slave node based on the diagonal stiffness matrix coefficients that are in the contact region, and SFAC is the value specified by the user in the SFAC field of the BCONP Bulk Data entry. The ADPCON value applies to all the contact regions in the model. During the analysis, if convergence problems are encountered, penalty values are automatically reduced. Still there may be some problems where convergence can not be achieved with default values. In such cases, analysis may be restarted with a lower value of ADPCON.

In some cases the default penalty values may be low. In such situations analysis may be restarted with a higher value of ADPCON.

Generally, penalty values are recalculated every time there is a change in stiffness. However, if ADPCON is negative, penalty values are calculated only at the beginning of each subcase, and penalty values are not adjusted during analysis. This is useful if the contact between two elastic bodies is being analyzed.

ADSTAT

Character; Default = YES

In transient analysis (SOLs 101,109 and 112 only), if there is a preload (see Case Control command [STATSUB \(Case\)](#), 585) and ADSTAT=YES, the static displacements, SPC forces, and MPC forces will be included in the transient output. There is a limitation that OLOAD is not included. Another limitation is that element stress and force recover will neglect thermal strains and, hence will be wrong if there is a TEMP(LOAD) request. For any other value of ADSTAT the preload effects will not be included in the output, hence the output will be the perturbation displacements, SPC forces and MPC forces.

AERODOF

Default=35

Digits 1 through 6 indicate which k-set degrees of freedom are to be output. This is only for Double Lattice lifting surfaces that provide output in the plunge (3) and pitch (5) degrees of freedom by default. If additional digits are specified (e.g. AERODOF=1235), then the k-set aerodynamic matrices are padded with null columns for the additional degrees of freedom. Nastran will not create aerodynamic data for these added degrees of freedom. This parameter is only useful to the user that knows how to augment the existing aerodynamic data from an external source. Degrees of freedom 3 and 5 may not be omitted.

AESDISC

Default = 1.E-8

Tolerance for discarding generalized coordinates in the RITZ method (see Case Control command, [AESMETH, 787](#)) which are not linearly independent.

AESMAXIT

Default = 15

Maximum number of iterations for the ITER method (see Case Control command, [AESMETH, 787](#)).

AESMETH

Default = SELECT

Solution method for static aeroelastic analysis.

- | | |
|--------|---|
| SELECT | selects the DIRECT method on models with less than 50000 DOF in the solution set; otherwise selects AUTO. |
| AUTO | selects the reduced basis method for an approximate solution, which is used as starting vectors for an ITER solution. |
| DIRECT | selects the direct solution. |
| RITZ | selects the reduced basis approximate solution. |
| ITER | selects the iterative solution. |

AESRNDM

Default = 2

Number of random vectors to use as generalized functions in the RITZ method (see Case Control command, [AESMETH, 787](#)).

AESTOL

Default = 1.E-10

Convergence criteria for the iterative solver.

AGGRROT

Default = YES

PARAM, AGGRROT, YES (Default) produces both translation and rotational terms in the structure-fluid interface matrix AGG. PARAM, AGGRROT, NO produces only translational terms in the structure-fluid interface matrix AGG. Since fluid elements see no rotational degrees of freedom, AGGRROT YES or NO should have no significant effect on the results of the fluid cavity. Results on the structural side may change especially near unconstrained boundaries where there may be a reversal in the signs of some acceleration terms.

ALPHA1, ALPHA2

Default = 0.0, 0.0

In frequency and transient response analysis, if PARAM,ALPHA1 and/or ALPHA2 are not equal to zero, then Rayleigh damping is added to the viscous damping. ALPHA1 is the scale factor applied to the mass matrix and ALPHA2 to the structural stiffness matrix. In SOL400, ALPHA1 and ALPHA2 are supported for both the linear and nonlinear elements. In SOL 129, Alpha1 scales the mass matrix, and ALPHA2 is not used.

$$[B'] = [B] + \text{ALPHA1} \cdot [M] + \text{ALPHA2} \cdot [K]$$

If ξ_i is the damping ratio for the i-th mode ω_i (radians/unit time), then ALPHA1 (α_1) and ALPHA2 (α_2) may be computed as

$$\begin{Bmatrix} \alpha_1 \\ \alpha_2 \end{Bmatrix} = \frac{2\omega_i\omega_j}{\omega_j^2 - \omega_i^2} \begin{bmatrix} \omega_j & -\omega_i \\ -1 & \frac{1}{\omega_i} \end{bmatrix} \begin{Bmatrix} \xi_i \\ \xi_j \end{Bmatrix}$$

and the damping ratio for any other ξ_l mode becomes

$$\xi_l = \frac{\omega_i\omega_j}{\omega_j^2 - \omega_i^2} \left[\left(\frac{\omega_j}{\omega_l} - \frac{\omega_l}{\omega_j} \right) \xi_i - \left(\frac{\omega_i}{\omega_l} - \frac{\omega_l}{\omega_i} \right) \xi_j \right]$$

Note:	<p>The use of Rayleigh damping with non-zero values of ALPHA1 may not be appropriate for enforced motion problems involving large mass since the resulting damping matrix may essentially violate the assumption of large mass in the problem and thus give wrong answers.</p> <p>Similarly, the use of Rayleigh damping with non-zero values of ALPHA2 may not be appropriate for enforced motion problems involving large stiffness since the resulting damping matrix may essentially violate the assumption of large stiffness in the problem and thus give wrong answers.</p> <p>Since Version 2005r3, the ALPHA1, ALPHA2 parameters apply only to stationary structural components. Prior to this version, the parameters applied to both stationary and rotating structural components. With the introduction of Version 2005r3, new parameters ALPHAR1, ALPHAR2 are defined on the RSPINR and RSPINT bulk data entry for the rotating component, see Remark 8. of the DAMPING bulk data description for a comment on this item.</p> <p>Rayleigh damping is designed to be applied only at Superelement, Part Superelement, or External Superelement residual assembly time. It is not applied in individual Parts or External superelements.</p>
--------------	--

ALPHA1FL, ALPHA2FL

Default = 0.0, 0.0

In frequency and transient response analysis, if PARAM,ALPHA1FL and/or ALPHA2FL are not equal to zero, then Rayleigh damping is added to the viscous damping. ALPHA1FL is the scale factor applied to the mass matrix and ALPHA2FL to the fluid stiffness matrix.

$$[B'] = [B] + \text{ALPHA1FL} \cdot [M] + \text{ALPHA2FL} \cdot [K]$$

If ξ_i is the damping ratio for the i-th mode ω_i (cycles/unit time), then ALPHA1FL (α_1) and ALPHA2FL (α_2) may be computed as

$$\begin{Bmatrix} \alpha_1 \\ \alpha_2 \end{Bmatrix} = \frac{2\omega_i\omega_j}{\omega_j^2 - \omega_i^2} \begin{Bmatrix} \omega_j & -\omega_i \\ -1 & 1 \\ \omega_j & \omega_i \end{Bmatrix} \begin{Bmatrix} \xi_i \\ \xi_j \end{Bmatrix}$$

and the damping ratio for any other ξ_l mode becomes

$$\xi_l = \frac{\omega_i\omega_j}{\omega_j^2 - \omega_i^2} \left[\left(\frac{\omega_j}{\omega_l} - \frac{\omega_l}{\omega_j} \right) \xi_i - \left(\frac{\omega_i}{\omega_l} - \frac{\omega_l}{\omega_i} \right) \xi_j \right]$$

Note:	<p>The use of Rayleigh damping with non-zero values of ALPHA1FL may not be appropriate for enforced motion problems involving large mass since the resulting damping matrix may essentially violate the assumption of large mass in the problem and thus give wrong answers.</p> <p>Similarly, the use of Rayleigh damping with non-zero values of ALPHA2FL may not be appropriate for enforced motion problems involving large stiffness since the resulting damping matrix may essentially violate the assumption of large stiffness in the problem and thus give wrong answers.</p> <p>Rayleigh damping is designed to be applied only at Superelement, Part Superelement, or External Superelement residual assembly time. It is not applied in individual Parts or External superelements.</p>
--------------	---

ALTRED

Default = NO

ALTRED=YES requests the alternate stiffness and load reduction technique for superelement analysis in SOLs 101 and 114. This technique is described in [Static Solutions in SubDMAP SEKRRS, 375](#), [Static and Dynamic Load Generation, 378](#), and [Data Recovery Operations in SubDMAP SEDISP](#) in the *MSC Nastran Reference Guide*.

ALTSHAPE

Default = 0

ALTSHAPE selects the set of displacement shape functions to be used in p-version analysis. PARAM,ALTSHAPE,0 selects the MacNeal set. PARAM,ALTSHAPE,1 selects the Full Product Space set. For ALTSHAPE=1, IN=1 and ISOP=1 must be specified on the PSOLID entry.

ARBMASP

Default = 2

ARBMASP defines the maximum aspect ratio of CQUAD4 generated for CP/OP options of PBMSECT if ARBMSTYP=timoshen. Since the thickness of a ply is usually the small dimension, ARBMASP affects the size of CQUAD4 lengthwise along a segment.

ARBMFEM

Default = YES

ARBMFEM controls the generation of '.bdf' file which contains the Finite Element Model of the arbitrary beam cross section. This parameter is functional for PBRSECT and PBMSECT only. To turn off the capability, set value of ARBMFEM to 'NO'.

ARBMNOW

Default = 0

ARBMNOW controls the overwrite of segment property for PBMSECT, arbitrary beam cross section. With default value of ARBMNOW, segment property overwrite will occur during the process of assigning the property for each segment. To turn off segment property overwrite, use PARAM,ARBMNOW,1.

ARBMP

Default = YES

ARBMP controls the generation of outline plot for arbitrary beam cross section in PostScript format. This parameter is functional for PBRSECT and PBMSECT only. To turn off capability, set value of ARBMP to 'NO'.

ARBMS

Default = NO

ARBMS controls the stress recovery for the whole arbitrary beam cross section and the companion 'screened' stresses. The stress recovery for the whole cross section is available in 'OP2' format and suitable for post-processing. The 'screened' stresses for CBAR and CBEAM elements is available in print file (f06) and can be utilized for design optimization via RTYPE=ABSTRESS on DRESP1. This parameter is functional for PBRSECT and PBMSECT only. To turn on the capability, set value of ARBMS to 'YES'.

Note: The recovery of 'screened' stresses will be turned on automatically if RTYPE=ABSTRESS is in use on DRESP1.

ARBMSTYP

Default = TIMOSHEN

ARBMSTYP controls the solution algorithm for composite arbitrary beam cross-section. Other valid option is 'TIMOFORC'. This parameter is functional for PBRSECT and PBMSECT only.

ARF

Default = 0.95

See [FLUIDMP, 828](#).

ARS

Default = 0.95

See [FLUIDMP, 828](#).

ASCOUP

Default = YES

In coupled fluid-structure analysis, by default, the coupling between the fluid and structure is computed. This interaction will be ignored if PARAM,ASCOUP,NO is specified. See also the Case Control command [FLSTCNT \(Case\), 379](#) for alternative selections.

ASING

Default = 0

ASING specifies the action to take when singularities (null rows and columns) exist in the dynamic matrices (or $[K_{ll}]$ in statics). If ASING=-1, then a User Fatal Message will result.

If ASING=0 (the Default), singularities are removed by appropriate techniques depending on the type of solution being performed. The procedures used are described in [Data Recovery Operations in SubDMAP SEDRCVR, 392](#) and [Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS](#) in the *MSC Nastran Reference Guide*.

AUNITS

Default = 1.0

AUNITS is used in SOL 144 to convert accelerations specified in units of gravity on the TRIM Bulk Data entry to units of distance per time squared. Accelerations are divided by this parameter. To convert accelerations input in g's into physical, consistent units, set AUNITS to 1/g.

AUTOADJ

Default = YES

In SOL 200, a value of yes will automatically choose the direct or adjoint sensitivity analysis based on the performance criteria. Other options for this parameter are:

NO - the adjoint method will not be selected.

NOSTAT - the adjoint method will not be selected for static analysis subcases, but may be selected for frequency response subcases based on the performance criteria.

NOFREQ - the adjoint method will not be selected for frequency responses subcases, but may be selected for static analysis subcases based on the performance criteria.

The default should be preferred in all cases. However, the other options allow investigation of the alternative of using direct sensitivity methods.

AUTOGOUT

Default = NO

PARAM,AUTOGOUT,YES simplifies grid point data recovery requests when only a set of elements is specified. In other words, given a set of elements the program will automatically determine all connected grid points and honor the data recovery requests for both grid points and elements. See also PARAM,OELMOPT description.

If PARAM,AUTOGOUT,YES is specified then the program assumes that SET 2 defines the set of desired elements and SET 1 defines the additional grids **not** connected to the elements in SET 2. If no additional grids are desired then the user need only specify SET 1=0. For example, in Case Control,

```
Param,autogout,yes
Set 2 = 27,35,25,41234,123,thru,134,9701,9901 $ elements
Set 1 = 0                                     $ additional grids
Displ=1
Gpfor=1
Stres=2
```

If the user prefers to use different set IDs then user PARAMs OSETELE and OSETGRD may be used to change the set ids used by this feature. For example:

```
Param,autogout,yes
Param,osetele,200
Param,osetgrd,100
Set 200 = 27,35,25,41234,123,thru,134,9701,9901 $ elements
Set 100 = 0                                     $ additional grids
Displ=100
Gpfor=100
Stres=200
```

And if the user wants to add more grids that are not connected to the element set:

```
Param,autogout,yes
Param,osetele,200
Param,osetgrd,100
Set 200 = 27,35,25,41234,123,thru,134,9701,9901 $ elements
Set 100 = 29                                     $ additional grids
Displ=100
Gpfor=100
Stres=200
```

AUTOMSET

Character, Default = NO (Except in SOL 400 with ANALYSIS=STATICS)

The relationship between dependent and independent degrees-of-freedom for rigid elements and MPCs may be altered from the user's input specification via PARAM,AUTOMSET,YES or PARAM,AUTOMSET,LUSOL. For PARAM, AUTOMSET,YES, or PARAM,AUTOMSET, LUSOL, dependent

degrees-of-freedom of the model, the M-set are automatically determined by employing a rectangular decomposition of the RMG matrix.

YES	For solutions other than SOL 600, the rectangular decomposition is performed via UMFPACK, which is used by permission. Copyright (c) 2002 by Timothy A. Davis. All rights reserved. Availability: http://faculty.cse.tamu.edu/davis/suitesparse.html
LUSOL	The rectangular decomposition is performed via LUSOL, which is capable of Rook or full pivoting during rectangular decomposition. LUSOL option is especially suitable for ill conditioned RMG matrix or problems with redundant multi-point constraints. Availability: http://web.stanford.edu/group/SOL/software/lusol/

Notes:

- When used in SOL 600, AUTOMSET=Yes maps to Marc's AUTOMSET. By default or if PARAM,AUTOMSET,NO is entered, Marc's AUTOMSET will be omitted. See Volume C of Marc for further details.
- PARAM,AUTOMSET,YES is not supported for SOL 200.
- RSSCON is not supported with AUTOMSET.

AUTOQSET

Default = NO

AUTOQSET=YES requests the automatic calculation of component modes without the need to define a q-set (generalized coordinates).

1. The calculation of component modes is attempted on all superelements including the residual structure.
2. All component modes are treated like SPOINTs which means that all component modes are “brought down” to and assigned to the q-set in the residual structure. In other words, component modes may not be assigned interior to a superelement.
3. Selected component modes may not be removed (constrained).
4. Since the generalized coordinates are automatically defined, the following entries may not be specified: QSETi, SEQSETi, SENQSET, or PARAM,NQSET.
5. This feature is not supported with:
 - a. Design optimization (SOL 200)
 - b. Aerodynamic analyses (SOLs 144, 145, 146)
 - c. Cyclic symmetry analyses (SOLs 114, 115, 116, 118)
 - d. SECSETi and SEBSETi Bulk Data entries that result in a fixed-free or free-free superelement. This restriction also applies to BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1entries in part (BEGIN SUPER) superelements.

PARAM,SQSETID may be used to specify the starting identification number of the automatically generated q-set degrees-of-freedom except when the EXTSEOUT command is present (see Remark 17. of the EXTSEOUT Case Control command). The default is 99000001.

AUTOSPC

Default = YES

This parameter is obsolete and replaced by the Case Control command [AUTOSPC \(Case\), 249](#). It is ignored for the residual structure in all nonlinear analysis solution sequences. SOL 400 only support the AUTOSPC Case Control command. For the residual structure in SOLs 106 and 129 see the description of parameter AUTOSPCR. AUTOSPC and the related parameters EPPRT, EPZERO, PRGPST, and SPCGEN are analogous to and described under the AUTOSPC Case Control command under the EPSING, EPS, PRINT, and PUNCH keywords, respectively. The specification of the AUTOSPC Case Control command overrides the specification of PARAM,AUTOSPC and its related parameters EPPRT, EPZERO, PRGPST, and SPCGEN.

AUTOSPCR

Default = NO (SOLs 106 and 129 only)

In SOLs 106 and 129 only, AUTOSPCR specifies the action to take when singularities exist in linear stiffness matrix of the residual structure after multipoint constraints are processed. AUTOSPCR=YES means that singularities will be constrained and AUTOSPCR=NO means they will not be constrained. It is recommended that all degrees-of-freedom attached to nonlinear elements be specified on ASETi entries. Parameters EPPRT, EPZERO, PRGPST, and SPCGEN may be used with AUTOSPCR.

AUTOSPRT

Default = YES

By default, free-free models will be automatically constrained for calculation of residual vectors (RESVEC Case Control command) as long as $F1 \leq 0.0$ on the EIGR (or EIGRL) Bulk Data entry. The auto-SUPPORT method may be deactivated by specifying a SUPPORTi entry, PARAM,AUTOSPRT,NO, or $F1 > 0.0$.

FZERO is the maximum frequency assumed for a rigid body mode. FZERO is used by the auto-SUPPORT method to extract the rigid body frequencies. The default is 1.0.

BAILOUT

Default = 0

See [MAXRATIO, 893](#).

BEAMBEA

Real, Default = 1000.0

Value Equivalent radius to be used for beam-beam contact problems. For tubes or round bars, use the outer radius. If the radii are different enter the largest outer radius. For beams, enter an equivalent radius calculated as follows:

$$I=0.5*(Ix+Iy)$$

$$R=sqrt(A/pi^2+2*I/A)$$

where A, Ix, Iy are the cross-section properties and $\pi^2 = \pi^2$.

BEIGRED

Default = YES

PARAM,BEIGRED,YES requests a more efficient method to reduce viscous damping matrices. CPU and disk space savings may be significant if there is a small number of viscous damping elements. BEIGRED is automatically disabled if the number of DOF connected to viscous dampers exceeds the value specified by PARAM,MAXDAMP.

MAXDAMP (default=1000) specifies the maximum number of DOF connected to viscous damping elements, such that an alternate reduction method is employed. When there are relatively few viscous dampers, an alternate reduction method is chosen automatically for efficiency. The efficiency is lost however if the number of viscous damping DOF exceeds about 1000. If the number of viscous damper DOF exceeds MAXDAMP, User Warning Message 9166 is printed and the alternate damping reduction logic is disabled. Also note that if MAXDAMP is exceeded, FASTFR is disabled.

BIGER, BIGER1, BIGER2, BIGER3

Default = 0.0

See S1.

BUCKLE

Default = -1

BUCKLE=1 requests a nonlinear buckling analysis in a restart run of SOLs 106 or 153. See the *MSC Nastran Handbook for Nonlinear Analysis*. (Not supported for SOL 600.)

BUCKLE=2 requests buckling in a SOL 106 cold start run. (Must be in Bulk Data for SOL 600.)

CASIEMA

Default = NO, SOL 400 CASI Solver only

When the CASI element-based iterative solver is specified (see the Case Control command [SMETHOD \(Case, 579\)](#)) SOL 400 skips various processes associated with the global structure stiffness matrix. The absence of a complete assembled stiffness matrix and its sub-sets can lead to a termination of the solution process in the NLSOLV module if the CASI solver detects a singularity. If the parameter is set to YES, SOL 400 performs all assembly and displacement set reduction operations involving the complete stiffness matrix allowing the

NLSOLV module to select either the matrix-based iterative solver or the sparse direct solver to continue in the event CASI detects a singularity.

Use a value of YES with caution as it adversely affects run-time performance. Substantial additional disk space could be required as well as longer run times. Keep in mind that invocation of the sparse direct solver for large solid element models may degrade performance even further.

This parameter only applies to SOL 400 usage of the CASI solver.

CASIMEST

Default = Yes

This parameter controls the memory estimation inside CASI iterative solver. (SOL 101, SOL 400 and SOL 200)

= No; Turns off memory estimation.

CASPIV

Default = 1.0e-10

Input DMAP parameter to the SOLVIT and NLSOLV modules for the CASI iterative solver. It is the pivot threshold for the CASI PCG (Preconditioned Conjugate Gradient) factorization. Pivot value less than CASPIV will exit with an error message.

CB1, CB2

Default = (1.0, 0.0)

CB1 and CB2 specify factors for the total damping matrix. The total damping matrix is:

$$[B_{jj}] = CB1 \cdot [B_{jj}^x] + CB2 \cdot [B_{jj}^2]$$

where $[B_{jj}^2]$ is selected via the Case Control command B2GG and $[B_{jj}^x]$ comes from CDAMPi or CVISC element Bulk Data entries. These parameters are effective only if B2GG is selected in the Case Control Section.

CDIF

Default = YES for shape optimization with or without property optimization.

Default = NO for property optimization only.

CDIF may be used to override the default finite difference scheme used in the calculation of pseudo loads in SOL 200. PARAM,CDIF,YES forces the selection of the central difference scheme used in the semianalytic approach regardless of the type of optimization requested. PARAM,CDIF,NO forces the selection of the forward difference scheme.

CDITER

Default = 0

If CDITER > 0, perform constrained displacement iterations in SOL 101. The value is the maximum number of iterations. If CDPRT=YES, print those negative displacements and tension forces which do not satisfy constraints. If CDPCH=YES, punch DMIG CDSHUT entries for final state; by default all gaps are closed. These can be used for initial conditions for restart. Potential contact points must be specified on the SUPORTi entries. The SUPORTi points must be in the residual structure. Optional DMIG entries to define the initial shut vector may be specified. Degrees-of-freedom that are specified on the SUPORT entry and have a value of 1.0 defined on the DMIG,CDSHUT entry will be considered closed initially.

If the DMIG,CDSHUT entry is not supplied, then all degrees-of-freedom specified on the SUPORT entries will be considered shut initially. A fatal message will be issued if this parameter is used and PARAM,INREL is specified.

CDPCH

Default = NO

See [CDITER, 797](#).

CDPRT

Default = YES

See [CDITER, 797](#).

CFDIAGP

Default = NO

If YES, randomly deleted CFAST elements will be printed. (See CFRANDEL)

CFRANDEL

Default = 0.

Represents a percent, expressed as a decimal fraction, of the number of CFAST elements to be randomly deleted.

CHECKOUT

Default = NO

CHECKOUT=YES requests a model checkout in SOLs 101 through 200. See [Geometry Processing in SubDMAP PHASE0](#) in the MSC Nastran Reference Guide. The run will terminate prior to phase 1 of superelement analysis. The PARAM,POST options are also available with PARAM,CHECKOUT,YES. The following options and their user parameters are also available with PARAM,CHECKOUT,YES:

1. PARAM,PRTGPL,YES

Prints a list of external grid and scalar point numbers in internal sort. It also lists external grid and scalar point numbers along with the corresponding sequence numbers in internal sort. The sequence numbers are defined as (1000*external number) and will reflect any user-requested resequencing.

2. PARAM,PRTEQXIN,YES

Prints a list of external and internal grid and scalar numbers in external sort. It also lists external grid and scalar numbers with the corresponding coded SIL number in external sort. The coded SIL numbers are defined as:

$$10 \cdot \text{SIL} + \begin{cases} 1 & \text{for grid point} \\ 2 & \text{for scalar point} \end{cases} \quad (1)$$

The SIL numbers correspond to degrees-of-freedom, i.e., one SIL number for scalar point and six SIL numbers for a grid point.

3. PARAM,PRTGPDT,YES

Prints, for each grid and scalar point, the following information in internal sort:

- Coordinate system ID in which grid point geometry is defined (ID=-1 for scalar points).
- Spatial location of grid points in the “CP” coordinate system. For scalar points, all entries are zero.
- Coordinate system ID for grid point displacements, forces, and constraints (ID=0 for scalar points).
- Permanent single-point constraints defined on GRID Bulk Data entries. A zero is entered for scalar points.

4. PARAM,PRTCSTM,YES

Prints for each coordinate system type the transformation matrix from the global to the basic coordinate system, and the origin of the indicated coordinate system in the basic coordinate system. Coordinate system types are: 1 = rectangular; 2 = cylindrical; 3 = spherical.

5. PARAM,PRTBGPDT,YES

Prints all grid and scalar points listed in internal sort with their x, y, and z coordinates in the basic coordinate system. In addition, the coordinate system ID for grid point displacements, forces, and constraints is indicated for each grid point (ID=-1 for scalar points). The x, y, and z coordinates of scalar points are zero.

6. PARAM,PRTGPTT,YES

Prints, for each temperature load set, information on element and grid point temperatures.

7. PARAM,PRTMGG,YES

Prints the g-size mass matrix labeled by grid point/degree-of-freedom.

8. PARAM,PRTPG,YES

Prints the g-size load vectors labeled by grid point/degree-of-freedom.

9. The summation of forces and moments of applied loads in the basic coordinate system is automatically output for each loading condition requested in the Case Control Section. Related to parameters GPECT and PROUT, and Case Control command ELSUM.

CK1, CK2

Default = (1.0, 0.0)

CK1 and CK2 specify factors for the total stiffness matrix. The total stiffness matrix (exclusive of GENEL entries) is

$$[K_{jj}^x] = CK1 \cdot [K_{jj}^z] + CK2 \cdot [K_{jj}^2]$$

where $[K_{jj}^2]$ is selected via the Case Control command K2GG and $[K_{jj}^z]$ is generated from structural element (e.g., CBAR) entries in the Bulk Data. These are effective only if K2GG is selected in Case Control. A related parameter is CK3.

Note: Stresses and element forces are not factored by CK1, and must be adjusted manually.

CK3

Default = (1.0, 0.0)

CK3 specifies a factor for the stiffness derived from GENEL Bulk Data entries. The total stiffness matrix is

$$[K_{jj}] = [K_{jj}^x] + CK3 \cdot [K_{jj}^y]$$

where $[K_{jj}^y]$ comes from the GENEL Bulk Data entries and $[K_{jj}^x]$ is derived using PARAMs CK1 and CK2. CK3 is effective only if GENEL entries are defined. Related parameters include CK1 and CK2.

CLOSE

Default = (1.0)

A value of 1.05 will set natural frequencies within 5% of each other as close. See SCRSPEC.

CM1, CM2

Default = (1.0, 0.0)

CM1 and CM2 specify factors for the total mass matrix. The total mass matrix is

$$[M_{jj}] = CM1 \cdot [M_{jj}^x] + CM2 \cdot [M_{jj}^2]$$

where $[M_{jj}^2]$ is selected via the Case Control command M2GG and $[M_{jj}^x]$ is derived from the mass element entries in the Bulk Data Section. These are effective only if M2GG is selected in the Case Control Section.

COMPMMAT

Default = NO, SOL 106 and SOL 400 only

In nonlinear statics (SOLs 106, 400 ANALYSIS=NSTAT), composite materials compute temperature-dependent properties for the plies only at the reference temperature given on the PCOMP Bulk Data entry. The ply properties are smeared and used for all load steps, regardless of whether the temperature is changing through application of thermal loads.

If the parameter is set to YES, the temperature-dependent properties for the plies are updated and smeared at the current temperature for each load step. For CQUAD4 and CTRIA3 elements with offsets, the large rotation offset method specified by "MDLPRM, OFFDEF, LROFF" in SOL 400 must be used.

If the parameter is set to NONSMEAR, the temperature-dependent properties for the plies are updated at the current temperature for each load step. This option, only available for the CQUADR and CTRIAR elements, is an alternative to the smeared approach.

This parameter only applies to SOLs 106 and 400 ANALYSIS=NSTAT, and only applies to composite CQUAD4, CTRIA3, CQUADR and CTRIAR elements that are not associated with the enhanced nonlinear materials (PSHLN1).

Prior to Nastran version 2019, when COMPMATT was set to YES, material nonlinear was assumed even when no composite plies had temperature-dependent materials or when there was no MATTi type entries associated with temperature dependent plies. This has caused confusion as to when SOL106 or SOL400 is doing linear or nonlinear material solutions. Starting in V2019, PARAM, COMPMATT, YES will only turn on material nonlinear if there are active MATTi entries as well.

Additionally, prior to Nastran version 2019, when PARAM, COMPMATT, YES was used in conjunction with TEMPP1 entries, ply stress computation used only the reference plane temperature. Starting in V2019, ply stress computation will use reference plane temperature plus the thermal gradient term applied at the center of each ply. (To revert to the old method, NASTRAN SDRCMPTP=1 may be added to the Nastran section.) The enhanced nonlinear materials (PSHLN1) do not support the TEMPP1 entry.

CONFAC

Default = 1.E-5

In superelement analysis, CONFAC specifies the tolerance factor used in checking the congruence of the location and displacement coordinate systems of the boundary points between image superelements and their primaries (see the Bulk Data entry, [CSUPER, 1659](#)). Specification of this parameter is recommended instead of DIAG 37 (DIAG 37 ignores User Fatal Messages 4277 and 4278).

COSUBCYC

Default = 1, SOL 700 only

Controls the growth of the subcycling interval in the coupling surface.

Example:

PARAM,COSUBCYC,2

Remarks:

1. The subcycling algorithm automatically estimates the number of subcycles to be used. This is updated throughout the calculation. This parameter controls how much the number of subcycles can grow. For example, COSUBCYC is set to 1, and the current number of time steps between updates of the coupling geometry is 4. If the solver estimates that the subcycling interval should be 7, the subcycling interval is increased by 1 until a value of 7 is reached.
2. There is no control on the amount by which the subcycling interval can decrease.

COSUBMAX

Default = 0, SOL 700 only

Defines the maximum number of subcycles that can occur in Euler/Lagrange coupling. During a subcycle, the geometry of the coupling surface is not updated.

Example:

PARAM,COSUBMAX,10

Remarks:

1. Updating the coupling geometry takes a lot of CPU time. Subcycling gives substantial savings in CPU time for coupled calculations.
2. The smaller the value of this parameter, the greater the accuracy of the analysis and the greater the cost. Conversely, larger values offer significant CPU savings, but very large values give incorrect results.
3. If the geometry of the coupling surface is changing rapidly, smaller values of PARAM,COSUBMAX should be used.

COUPMASS

Default = -1

COUPMASS > 0 Requests the generation of coupled rather than lumped mass matrices for elements with coupled mass capability, as listed in [Table 3-1](#) in the *MSC Nastran Reference Guide*. This option applies to both structural and nonstructural mass for the following elements: CBAR, CBEAM, CONROD, CQUAD4, CHEXA, CPENTA, CPYRAM, CQUAD8, CROD, CTETRA, CTRIA3, CTRIA6, CTRIAX6, CTUBE.

COUPMASS < 0 (Default) causes the generation of lumped mass matrices (which may include torsion inertia for beam elements, and some coupling if there are beam offsets) for all of the above elements. However, if SYSTEM(414) is greater than zero, (Default = 0) then the lumped mass matrices will contain translational components only for the CBAR and CBEAM elements.

P-elements are always generated with coupled mass and are not affected by COUPMASS.

CP1, CP2

Default = (1.0, 0.0)

The load vectors are generated from the equation

$$\{P_j\} = CP1 \cdot \{P_j^x\} + CP2 \cdot \{P_j^2\}$$

where $\{P_j^2\}$ is selected via the Case Control command P2G, and $\{P_j^x\}$ comes from Bulk Data static load entries. These parameters are effective only if P2G is selected in the case control section.

CQC

Default = -1

See [SCRSPEC, 987](#).

CURV

Default = -1

PARAM,CURV,1 requests that the CTRIA3 and CQUAD4 element stress and/or strain output be computed in a material coordinate system (normal output is in the element or basic coordinate system) and/or to interpolate it to grid points. (CQUAD4 element corner stress output is not supported.)

The integer parameter OG controls the calculation of stress and/or strain data at grid points. If OG is set to -1, the calculation for stresses and/or strain data at grid points is not performed. The default value of zero provides the calculation of these quantities at those grid points to which the selected elements connect.

User parameters S1G, S1M, S1AG, and S1AM, set to 1, request the printout of stresses at grid points, stresses in the material coordinate system, strains at grid points and strains in the material coordinate system, respectively.

The integer parameter OUTOPT may be set in accordance with the below options to select print, punch, and/or plotter output for stress and/or strain data that are computed in user-defined material coordinate systems.

OUTOPT Value	Description
0	Default-standard MSC Nastran device codes are used.
1	Print only
2	Plot only
4	Punch only

The above values may be combined additively to select two or more forms of output. For example, OUTOPT=6 requests both plot and punch output. Related parameters include BIGER, CURVPLOT, DOPT, NUMOUT, NINTPTS, S1G, S1M.

For stress and/or strain/curvature output in a user-defined material coordinate system MCSID must be defined on MAT1 and MAT2 Bulk Data entries. The values of MCSID reference CORDiR, CORDiC, and CORDiS Bulk Data entries. A value of zero for MCSID does not imply the basic coordinate and will eliminate all elements which reference the MATi from the subject calculations.

1. If these data are requested at the element centers, the program will compute the unit vector i_m along the T1 or x-axis of the material coordinate system, and compare

$$|\bar{n} \cdot i_m|$$

for each element that references the material coordinate system, where n is the normal to the surface of the element. If

$$|\bar{n} \cdot i_m|^2 \geq .4$$

the projection of the y-axis on the surface of element is taken as the reference axis. Otherwise, the projection of the x-axis on the surface of the element is taken as the reference axis. The angle between the x-axis of the element coordinate system and the projection of the selected reference axis of the material coordinate system is used to transform the stress and/or strain data into the material coordinate system at the element centers.

2. If, on the other hand, the user requests these data at the grid points to which the elements connect the program will interpolate the results from (a) to the grid points to which the elements connect. The parameter NINTPTS=N, the stress and/or strain data at the N closest element centers to the grid point in question will be used in the interpolation. The program may include more than N points in the interpolation if the distance of other element centers is not more than 10% greater than the closest N element centers.

The following specifies the output headings for stresses and/or strains in the material coordinate system.

Element stresses (PARAM,S1M,1)

1. Available in CQUAD4 and CTRIA3 elements
2. Page headings:
 STRESSES IN QUADRILATERAL ELEMENTS (CQUAD4)
 STRESSES IN TRIANGULAR ELEMENTS (CTRIA3)
3. Under the column FIBER DISTANCE:
 Z1 is replaced by MCSID.
 Z2 is replaced by 1.0 if the x-axis of the material coordinate system is selected as the reference axis, and by 2.0 if the y-axis of the material coordinate system is selected as the reference axis.

Grid point stresses (PARAM,S1G,1 and PARAM,OG,1)

1. Available for CQUAD4 and CTRIA3 elements
2. Page heading:
 STRESSES AT GRID POINTS
3. Under the column are:

$$\left\{ \begin{array}{c} \text{MAT1} - \text{COORD1} - \text{ID} \\ \text{PROJ-CODE} \end{array} \right\}$$
 Z1 is replaced by MCSID.
 Z2=A+10*N where A is 1.0, 2.0, or 3.0, depending on whether the x-, y-, or z-axis of the material coordinate system is most nearly normal to the projected plane of the field of elements involved in the calculation.

Element strains (PARAM,S1AM,1)

1. Available for CQUAD4 and CTRIA3 elements
2. Page headings:

STRAINS IN QUADRILATERAL ELEMENTS (CQUAD4)

STRAINS IN TRIANGULAR ELEMENTS (CTRIA3)

3. Under the column FIBER DISTANCE:

Z1 is replaced by MCSID.

Z2 is replaced by 1.0 if the x-axis of the material coordinate system is selected as the reference axis, and by 2.0 if the y-axis of the material coordinate system is selected as the reference axis.

Grid point strains (PARAM,S1AG,1 and PARAM,OG,1)

1. Available for CQUAD4 and CTRIA3 elements.

2. Page heading:

$Z2=A+10^*N$ where A is 1.0, 2.0, or 3.0, depending on whether the x-, y-, or z-axis of the material coordinate system is most nearly normal to the projected plane of the field of elements involved in the calculation.

STRAINS AND CURVATURES AT GRID POINTS

3. Under the column are:

$$\left\{ \begin{array}{c} \text{MAT1} - \text{COORD1} - \text{ID} \\ \text{PROJ-CODE} \end{array} \right\}$$

Z1 is replaced by MCSID.

CURVPLOT

Default = -1

PARAM,CURVPLOT,1 requests that x-y (or curve) plots whose abscissas are a sequence of grid points and whose ordinates may be displacements, loads, SPC forces, or grid point stresses. To obtain stress plots, set the CURV parameter to +1. The default for DOPT is the length between grid points.

Specify the XYOUTPUT Case Control command in the usual manner, replacing the point ID with the SID of SET1 Bulk Data entries.

The SET1 Bulk Data entries must contain unique SIDs for each set of grid points to be plotted.

User requests for xy-plots of output quantities appear in the Case Control Section in the standard form. For example,

```
.
.
.
OUTPUT (XYOUT)
.
.
XYPLOT DISP 1/4 (T3)
.
.
XYPLOT SPCF 2/5 (T1)
.
.
BEGIN BULK
```

The first XYPILOT command will produce an xy-plot from the displacement output of subcase 1. The abscissa of the curve will reflect the grid point IDs listed on the SET1 entry with an SID of 4, and the ordinate will reflect the T3 component of displacement at these grid points. The second XYPILOT command will produce an xy-plot whose ordinates are the T1 components of the forces of constraint in subcase 2 at the grid points listed on the SET1 entry with an SID of 5.

The user has some degree of control over the scaling of the abscissas on these xy-plots. This control is exercised through the parameter DOPT on a PARAM Bulk Data entry. The legal values of this parameter provide the following scaling options for the abscissas.

Value of DOPT	Scaling for Abscissa
0 (Default)	$\ g_j - g_i\ $
1	$ x_j - x_i $
2	$ y_j - y_i $
3	$ z_j - z_i $
4	1.

Thus, the default value of DOPT will place the first grid point listed on the referenced SET1 entry at the origin, and subsequent grid points will be located along the abscissa at intervals proportional to the distance between that grid point and its predecessor. Values of DOPT equal to 1, 2, or 3 will scale the abscissa so that the interval between adjacent grid points is proportional to the difference in the X, the Y, and the Z components of the subject grid points respectively. DOPT=4 will space the grid points equally along the abscissa.

CWDIAGP

Default = NO

For CWELD element: prints elements randomly deleted if set to yes.

CWRANDEL

Default = 0.0

For CWELD element: if not zero, then it specifies as a decimal percent for the number of fasteners to randomly delete.

DBALL

Default = DBALL

By default, all data to be stored on the database for restart purposes will be located on the DBALL database set DBset). These parameters permit the storage of some data blocks on DBsets other than DBALL, which are defined by the user and specified on the INIT File Management statement. Any or all of these parameters may be set to SCRATCH in order to reduce overall disk space usage; e.g., PARAM,DBUP,SCRATCH or PARAM,DBALL,SCRATCH. However, automatic restarts will be less efficient because data normally assigned to a permanent DBset will have to be recomputed.

A unique value for each superelement may be specified in the Case Control Section for the parameters DBALL, DBDN, DBRCV, and DBUP. Certain DBsets may be taken offline depending on which phase (see [Summary of Solution Sequence Operations](#) in the *MSC Nastran Reference Guide*) of superelement analysis is being performed. PARAM,DBALL specifies the default value for parameters DBDN, DBUP, and DBRCV.

The DBDN DBset contains data blocks necessary for “downstream” processing. For example, the stiffness, mass, damping, and static loads matrices that have been reduced to the boundary of the superelement are stored in this DBset.

The DBRCV DBset contains data blocks that must be online during the first pass through data recovery (Phase 3). These data blocks are used to recover the total displacement vector u_g of the superelement. This operation is performed by the SSG3 and SDR1 modules. On subsequent data recovery restarts, this DBset may be taken offline. Its default is determined from the value of DBUP.

The DBUP DBset contains data blocks necessary for “upstream” processing. For example, the geometry and property tables along with the stiffness, mass, damping, and static loads matrices related to the interior grid points of the superelement are stored in this DBset. These matrices and tables must be online during the reduction (Phase 1) and data recovery (Phase 3) of the superelement.

The IFP DBset contains data blocks that are required for all phases of the analysis. These data blocks are related to the entire model; examples are Bulk Data, superelement map, IFP module outputs, and resequenced grid points. This DBset must be online for all runs.

PARAM,DBALL also specifies the default value for PARAMs DBEXT and DSO described below.

DBCCONV

Default = XL

See POST=0.

DBCdiag

Default = 0

See POST=0.

DBCOVWRT

Default = YES

See POST.

DBDICT

Default = -1

Controls the printout of the database directory at the beginning and end of the run. See DBDICT FMS statement description in Section 2. If DBDICT=0, then the database directory will be printed at the start of the run. If DBDICT=1, then the directory will be printed at the end of the run. If DBDICT \geq 2, then it will be printed at the beginning and end of the run. If DBDICT=-1 (the default), the directory is not printed at the beginning or end of the run.

If multiple versions and/or projects exist on the database, then the parameters DBDRPRJ and DBDRVER allow the user to select the desired project and version, respectively. The appropriate values may be found in

the Project/Version Table that is printed upon restart. If DBDRVER=0 (or DBDRPRJ=0), then the current version (or project) is selected. If DBDRPRJ=-1 (or DBDRVER=-1), then all projects (or versions) are selected.

DBDN

Default = value of PARAM,DBALL.

See DBALL.

DBDRPRJ

Default = 0

Specifies the desired project-identification number. See DBDICT.

DBDRVER

Default = 0

Specifies the desired version-identification number. See DBDICT.

DBEXT

Default = value of PARAM,DBALL

Specifies the DBset location to store the external superelement information. External superelement information is generated by the EXTSEOUT Case Control command and the user PARAMeter EXTOUT.

DBRCV

Default = value of PARAM,DBUP.

See DBALL.

DBUP

Default = value of PARAM,DBALL.

See DBALL.

DDRMM

Default = 0

DDRMM is only recognized if PARAM,SPARSEDRL,NO is specified.

By default, the matrix method of data recovery is used in the modal transient and frequency response solutions. DDRMM=-1 will force calculation of complete g-set solution vectors by the mode displacement method and is required for FATIGUE analysis using SOL 108 or SOL 111 if PARAM,SPARSEDRL,no is also specified.

DELCLUMP

Default = 0.5, SOL 700 only

Format:

PARAM,DELCLUMP,VALUE

Example:

PARAM,DELCLUMP,0.1

Prevents small clumps in the Euler mesh from determining the time step and prevents the leakage of small masses to isolated regions.

VALUE Value of DELCLUMP. See Remark 1. (Real ≥ 0.0)

Remarks:

1. Material in Eulerian elements of a clump with:
 $f_{\text{func}} < \text{DELCLUMP} \cdot f_{\text{blend}}$
is eliminated
2. See also parameter FBLEND.

DESPCH

Default = 0

For sizing and shape topography optimization, DESPCH specifies in SOL 200 when the optimized (updated) bulk data entries are written to the PUNCH file. Currently, all the property entries, material entries, and connectivity entries that can be designed and DESVAR, DRESP1, and GRID entries can be written. Notice that the DRESP1 entries will be written if only when a mode tracking is performed and the DRESP1 responses have type FREQ or EIGN.

For topology/topometry optimization, DESPCH specifies when the topology optimized element density values (or topometry optimized values) are written to the element result file jobname.des. This file can be directly read in PATRAN or third party post-processor to displace and animate the topology/topometry optimization results.

< 0	Never
= 0	at the last design cycle only (Default)
> 0	at every design cycle that is a multiple of DESPCH and the last design cycle. For example, if n=2 and the maximum number of design cycles is 5 (DESMAX=5 on the DOPTPRM entry), then, DESVAR and GRID entries at design cycle 2, 4, and 5 are written in the punch file.

DESPCH1

Default = 6

DESPCH1 specifies in SOL 200 the amount of data to be written to the .pch and .des file. A positive DESPCH1 value request large field formats while a negative value requests small field formats. For a shape

optimization job, if DESPCH1<>0, the updated GRID entries of the whole model will be written in the .pch file.

Descriptions of various DESPCH1 values are given below:

- 0 Write no data.
- ± 1 Write the property entries that are designed.
- ± 2 Write all the property entries of a given type when one or more property of that type is designed.
- ± 4 Write DESVAR and DRESP1 entries.
- $\pm n$ Write combine quantities by summing the DESPCH1 values. For example, $n=1+4=5$ requests writing all the designed property entries, DESVAR and DRESP1 entries to the .pch file for normal modes analysis.
- > 0 Write all (topology designed and non-designed) element density values to the topology element density history file jobname, des.
- < 0 Write topological designed element density values to the topology element density history file jobname.des.

DFREQ

Default = 10^{-5}

DFREQ specifies the threshold for the elimination of duplicate frequencies on all FREQi Bulk Data entries. Two frequencies, f_1 and f_2 , are considered duplicated if

$$|f_1 - f_2| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

where f_{MAX} and f_{MIN} are the maximum and minimum excitation frequencies of the combined FREQi entries.

DIROUT

Default = NO

See CQC under [SCRSPEC, 987](#).

DOPT

Default = 0

See CURVPLOT.

DPEPS

Default = 1.0E-4

In SOL 200, if any difference between the property value on the property entries and the value calculated from the design variable values on the DESVAR entry (through DVCRELi, DVMRELi, DVPRELi relations) is greater than DPEPS, the design model values override the analysis values. If all the differences are less than DPEPS, analysis results from a previous run are accepted in a subsequent sensitivity/optimization task,

thereby avoiding a reanalysis. The PTOL parameter on the DOPTPRM entry is a related parameter that checks the maximum difference.

DPHFLG

Integer ≥ 0 ; Default = 0

Select the method for performing eigenvector sensitivity analysis. Unless repeated eigenvalues are anticipated, the default value is recommended.

DPHFLG	0	Nelson's method (Default)
	1	Subspace iteration method. Recommended when there are repeated roots.
	2	High's method, # of modes for iteration = $\min(2n, n+8, m)$ n: the highest constrained mode m: the number of modes request by EIGR
	3	High's method will all modes requested by EIGR used in the iterator. DPHFLG = 2 and 3 have further control parameters
ITERATE	yes	Do iteration, for improved sensitivity value. (Default)
	no	No iteration, equivalent to Fox's method and provides the most efficient method, but not the most accurate.
ITMAX		Maximum number of iteration. (Default = 10)
TOL		Tolerance for convergence in iteration. (Default = 1.0e-4)
LAMBDA		Shift factor. (Default = 0.0)
KORTHO	no	Use mass for Gram Schmidt orthogonalization. (Default)
	yes	Use stiffness K for Gram Schmidt orthogonalization.
ITRPRNT	no	Do not print sensitivity for each iteration. (Default)
	yes	Print sensitivity for each iteration.
ITFPRNT	no	Do not print final sensitivity, leave print to SOL 200.
	yes	Print final sensitivity inside High's method computation
MDOF	no	Do not reduce DPHI to USET 'U6' DOF. (Default)
	yes	Reduce DPHI to USET 'U6' DOF.

DSNOKD

Default = 0.0

DSNOKD specifies a scale factor to the differential stiffness matrix in buckling design sensitivity analysis. If DSNOKD > 0.0 , the effect of the differential stiffness matrix is included in buckling the design sensitivity analysis.

If PARAM,DSNOKD,1.0 is specified in SOL 200, the differential stiffness sensitivity calculation is performed more accurately; i.e., the change in the stiffness matrix due to the changes in the displacements are computed. However, the calculation is more expensive than with PARAM,DSNOKD,0.0.

Non-zero values of PARAM,DSNOKD cannot be used in SOL 200 with multiple buckling design subcases less each subcase contains the same STATSUB command.

DSO

Default = value of PARAM,DBALL

Specifies the DBset location to store datablocks created for design sensitivity and optimization in SOL 200.

DSZERO

Default = 0.0

DSZERO specifies the minimum absolute value for the printout of design sensitivities.

DV3PASS

Integer, Default = 1 for SOL 200 with nonlinear property sensitivity evaluation.

DV3PASS controls the number of optimizer internal cycles must pass before the re-evaluation of sensitivity of nonlinear properties, such as I1/I2/J of PBARL/PBEAML/PBRSECT/PBMSECT. With default value of 1, the sensitivity of nonlinear properties is computed for every internal cycle of optimizer. Setting DV3PASS to 10 causes the sensitivity to be evaluated at optimizer internal cycles of 1, 11, 21 and etc. The sensitivity of nonlinear properties is considered as invariants if a sufficiently large number, such as 100, is assigned to DV3PASS.

DYNSPCF

Default = NEW

PARAM,DYNSPCF,NEW requests that mass and damping coupled to ground be included in the SPCForce calculations for the linear dynamic solutions: SOLs 103, 107 through 112, 115, 118, 145, 146, and 200. OLD neglects these effects and gives the same SPCForce results obtained in versions prior to Version 68.

Note: Some versions of Dytran have trouble with permanent constraints and issue a bogus error message in the d3hsp file. This is the case for MSC Nastran 2005 r2.

EIGFILT

Default=1.0E-13

Threshold for zero-frequency eigenvalue for ACMS VERSION=NEW (module ACMS1).

ENFMETH

Default = TOTAL or ABS

This parameter controls the *solution method* when dynamic enforced motion analysis via SPC/SPCD is used in SOLs 108, 109, 111, 112, 146 and 200.

The total solution of a dynamic enforced motion analysis using SPC/SPCD can be regarded as a combination of a static enforced motion solution (similar to what is done in SOL 101) and a dynamic enforced motion solution that is relative to this static-based solution. The default value of TOTAL/ABS implies that the program solves directly, *in one step*, for the TOTAL solution of the dynamic analysis which includes both the static-based solution and the dynamic solution that is relative to the static-based solution. If the value is specified as REL, then the program obtains the total solution of the dynamic analysis *in two steps*, by first solving for the static-based solution and then solving separately for the dynamic solution RELATIVE to the static-based solution. In general, the TOTAL/ABS and REL solution methods both yield essentially the same results.

The TOTAL/ABS solution method is computationally more efficient. This is also the only method that is meaningful and that should be employed when a problem involves the use of NOLINI or NLRGAP entries. An important point to note regarding this method is that, for modal dynamic analysis, residual vectors are absolutely critical in order for this method to get correct answers.

The REL solution method, though less efficient, may be more accurate for transient solutions and for modal frequency response solutions at very low forcing frequencies. Also, for modal dynamic analysis, this method is not as critically dependent on residual vectors as the TOTAL/ABS solution method. In addition, the current implementation of the REL solution method does not support fluid DOFs with enforced motion. So, if fluid DOF are among the DOFs with enforced motion, the job will be switched to the ABS solution method automatically.

It should be emphasized here that PARAM,ENFMETH and PARAM,ENFMOTN are completely separate, distinct and independent of each other and should not to be confused with each other. The former controls the *solution method* when dynamic enforced motion analysis via SPC/SPCD is used while the latter controls how the results of such an analysis are *output*.

ENFMOTN

Default = TOTAL or ABS

This parameter controls how the results of the analysis are *output* when dynamic enforced motion analysis via SPC/SPCD is used in SOLs 108, 109, 111, 112, 146 and 200.

The total solution of a dynamic enforced motion analysis using SPC/SPCD can be regarded as a combination of a static enforced motion solution (similar to what is done in SOL 101) and a dynamic enforced motion solution that is relative to this static solution. The default value of TOTAL/ABS implies that the output results of the analysis represent the TOTAL solution of the dynamic analysis which includes both the static solution and the dynamic solution that is relative to this static solution. If the value is specified as REL, then the output results represent the dynamic solution RELATIVE to the static solution.

It should be noted that, in general, the static solution mentioned above may *not* be a stress-free solution. Therefore, in general, the usage of PARAM,ENFMOTN,TOTAL (or ABS) and PARAM,ENFMOTN,REL in an SPC/SPCD enforced motion analysis may give *different results not only for displacements, but also for stresses*.

In general, it is not easy to identify all cases wherein the static solution represents a stress-free solution. However, one can give examples of such cases.

Thus, for instance, if the SPC/SPCD enforced motion is specified at a single grid point and there are no constraints at any other grid point, then the static solution will be a stress-free solution.

Similarly, if the SPC/SPCD enforced motion is specified at more than one grid point for a specific grid point component, but the magnitude and direction of the enforced motion is the same for all of the enforced motion points and there are no other constraints in the model, then all of the enforced motion points will move in unison and the static solution will be a stress-free solution.

In contrast to the above, consider the situation in which the SPC/SPCD enforced motion is specified at more than one grid point for a specific grid point component, but the magnitude and direction of the specified enforced motion is *different* for the various enforced motion points. In this case, the static solution will *not* be a stress-free solution.

If the static solution represents a stress-free solution, then it can be regarded as the base motion of the model. In such cases, the usage of PARAM,ENFMOTN,TOTAL (or ABS) and PARAM,ENFMOTN,REL will give *different* displacements, but the same stresses. Further, in modal dynamic analysis for such cases, the results using PARAM,ENFMOTN,TOTAL (or ABS) will give the *same* results as those obtained by using the large mass approach with all zero (or nearly zero) frequency modes included and the results using PARAM,ENFMOTN,REL will give the same results as those obtained by using the large mass approach with all zero (or nearly zero) frequency modes excluded.

If the static solution does *not* represent a stress-free solution, , the usage of PARAM,ENFMOTN,TOTAL (or ABS) and PARAM,ENFMOTN,REL will give *different results not only for displacements, but also for stresses*. In this case, the results from the usage of PARAM,ENFMOTN,REL do not have the same significance as in the case of the stress-free static solution except for the fact that these results represent the dynamic solution RELATIVE to the static solution.

Note: Currently, the usage of PARAM,ENFMOTN,REL is not supported when PARAM,ENFMETH,ABS is specified (or implied). If this usage is employed, the program terminates the execution with an appropriate fatal message.

EPPRT

Default = 1.E-8

EPPRT is analogous to and described under the EPSSING keyword on the AUTOSPC Case Control command. The specification of the AUTOSPC command overrides the specification of PARAM,EPPRT.

EPSILONT

Default = SECANT

In nonlinear statics (SOL 106 and SOL 400), thermal loads are computed using the secant method:

$$\epsilon_T = \alpha_{\text{load}}(T_{\text{load}} - T_{\text{ref}}) - \alpha_{\text{init}}(T_{\text{init}} - T_{\text{ref}})$$

If the parameter is set to INTEGRAL, thermal loads are computed using the integral method:

$$\epsilon_T = \int_{T_{\text{init}}}^{T_{\text{load}}} \alpha(T) dT$$

This parameter only applies to SOLs 106 and 400 ANALYSIS=NLSSTAT, and only applies to CQUAD4, CTRIA3, CQUADR, and CTRIAR elements that are not associated with the enhanced nonlinear materials (PSHLN1).

EPZERO

Default = 1.E-8

EPZERO is analogous to and described under the EPS keyword on the AUTOSPC Case Control command. The specification of the AUTOSPC command overrides the specification of PARAM,EPZERO.

ERPC

Default = 1.0

This parameter represents the phase speed of the fluid for Equivalent Radiated Power (ERP) analysis. This parameter may alternatively be set using the ERP Case Control command.

ERPREFDB

Default = 1.0

This parameter represents a peak reference ERP value used in the computation of ERP in units of dB. The dB level is defined as:

$$\text{ERPdB} = 10 \log \left(\text{RHOCP} \cdot \frac{\text{ERP}}{\text{ERPREFDB}} \right)$$

This parameter may alternatively be set using the ERP Case Control command.

ERPRHO

Default = 1.0

This parameter represents the fluid density for Equivalent Radiated Power (ERP) analysis. This parameter may alternatively be set using the ERP Case Control command.

ERPRLF

Default = 1.0

This parameter represents a Radiation Loss Factor scale factor for the Equivalent Radiated Power (ERP) analysis. This parameter may alternatively be set using the ERP Case Control command.

ERROR

Default = -1

For SOLs 111-112, when the constraint modes have non-zero generalized force the parameter ERROR set to 0 causes the fatal error exit to be branched over and the dynamic response is computed and output. This option is intended for model checkout work, where bad results are better for diagnostic work than no results at all. This parameter is a traditional feature also used in other, similar circumstances.

ESLFSAV

Character, Default = NO

ESLFSAV = YES requests that all the intermediate files from an ESLNRO job be saved on disk. The destination of these files can be directed with the 'sdir=' option on a Nastran submittal command line.

ESLMOVE

Integer, Default = 0

ESLMOVE = 0 selects a move limit scheme that poses restrict lower and upper bounds on design variables during the linear response optimization. The range of the bounds is determined by

$$X_k^L = \max(X_o^L, X_{k-1} - MOVE)$$

$$X_k^U = \min(X_o^U, X_{k-1} + MOVE)$$

$$MOVE = \max(DXMIN, abs(X_{k-1}) \cdot DELXESL)$$

ESLMOVE = 1 selects a move limit scheme that scales back the design move proposed from a linear response optimization. The amount of scaling back is determined by

$$X_k^* = X_{k-1} + (X_{k-1}^I - X_{k-1}) \cdot DELXESL$$

where X_k^* is the scaled design variable for the k-th design cycle, X_{k-1} is the design variable at (k-1)th design cycle, X_{k-1}^I is the proposed design from the linear optimization solution at (k-1)th design cycle.

Parameters DXMIN and DELXESL can be specified on the DOPTPRM entry.

ESLLCOMP

Default = No

ESLLCOMP selects types of compliance response to be included in the design task. The nonlinear compliance response is defined using a DRESP1 entry with RTYPE=COMP for the ESLNRO topology optimization tasks. As the default, it is computed by the product of the applied nonlinear loads and corresponding nonlinear displacement components. Alternatively, ESLLCOMP=YES selects a linear compliance response that computed as the total work done by the equivalent static loads on the linear system.

ESLMPC1

Default = 0

This parameter applies only to the ESLNRO jobs with 3D contact. Its default has different meanings depending on the type of contact applications. As the default, for a glued contact ESLNRO job, a linear response optimization task will include a set of MPC entries that are created from the nonlinear analysis. For a touching contact ESLNRO job, the linear response optimization task will not include the MPC entries by default. Setting ESLMPC1 to a positive number will turn on the MPC inclusion.

- 1 uses the MPC entries created from the nonlinear analysis at the converged nonlinear analysis.
- 2 uses the MPC entries created at the beginning of the very first nonlinear analysis.

ESLOPTEX

Default = 0

This parameter allows the user to perform an ESLNRO job at a targeted exit point. The allowable values of ESLOPTX are listed below with their description.

- 0 Do not exit. Proceed with ESLNRO nonlinear response optimization.
- 1 Exit after the initialization of the analysis and design model but before nonlinear FE analysis begins.
- 2 Exit after nonlinear FE analysis ends.
- 3 Exit after design constraint evaluation and screening.

ESLPRT

Default = 0

ESLPRT specifies how often the ESLNRO results are printed in the .f06 file and saved in the .xdb file. By default, the program will print the results to the .f06 file at the first and the last design cycles and save the results to .xdb (or .op2) at the first and last design cycles on the disk. (See ESLPRT1 for selection of result contents.)

- > 0 then the results are printed at the first design cycle; at every design cycle that is a multiplier of ESLPRT; and the last design cycle.
- < 0 the no results are printed and saved.

ESLPRT1

Default = 7

ESLPRT1 specifies what type of results to be written to the .f06 and to .xdb (or .op2). It may take any of the following base values or a combination of these base values:

- 0 write no data
- 1 write the nonlinear analysis results to the .f06 file.
- 2 write the optimization data controlled by P1 and P2 to the .f06 file.
- 4 save the nonlinear analysis results to the .xdb (or .op2) file.
- 8 save the linear response optimization results to the .xdb (or .op2) file.

For example, by default, results from the nonlinear analysis, the optimization data will be written to the .f06 file and result data will be written to .xdb or .op2.

ESLRCF,user_rc_file

Character*8, must be lower case. Default = blank

ESLRCF, user_rc_file allows a user to define a custom RC file for the internally spawned jobs.

Example:PARAM,ESLRCF,*myrc*The contents of the *myrc* file are:

```
mem=200m
exe=-local_path/MSCNASTRAN
del=-local_path/SSS
```

The example shows the user-defined RC file, *myrc*, specifies its own memory allocation and its local Nastran executable and local DMAP database.

ESLTOPCV

Default = 80

ESLTOPCV specifies an alternate convergence tolerance for an ESLNRO's topology optimization job. An ESLNRO's topology optimization job will be terminated when the percentage of design variables whose changes are less than a given value, CONVDV specified on the DOPTPRM Bulk Data entry is greater than ESLTOPCV.

ESLUNT2

Default = 54

ESLUNT2 is a file unit number used by the nonlinear analysis and the linear response optimization runs in an ESLNRO's topology optimization job for the access of design variables and designed properties. It only needs to be changed if there is a conflict with the default.

ESLUNT1

Default = 53

ESLUNT1 is a file unit number used by the linear response optimization run in an ESLNRO's topology optimization job for the storage of design variables and designed properties. It only needs to be changed if there is a conflict with the default.

EST

Replaced by the ELSUM Case Control command.

EULBND

Default = ELEMENT, SOL 700 only

Defines boundary treatment for Euler boundaries.

Format:

PARAM,EULBND,option

EXTRAPOL	The pressure that a wall or coupling surface exerts on the adjacent Euler element is obtained from extrapolating the element pressure toward this boundary. (Character)
ELEMENT	The pressure that a wall or coupling surface exerts on the adjacent Euler element equals the pressure inside this element. (Character)

Remarks:

1. The finite volume representation in general assumes that element values are constant within each element. While this assumption is adequate for the large majority of applications, fluid models involving hydrostatic pressure gradients require that the pressure gradient be also recognized to exist within the element. When element-internal hydrostatic gradients are not accounted for, the calculation will be less accurate and will suffer from numerical symptoms like pair forming of element pressures. By activating the option EXTRAPOL hydrostatic gradients inside the element are taken into account. For meshes without bias, option EXTRAPOL only modifies the numerical schemes along the boundary.
2. When coupling surfaces are used DYPARAM,FASTCOUP has to be activated as well.

EULSTRES

Default = VOLUME, SOL 700 only

Defines the update logic for stresses when material is transported in Euler elements.

Format:

PARAM,EULSTRES,option

VOLUME	The pressure that a wall or coupling surface exerts on the adjacent Euler element is obtained from extrapolating the element pressure toward this boundary. (Character)
MASS	The pressure that a wall or coupling surface exerts on the adjacent Euler element equals the pressure inside this element.

Remarks:

1. Only used for the MMSTREN solver.
2. Stresses are a material property and when material flows in or out an element the stress state in the element is changed. This is analogous to temperature and energy. Not the temperature is transported, but energy. After transporting energy the temperature is re-computed by dividing the energy by element mass and specific heat. In case of stress the “energy” is given by mass times stress. After transporting this “energy” the new stress follows by dividing it by mass. As shown in Chapter 6 of the Theory Manual this gives a correct updating procedure for stresses. There it also proven that stress times mass is conserved during transport.
3. In most simulations variations in density are small and the multiplication by mass can be replaced by a multiplication by volume. This method is activated by option VOLUME which is the default option. Using the MASS option may have some influence on simulations with large density variations. The option MASS will give the most accurate results.

4. The transport logic of the effective plastic strain is identical to that of stresses. When using option MASS the plastic strain is computed more accurately when material is compressed.
5. The (single material) Euler with strength solver makes use of the multiplication by mass. The multiplication by volume is not implemented for this solver.

EUSUBCYC

Default = 1, SOL 700 only

Controls the maximum growth of the subcycling interval in Euler computations.

Example:

PARAM,EUSUBCYC,2

Remarks:

1. The subcycling algorithm automatically estimates the number of subcycles to be used. This is updated throughout the calculation. This parameter controls how much the number of subcycles can grow. For example, EUSUBCYC is set to 1, and the current number of time steps between updates of the Euler variables. If the solver estimates that the subcycling interval should be 7, the subcycling interval is increased by 1 until a value of 7 is reached.
2. There is no control on the amount by which the subcycling interval can decrease.

EXCLUDE

Default = 0.

PARAM, EXCLUDE, Integer. Add PARAM, EXCLUDE, n in the buckling subcase(s) to activate linear buckling with local exclusion of differential stiffness. If n is positive, it selects the SET1 entry ID in the bulk data that defines the list of GRID points for which differential stiffness will be ignored. If n is negative, the absolute value selects the SET1 entry ID in the bulk data that defines the list of GRID points for which differential stiffness will be retained.

To select the GRID points care must be exercised by the user which follow natural mesh lines as much as possible, otherwise an element may be connected to some GRID points that have differential stiffness and some GRID points that do not. Sometimes this is impossible to ensure, and so the transition zone between retained-excluded differential stiffness should not be in, or close to, a region of interest. Selecting stiff (unlikely to buckle) zones for the transition zones is also advised.

Multiple buckling subcases with different exclude values are allowed. For example:

```
DISPLACEMENT (PLOT, SORT1, REAL) =ALL
SUBCASE 1
    SUBTITLE=Static Load
    SPC = 2
    LOAD = 2
    analysis=statics
subcase 2
    SUBTITLE = Buckling Subcase
    SPC=2
    statsub=1
    analysis=buck
```

```
METHOD=105
param,exclude,-1
subcase 3
SUBTITLE = Buckling Subcase
SPC=2
analysis=buck
statsub=1
METHOD=105
param,exclude,-2
BEGIN BULK
```

EXTDR

EXTDR Default = NO. See EXTOUT.

EXTDROUT

EXTDROUT Default = NO.

EXTDRUNT

EXTDRUNT Default = 31. See EXTOUT.

EXTOUT

When inputting the matrices for a reduced external superelement (SEBULK, CSUPER), there are four options that can be selected using the parameter EXTOUT. EXTOUT must be placed in the Case Control Section above any subcase or in the main Bulk Data Section. The options for Step 1 (see the table at the end of this discussion) are as follows:

If EXTOUT is set to MATRIXDB, the reduced structural matrices and loading are stored on the database.

If EXTOUT is set to DMIGDB, the reduced structural matrices and loading are stored on the database in a format which allows automatic connection to the analysis model if the identification numbers of the reduction grid points and scalar points are the same as the grid points and scalar points used in the analysis model.

If EXTOUT is set to DMIGOP2, the reduced structural matrices and loading are written in OUTPUT2 format to a tape unit specified by the parameter EXTUNIT (Default is 31). The storage format is the same as the DMIGDB option and allows automatic connection to the analysis model if the identification numbers of the reduction grid points and scalar points are the same as the grid points and scalar points used in the analysis model. The output unit can be assigned to a specific file by using an ASSIGN command in the File Management Section.

If EXTOUT is set to DMIGPCH, the reduced structural matrices and loading are output on the punch file (.pch) in DMIG format. Also, the model cannot have rotors (ROTOR, ROTORG, ROTORSE, and ROTORAX Bulk Data entries).

The procedure for accessing the external superelement information depends on the option used to output the external superelement in Step 1. The methods are as follows:

1. If EXTOUT was MATRIXDB or DMIGDB in Step 1, use the following commands in the File Management Section:

ASSIGN SEXXX='step1.MASTER'
DBLOCATE DATABLK=(EXTDB, EXTROTDB)CONVERT(SEID=xxx),
LOGICAL=SEXXX

where:

step1.MASTER is the database from the Step 1.

xxx is the superelement identification number given to the partitioned Bulk Data Section for the external superelement.

2. If EXTOUT was DMIGOP2 in Step 1, then use the following commands in the File Management Section:

ASSIGN INPUTT2='step1_output2_file',UNIT=extunit

where:

step1_output2_file is the OUTPUT2 file from Step 1.

extunit is the unit number specified by the parameter EXTUNIT (Default=30).

3. If EXTOUT was DMIGPCH in Step 1, then include the punch file from step one in the partitioned Bulk Data Section. In addition, add the following Case Control commands in the subcase for the external superelement:

K2GG=KAAX

P2G=PAX

The SEBULK entry defining the superelement as an external superelement and the EXTRN entry in the partitioned Bulk Data Section should not be specified.

If data recovery is desired for the external component in SOLs 101, 103, and 107 through 112, there are three methods to transmit the displacements of the reduced model to the external full model. The method is selected by the parameter EXTDROUT in the partitioned Bulk Data Section. The options are as follows:

1. EXTDROUT set to MATRIXDB. The displacements of the reduced component model are stored directly on the database. The sequencing of the displacement degrees-of-freedom corresponds to the sequencing in the reduced model.
2. EXTDROUT set to DMIGDB. The displacements of the reduced model are stored on the database in a format which allows automatic connection to the reduced component model if the reduction grid points and scalar points are the same grid points and scalar points used in the analysis model. This option can only be used if EXTOUT was set to DMIGDB or DMIGOP2.
3. EXTDROUT set to DMIGOP2. The same as EXTDROUT set to DMIGDB except that the displacements of the reduced model are written in OUTPUT2 format to a tape unit specified by parameter EXTDRUNT (Default=31). The output unit can be assigned to a specific file by using an ASSIGN command in the File Management Section. This option can only be used if EXTOUT was set to DMIGDB or DMIGOP2.

Data recovery for the external component is limited to SOLs 101 and 103 and 107 through 112 and 400. Data recovery is accomplished using a restart procedure from the data base created in step one and setting parameter EXTD to YES. The method on inputting the reduced displacements into the component model depends on the method used to output the external component in Step 2. The input methods are as follows:

1. If EXTDROUT was MATRIXDB or DMIGDB in Step 2, then add the following commands in the File Management Section:

```
ASSIGN SEXX='step1.MASTER'  
RESTART LOGICAL=SEXX  
ASSIGN SEYYY='step2.MASTER'  
DBLOCATE DATABLK=(EXTDB) WHERE(SEID=yyy),  
LOGICAL=SEYYY
```

where:

step1.MASTER is the database from the Step 1.

step2.MASTER is the database from the Step 2.

yyy is the superelement identification number given to the partitioned Bulk Data Section for the external superelement in Step 2.

2. If EXTDROUT was DMIGOP2 in Step 2, then add the following commands in the File Management Section:

```
ASSIGN INPUTT2='step2_output2_file',UNIT=extdrunt
```

where:

step2_output2_file is the OUTPUT2 file from Step 2.

extdrunt is the unit number specified by the parameter EXTDRUNT (Default=31).

For SOL 101, the Case Control structure must match the system model subcase structure in the numbers of loading conditions. The loading used in step one to generate the loads transmitted to the analysis model must also be specified in this step. If the analysis model had more loading conditions than the component model, then the loadings defined in Step 1 must be specified first.

For SOL 103 and 107 through 112, the Case Control structure must match the analysis model subcase structure in the number of eigenvalue extractions, FREQ/DLOAD or TSTEP/DLOAD subcases.

Step 1 - Create External SE	Step 2 - Perform Analysis	Step 3 - Data Recovery for External SE
PARAM,EXTOUT, MATRIXDB	a. ASSIGN SEXX='step1.MASTER' DBLOCATE DATABLK=(EXTDB, EXTROTDB), CONVERT (SEID=xxx), LOGICAL=SEXX b. PARAM,EXTDROUT,MATRIXDB	a. ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN SEYYY='step2.MASTER' DBLOCATE DATABLK=(EXTDB), WHERE(SEID=YYY), LOGICAL=SEYYY b. PARAM,EXTDR,YES
PARAM,EXTOUT, DMIGDB	a. ASSIGN SEXX='step1.MASTER' DBLOCATE DATABLK=(EXTDB, XTROTDB), CONVERT (SEID=xxx), LOGICAL=SEXX b. PARAM,EXTDROUT,MATRIXDB or PARAM,EXTDROUT,DMIGDB or PARAM,EXTDROUT,DMIGOP2	a. ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN SEYYY='step2.MASTER' DBLOCATE DATABLK=(EXTDB) WHERE(SEID=YYY) LOGICAL=SEYYY ASSIGN INPUTT2= 'step2_output_file', UNIT=Extdrnt b. PARAM,EXTDR,YES
PARAM,EXTOUT, DMIGOP2	a. ASSIGN INPUTT2='step1_output2_file', Unit = extunit b. PARAM,EXTDROUT,MATRIXDB or PARAM,EXTDROUT,DMIGDB or PARAM,EXTDROUT,DMIGOP2	a. ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN SEYYY='step2.MASTER' DBLOCATE DATABLK=(EXTDB), WHERE(SEID=YYY), LOGICAL=SEYYY ASSIGN INPUTT2= 'step2_output_file', UNIT=Exdrnt b. PARAM,EXTDR,YES
PARAM,EXTOUT, DMIGPCH	Include the .PCH file in partitioned Bulk Data Section and for external SE subcase a. K2GG=KAAX PG=PAX	

EXTRCV

Default = 0

EXTRCV > 0 indicates that data recovery is to be performed on an external superelement. In this type of run, the database for the external superelement must be attached as the primary database ([Database Concepts](#), [453](#) in the *MSC Nastran Reference Guide*), and the database that contains the solution vectors, superelement map, and external/internal grid point equivalence table for its downstream superelement must be attached via the DBLOCATE statements. The value of EXTRCV must also be specified in the CONVERT clause of the DBLOCATE statement for the EMAP data block.

The following example shows the DBLOCATE statements for external superelement data recovery in SOL 101.

EXTUNIT

Default = 31. See EXTOUT.

FACTOR

Default = 10000

See OLDSEQ.

FASTFR

Default = AUTO

In MSC Nastran 2004, the FASTFR method was introduced for modal frequency response analysis. It can be selected via the Bulk Data entry, PARAM,FASTFR,YES and shows significant performance improvement for certain models in the mid-frequency range. PARAM,FASTFR,NO deselects the FASTFR method causing the program to use the standard method for modal frequency analysis.

By default, the program will decide automatically which solution method will be most efficient for the frequency response part in a SOL 111 analysis. Based on the size of the modal space and some other heuristic criteria, either the FASTFR solution method will be run, or the FRRD1 module with or without the iterative solver will be used.

Limitations for the FASTFR method

1. The FASTFR method works only for SOL 111 and for SOL 200 with ANALYSIS=MFREQ.
2. SESDAMP and FASTFR are not allowed in the same run.
3. To force the iterative solver in FRRD1, FASTFR=yes is required.

FBATOLR

Default = 1.0E-05

This is the tolerance that is applied to grid point coordinates in order to determine connections between potential connection grid points of various FRF components in the FBA process.

The default tolerance should be satisfactory for most situations. A looser tolerance may be needed in certain situations. An example is the case where the potential connection points of an FRF component are associated with the shell elements of RSSCON solid-to-shell element connectors. In this case, a looser tolerance may need to be specified in order to achieve proper connections between FRF components in the FBA process.

FBLEND

Default = 0.6667, SOL 700 only

Eulerian elements with uncovered fractions smaller than FBLEND are blended with adjacent elements to form a clump so that they do not control the time step.

Format:

PARAM,FBLEND,VALUE

Example:

PARAM,FBLEND,0.5

VALUE The uncovered fraction below which blending occurs. ($0.0 \leq \text{Real} < 1.0$)

Remarks:

1. The default value is satisfactory for virtually all calculations.
2. Elements are blended only if they would have controlled the time step otherwise.
3. Elements with uncovered fractions greater than FBLEND are not blended and are allowed to control the time step.
4. Large values of FBLEND produce a larger time step but many blends. Small values produce a smaller time step and fewer blends.
5. In a calculation with a coupling surface, STEPFCT is smaller or equal FBLEND to avoid instabilities (see PARAM,STEPFCT).

FDRLDS

Default = YES

PARAM, FDRLDS, YES turns on the logic to generate additional residual vectors for SOL 111 based on the differences of structural and structural damping matrices. For free-free models, a SUPPORT entry may improve the free-free modes and yield additional augmented eigenvalues (PARAM, BAILOUT, -1 may occasionally be needed even for a good static SUPPORT set). PARAM, FDRLDS, NO can be used to turn off the generation of frequency dependent residual vectors.

FIXEDB

Default = 0

FIXEDB is used to reduce the cost of superelement checkout.

-2	(SOL 101 only) is used on the initial runs when the user suspects that the superelement may contain errors and that only operations necessary for fixed-boundary solutions need be performed. In particular, the generation of the $[G_{oa}]$ matrix is branched over in the SEKR operation and $[P_a]$ is not generated in the SELR operation. These operations typically result in 50% of the reduction cost and are not needed in the fixed-boundary data recovery operations described in the next paragraph. After this operation has been completed, the keyword SELANG will appear in the database dictionary, indicating that the $[P_a]$ stored there is incomplete, and should not be summed into the downstream superelement, because System Fatal Message 4252 will be issued.
≤ -1	(SOLs 101 and 103 only) allows uncoupled solutions for any superelement with conventional output requests. This output may be obtained as soon as the superelement is successfully generated and reduced and does not require that the entire model be assembled. In superelement statics, the solution is the component due to the $\{u_o^o\}$ vector, i.e., a fixed-boundary solution. In superelement modes, the solution is the uncoupled eigenvectors of the component. If PARAM,FIXEDB,-1 is specified in the Bulk Data or in the residual structure subcase, the modes of the residual structure will not be computed. For a printout or plotting of the component mode eigenvectors it is recommended that PARAM,FIXEDB,-1 be specified in the Bulk Data Section or above the subcase level in Case Control. If the modes of the residual structure are desired, then PARAM,FIXEDB,0 should be specified in the residual structure subcase. Exterior degrees-of-freedom listed on SECSETi and SESUP entries are free, and those on SEBSETi degrees-of-freedom are fixed. Data recovery for the residual structure should not be requested for this option.
+1	(SOL 101 only) is used after the superelement has been proven valid. In the SEKR and SELR operations, it provides a branch over all operations already completed in the SEKR and SELR phases and completes the generation of the $[G_{oa}]$ matrix and the boundary stiffness and load matrices. It is also a method to recover the factor of the $[K_{oo}]$ matrix if the run aborted while computing $[G_{oa}]$.

FKSYMFACT

Default = 0.024

Follower force stiffness is typically unsymmetric, but in some cases making it symmetric may improve convergence in nonlinear problems, whereas in other cases it may hinder it.

FKSYMFACT controls the symmetrization of the follower force stiffness in SOL 106 and SOL 400. If FKSYMFACT = 1.0 is specified, the follower force stiffness K_f is symmetrized as:

$$K_{fs} = \frac{1}{2}(K_f + K_f^T)$$

and the symmetric part K_{fs} is used for efficiency. If FKSYMFAC= 0. is specified, the original follower force stiffness K_f is used. If a value of $0. < \text{FKSYMFAC} < 1.$ is specified, the non-symmetric part of the follower force stiffness is calculated as:

$$K_{fn} = K_f - K_{fs}$$

and the ratio of unsymmetry:

$$r = \frac{\|K_{fn}\|}{\|K\|}$$

is compared with the user specified value of FKSYMFAC. The norm $\| . \|$ is the absolute maximum number of the matrix.

If $r < \text{FKSYMFAC}$, the symmetric stiffness K_{fs} is used.

If $r > \text{FKSYMFAC}$, the original unsymmetric stiffness K_f is used.

The default value for FKSYMFAC was determined by a parametric study and for most cases this will make the follower force stiffness symmetric, which will give sufficiently accurate answers. The asymmetry ratio(r) increases as geometric nonlinearity intensifies so when it becomes greater than FKSYMFAC, the unsymmetric follower force stiffness is used.

It is considered important to retain the full unsymmetric follower force stiffness, set FKSYMFAC to zero.

The parameter FKSYMFAC is applicable to SOL 106 and SOL 400 only, all other solution sequences symmetrize the follower force stiffness. See parameter FOLLOWK for a list of solution sequences which calculate the follower force stiffness.

FLEXINCR

Default = NO

In SOL 144, a value of YES will cause the TRIM subcases to be ignored. Instead, the TRIM Bulk Data will be used to obtain the set of Mach, Dynamic pressure and symmetry values for Unit Solutions (Flexible Increments). These data can be archived in the aeroelastic database for subsequent reuse. (Flexible Increments are always computed. This param merely avoids the TRIM subcase if these increments are all that is required.)

FLUIDMP

Replaced by options on the Case Control command [FLSPOUT \(Case\), 376](#).

FLUIDNE

Default = 500

The Householder method of eigenvalue extraction is more reliable and is automatically selected for the fluid's system modes if the acoustic cavity is defined in a superelement and there exists fluid boundary points. The switch to Householder occurs if the number of estimated fluid modes is less than or equal to the value specified by PARAMeter FLUIDNE.

FLUIDSE

Default = 0

PARAM,FLUIDSE,seidf specifies a special superelement reserved for fluid elements. Frequency dependent fluid elements must still be in the residual. The newer partitioned superelements are not supported.

FMULTI

Default = .10, SOL 700 only

Defines the dimension of the multimaterial element array.

Format:

PARAM,FMULTI,VALUE

Example:

PARAM,FMULTI,.25

VALUE The relative amount of multimaterial elements. (0.0 < Real < 1.0)

Remark:

The multimaterial Eulerian elements use an overflow array in which to store material data. This array can hold FMULTI times the total number of Eulerian elements. In a problem where more than 10% of the elements have more than one material, the default value of FMULTI must be increased.

FOLLOWK

Default = YES

In SOLs 101,103, 105, 106, 107, 108, 109, 110, 111, 112, 115, 116, and SOL 400, FOLLOWK=YES (Default) requests the inclusion of follower force stiffness in the differential stiffness. FOLLOWK=NO requests that the follower force stiffness not be included. For FOLLOWK=YES in SOLs 101,103, 105, 107, 108, 109, 110, 111, 112, 115, and 116, a separate static subcase is required and the STATSUB command is also required in the eigenvalue subcase. In nonlinear analysis (SOL 106 and 400), the follower force is included if PARAM,LGDISP,1 is specified. FOLLOWK is ignored in SOL 106 and SOL 400 if LGDISP is not specified.

FRQDEPO

Default = NO

By default, frequency-dependent elements cannot be connected to o-set degrees-of-freedom. PARAM,FRQDEPO,YES allows frequency-dependent elements to be connected to o-set degrees-of-freedom. However, results may not be reliable.

FULLSEDR

Default = NO

In a run with superelements, PARAM,FULLSEDR,YES will merge results (DISPL, STRESS, etc.) from all of superelements into a single result as if the run contained no superelements. This is not supported for BEGIN BULK superelements (parts) unless the element and grid identification numbers are unique across all part superelements and the residual structure.

FZERO

Default = 1.0

See AUTOSPRT.

G, GFL

Default = 0.0

G and GFL specify the uniform structural and fluid-damping coefficient in the formulation of dynamics problems. In coupled fluid-structure analysis, G is applied to the structural portion of the model and GFL to the fluid portion of the model. To obtain the value for the parameter G or GFL, multiply the critical damping ratio, C/C_o , by 2.0. PARAM,G and GFL are not recommended for use in hydroelastic or heat-transfer problems. If PARAM,G (or GFL) is used in transient analysis, PARAM,W3 (or W3FL) must be greater than zero or PARAM,G (or GFL) will be ignored. See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide*.

In frequency response and complex eigenvalue analyses, the use of G or GFL will cause the stiffness matrix corresponding to the structural or fluid portion of the model to become complex. The user should be aware that this will, in general, require more memory and resources for the calculations.

GEOMU

Default = 40

See POST=0.

GPECT

Default = -1

GPECT controls the printout of all elements connected to each grid point. GPECT=+1 requests the printout. In superelement analysis, the list is printed if PARAM,CHECKOUT,YES is specified or the SEMG or SEALL Case Control command selects the superelement. GPECT=-1 suppresses the printout.

GRADMESH

Default = OFF, SOL 700 only

Glues fine meshes to coarse meshes. See the section on Graded meshes in the user manual for further information.

Format:

PARAM,GRADMESH,OPTION

Example:

```
PARAM,GRADMESH,MINVOL
```

OPTION	OFF	Graded mesh gluing is not used (Character)
	MINVOL	If an element of one mesh is covered by an element of another mesh the element with the largest volume will be inactivated. It will also be removed from the output request for Eulerian archives.
	ELNUM	If an element of one mesh is covered by an element of another mesh the element with the smallest element number will be inactivated. It will also be removed from the output request for Eulerian archives.

Remarks:

1. This parameter can be used to build block-structured meshes.
2. All Euler elements have to be either orthogonal or axial symmetric.
3. To get meaningful physical results, the change in mesh size going from one element to the next should not be larger than 1.4 or smaller than 0.7.

GRAVSET

Default = 0

A PARAM, GRAVSET, n where n is a Bulk Data SET1 entry, allows the user to apply gravity loading (GRAV) to just a portion of the structure. GRAVSET like GRAV loading is global and is computed once at the beginning of the analysis. The Bulk Data SET1 n defines the GRIDS to which the gravity loading is to be applied. GRAVSET can appear in either the Bulk Data or ABOVE or IN the 1st Subcase. A GRAVSET appearing in any other Subcase will be ignored and may cause incorrect results. A GRAVSET appearing in or above the first subcase takes precedence over a GRAVSET appearing in Bulk Data. See also the companion RFORSET entry. If there is no GRAV loading called out, then PARAM, GRAVSET, n needs to be removed from the run

GRDPNT

Default = -1

GRDPNT > -1 will cause the grid point weight generator to be executed. The default value (GRDPNT = -1) suppresses the computation and output of this data. GRDPNT specifies the identification number of the grid point to be used as a reference point. If GRDPNT = 0 or is not a defined grid point, the reference point is taken as the origin of the basic coordinate system. All fluid-related masses and masses on scalar points are ignored. The following weight and balance information is automatically printed following the execution of the grid point weight generator.

- Reference point.
- Rigid body mass matrix [MO] relative to the reference point in the basic coordinate system.
- Transformation matrix [S] from the basic coordinate system to principal mass axes.
- Principal masses (mass) and associated centers of gravity (X-C.G., Y-C.G., Z-C.G.).

- Inertia matrix I(S) about the center of gravity relative to the principal mass axes. Note: Change the signs of the off-diagonal terms to produce the “inertia tensor.”
- Principal inertias I(Q) about the center of gravity.
- Transformation matrix [Q] between S-axes and Q-axes. The columns of [Q] are the unit direction vectors for the corresponding principal inertias.

In superelement static or geometric nonlinear analysis, GRDPNT > -1 also specifies the grid point to be used in computing resultants, in the basic coordinate system, of external loads and single point constraint forces applied to each superelement. If GRDPNT is not a grid point (including the default value of -1), then the resultants are computed about the origin of the basic coordinate system. In superelement analysis, weights and resultants are computed for each superelement without the effects of its upstream superelements.

For axisymmetric elements, the GRDPNT output may be misleading as the algorithm is designed for 3D elements. For the TRIAX6 element, the mass is for the entire model and the center of gravity and inertias are for the cross section in the x-z plane. For the hyperelastic TRIAX and QUADX elements, the mass is for one radian and the center of gravity and inertias are for the cross section in the x-y plane. For the harmonic TRIAX and QUADX elements, the mass is for the entire model and the center of gravity and inertias are for the cross section in the element coordinate system x-y plane.

GUSTAERO

Default = 1

If gust loads are to be computed, for example on restart, set GUSTAERO to -1. The default is recommended if no gust loads are to be computed after the flutter analysis.

GYROAVG

Default = 0

Used to specify one of two formulations for frequency response analysis using the rotor dynamic capability. The default is to determine any frequency-dependent terms for each frequency. This option activates the frequency-dependent looping option. Setting the value < 0 uses an ‘average’ frequency formulation. This option avoids using the frequency-dependent looping and results in a shorter execution time. For this option, PARAM,WR3 and PARAM,WR4 must be specified to include rotor damping.

HEATCMD

Character*16, Default=nastran

Name of a command to run SOL 600 thermal contact runs. Nastran first sets up an Marc run to determine the thermal contact conditions which are output in a file named jid.nthcnt. Next, Nastran converts these to standard Nastran thermal elements, and finally spawns a second Nastran job from the primary Nastran job. The command to run the second Nastran job is provided using this parameter. For example, if nast2005t1 is desired, enter CMD=nast2005t1. If the command “Nastran” is desired, either leave the parameter out or enter “nastran”. The MSC Nastran run to be spawned will have the form:

CMD jid.nast.dat rcf=RCF

Where file RCF depends on PARAM,MARHEATM.

Remarks:

1. See PARAM,MRSPAWN2 for structural analysis.
2. CMD will be converted to lower case regardless of the case entered.

HEATSTAT

Default = NO

In SOL 101, if PARAM,HEATSTAT,YES is entered, then temperatures are computed in a linear steady state heat transfer and then applied as thermal loads in a subsequent thermal stress analysis. Two subcases are required. The first defines the temperature loads, boundary conditions, and output requests for the heat transfer analysis and the second subcase defines the thermal loads, boundary conditions, and output requests for the thermal stress analysis. Thermal loads in the second subcase are requested through the command

TEMP(LOAD) = Heat Transfer Subcase ID

If this default is not acceptable, then in heat transfer subcase add the Case Control word TSTRU=SID and in structures subcase here

TEMP(LOAD) = SID

See the Case Control command, [TSTRU \(Case\), 632](#). PARAM,NESET is no longer used. HEATSTAT not supported for p-elements.

HFREQ, HFREQFL

Default = 1.+30

The parameters LFREQ, HFREQ, LFREQFL, and HFREQFL specify the frequency range in cycles per unit time of the modes to be used in the modal formulations. (LFREQ and LFREQFL are the lower limits and HFREQ and HFREQFL are the upper limits.) In coupled fluid-structure analysis, HFREQ and LFREQ are applied to the structural portion of the model and HFREQFL and LFREQFL are applied to fluid portion of the model. The default for HFREQ and HFREQFL will usually include all vectors computed. Related parameters are LMODES and LMODESFL.

Note:

If the MODESELECT Case Control command is used, it takes precedence over the parameters LMODES, LFREQ and HFREQ (or LMODESFL, LFREQFL and HFREQFL if MODESELECT refers to fluid modes). For the hierarchy of usage when the MODESELECT Case Control command is used in conjunction with these parameters, refer to the Remarks in the description of the MODESELECT Case Control command. See also the FLSFSEL Case Control command for an alternative selection.

HTOCITS

Default = 20 (SOL 106 only)

HTOCITS sets the maximum allowable iterations in a hot-to-cold analysis. (See the ANALYSIS=HOT2COLD Case Control command).

HTOCPRT

Default = NO (SOL 106 only)

PARAM,HTOCPRT,YES requests the printout of the final cold shape's grid locations in a hot-to-cold analysis. (See the ANALYSIS=HOT2COLD Case Control command).

HTOCTOL

Default = 1.E-2 (SOL 106 only)

HTOCTOL is used to determine convergence of cold shape in hot-to-cold analysis. (See the ANALYSIS=HOT2COLD Case Control command). The parameter is used to compare the geometries as the model deforms from its "hot" to "cold" shape.

HTSYM

Default = 0

This parameter controls the decomposition method for SOL 400 thermal analysis.

- = 0 Use symmetric decomposition solver
- = 1 Use unsymmetric decomposition solver

ICOPT

Default = 1, SOL 400 Only

Parameter ICOPT works together with the NLIC Case Control Command. The user input loads may or may not be in equilibrium with the initial condition. If ICOPT=0, MSC Nastran will compute the initial acceleration based on user's inputs. Otherwise, it will be assumed that the initial acceleration is null. In other words, when ICOPT=1 (the default), it is assumed the whole structure is in equilibrium automatically. Theoretically, ICOPT=0 gives better solution. However, due to that the matrix is highly singular, a large amount CPU time may be required and the accuracy of the result may be in doubt for the solution with ICOPT=0.

IFP

Default = value of PARAM,DBALL.

See DBALL.

INREL

Default = 0

INREL controls the calculation of inertia relief or enforced acceleration in linear static analysis and buckling. INREL = -1 or -2 requests that inertia relief or enforced acceleration be performed.

Enforced accelerations, if desired, are input on the DMIG,UACCEL Bulk Data entry. (See Section 7.2 of the *MSC Nastran Reference Guide* for the theoretical basis.)

Inertia Relief is not currently supported with external superelements and in the contact analysis.

- 1 SUPORT or SUPORT1 entries are required on one or more grid points in the Bulk Data Section which restrain rigid body motion. The total number of degrees-of-freedom specified on SUPORT and SUPORT1 entries must be less than or equal to six.
In SOL 105, SUPORT1, not SUPORT, Bulk Data entries must be used to define the supported degrees-of-freedom and the SUPORT1 Case Control command may only be specified in a separate static subcase.
Loads due to unit rigid body accelerations at the point referenced by PARAM,GRDPNT are computed and then appended to the external loads. If PARAM,GRDPNT is specified in superelement analysis, then the point must be interior to the residual structure and exterior to all superelements.
- 2 The value of PARAM,INREL,-2 will provide an inertia relief analysis without the use of SUPORTi entries. To use this capability the structure must contain six and only six rigid body degrees of freedom. SUPORTi entries must not be present when using this option. If the structure has either more or less than six rigid body degrees of freedom, the analysis will either fail or give incorrect results.

INRLM

Replaced by the INRLOD keyword on the RESVEC Case Control command.

IRES

Default = -1

IRES=1 requests that the residual load vectors RULV and RUOV be output in all solution sequences. In superelement analysis, the parameters PRPA and PRPJ may also be used to request output of the partial load vectors $\{P_a\}$ and $\{P_j\}$, respectively. In geometric nonlinear analysis, PARAM,IRES,1 will cause the printing of the residual vector

$$\{\Delta P_f\} = [K_{ff}] \{u_f^{n+1} - u_f\} + \{F_f\} - \{P_f\}$$

ITAPE

Default = -1

ITAPE specifies the output status of the DSCMR matrix in SOLs 101, 103, and 105; and the DSCMCOL table and the DSCM2 matrix in SOL 200. (See the OUTPUT2 and OUTPUT4 module descriptions in the [MSC Nastran DMAP Programmer's Guide](#).)

IUNIT

Default = 11

IUNIT specifies the FORTRAN unit number on which the DSCMR matrix in Design Sensitivity SOLs 101, 103, and 105 and the DSCMCOL table and the DSCM2 matrix in SOL 200 will be written. (See the OUTPUT2 and OUTPUT4 module descriptions in the [MSC Nastran DMAP Programmer's Guide](#).)

JWLDET

Default=NOLINK

With the default multiple denotations with EOSJWL are NOT LINKED: the detonation wave of one explosive cannot ignite another explosive. When the option is LINK, multiple denotations with EOSJWL are LINKED: the detonation wave of one explosive can ignite another explosive. For option NOLINK: TDET is set to -1 for elements that have no JWL material. The "NOLINK" option is only valid with true JWL materials - not valid for the "Static Detonation/Ideal Gas" model. Setting this parameter as "NOLINK" will prevent "sympathetic ignition". Each charge will ignite at the specified "TDET" in its own DETSPH card.

KDAMP, KDAMPFL

Default = 1

If KDAMP or KDAMPFL is set to -1, viscous modal damping is entered into the complex stiffness matrix as structural damping. In coupled fluid-structure analysis, KDAMP is applied to the structural portion of the model and KDAMPFL to the fluid portion of the model. See [Superelement Analysis in the MSC Nastran Reference Guide](#).

KDIAG

Default = -1.0 (SOLs 106, 153, and SOL 400 with non-contact analysis), or 0.0 (SOL 400 with contact analysis)

In SOLs 106 (nonlinear static analysis), 153 (steady state heat transfer), and SOL 400 (nonlinear static and transient analysis), KDIAG may be used to eliminate spurious mechanisms and singularities in the nonlinear stiffness matrix. The absolute value of KDIAG will be added to some or all of the diagonal terms in the nonlinear stiffness matrix as follows:

- < 0.0 Then add the absolute value of KDIAG to the null diagonal terms only (SOLs 106 and 143).
For SOL 400, the absolute value of KDIAG is added to the diagonal term of null columns only.
- = 0.0 Then no action is taken.
- > 0.0 Then add the value of KDIAG to all diagonal terms.

KDMFILT

Default = 0

- 0 Differential stiffness correction will NOT be added to rigid body behavior of pre-stressed models for modal analysis.
- 1 Differential stiffness correction will be added to rigid body behavior of pre-stressed models for modal analysis.

1. Currently, PARAM, KDMFILT, 1 is active in SOL103 with a preload subcase (STATSUB) and a single modal (METHOD) subcase.
2. Currently, PARAM, KDMFILT, 1 is active in SOL400 with a series of preload steps and a single ANALYSIS=MODES step. Also, in SOL400, the correction is not available when enhanced nonlinear elements are used via NLMOPTS or PSHLN1, etc., type entries.

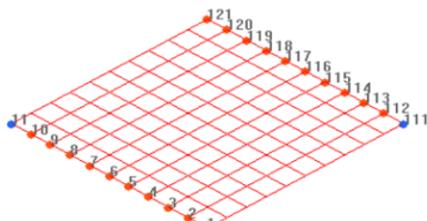
In nonlinear mechanics, in theory a tangent matrix is computed, using a deformation gradient. By polar decomposition, a consistent rotation matrix can be obtained.

In practice, most finite elements such as beam, shell and solid elements, are developed in a heuristic fashion with polynomial type functions representing displacement and volume characteristics. The tangent stiffness is composed of several matrices. A linear K matrix, a K_D for differential stiffness or geometric stiffness, and often a K_F for follower load stiffness.

In general, the K_D matrix which represents a stiffening to the structure due to pre-load of the structure, does not meet rigid body requirements. When a structure is preloaded with a self-equilibrating set of loads and statically supported, for a free-free modal analysis of the pre-loaded structure the static support set is removed. Often because of the deficiency of the K_D matrix to correctly represent rigid body modes, the resulting free-free modes are poor or missing.

PARAM, KDMFLT, 1 turns on a method to orthogonalize and normalize the computed rigid-body modes.

It cannot be over emphasized the importance of the need to have a self-equilibrating set of loads using a static support system.



The above structure represents a plate structure to be tension pre-stressed in the x-direction. This is accomplished by applying FORCE1 entries on grids 111 thru 120 in the positive x-direction defined by grids 1 thru 11 to grids 111 to 121 and a set of FORCE1 entries along grids 1 thru 11 in the negative x-direction defined by grids 111 thru 121 to grids 1 thru 11.

A static support set is applied at grids 1, 11, and 111. This set is removed for the modal analysis.

If, instead of the FORCE1 entries along grids 1 thru 11, the user chose to apply SPC1 entries for these grids, constraining then in the x-direction, a self-equilibrating loading would not be obtained. This is so, because FORCE1 entries are follower loads while SPC1 entries are not. With the SPC1 entries, the structure would not be in true static equilibrium upon removal to the static support set.

If, instead, along grids 1 thru 11, the user chose to apply SPC1 entries for these grids, constraining then in the x-direction and non-follower FORCE entries along the grids 111 thru 121, an almost self-equilibrating loading is achieved and PARAM, KDMFLT, 1 will get six good free-free modes. In this case, however, the Lanczos Method has some stability issues and AHOU should be used.

K4RITZ

Default = 0

Setting PARAM,K4RITZ to a positive value activates logic to generate additional residual vectors for SOL 111 when using ACMS to compute the modal space. Structural damping degrees of freedom (K4) are used to augment the existing set of residual vectors in order to capture damping effects when GE inputs might contribute significantly to overall frequency response. The value of K4RITZ specifically controls the number of iterations performed to compute the basis vectors. Due to extra computation required, this parameter should be used with caution and should rarely be set greater than 1.

K6ROT

Default = 100.

K6ROT specifies the scaling factor of the penalty stiffness to be added to the normal rotation for CQUAD4 and CTRIA3 elements. The contribution of the penalty term to the strain energy functional is

$$\Pi_p = 10^{-6} K6ROT \frac{1}{2} G \int_A (\Theta_z - \Omega_z)^2 t dA \text{ with } \Omega_z = \frac{1}{2} \left(\frac{\partial u_y}{\partial x} - \frac{\partial u_x}{\partial y} \right)$$

where A is the area of the shell element, t is the shell thickness, G is the in plane shear modulus, see the MID1 material identification number on the PSHELL Bulk Data entry. The in plane displacements u_x, u_y and the normal rotation Θ_z are shown in [Figure 6-1](#). The normal rotation has no physical meaning and should be ignored. The penalty stiffness removes the singularity in the normal rotation. A higher value than K6ROT=100. is not recommended because unwanted stiffening effects may occur. If you use only the MID1 field, Nastran automatically sets K6ROT=0.0

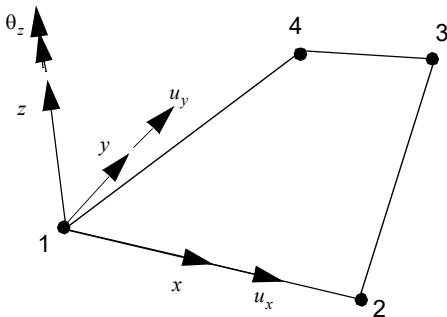


Figure 6-1 In plane displacements u_x , u_y , and normal rotation Θ_z

LANGLE

Default = 1

LANGLE specifies the method for processing large rotations in nonlinear analysis. By default, large rotations are computed with the gimbal angle method in nonlinear analyses SOLs 106, 129, 153, and 159 with geometric nonlinearity (PARAM,LGDISP,1). If PARAM,LANGLE,2 is specified, then they are computed with the Rotation Vector method. The value of LANGLE cannot be changed in a subsequent restart. For SOL 400, users should not use LANGLE. SOL 400 will use the appropriate method depending on type of element or type of analysis.

LDSUM

Default = 0

Dictates what trim information is to be stored on a CSV (comma separated values) file in a SOL 144 (static aeroelasticity) task. The unit the CSV file is stored to is specified by PARAM, XYUNIT, n. LDSUM has the following options:

0	(Default) – Do not create a CSV file for static aeroelasticity
1	Create a CSV file that contains for each static aeroelastic subcase: <ul style="list-style-type: none"> ■ Subcase ID ■ Mach number ■ Dynamic Pressure ■ Trim Values ■ Mass and CG information (mass, xcg,ycg,zcg, IXX,IYY,IZZ,IXY,IXZ and IYZ)
2	Same as 1 plus net structural monitor point (MONPNT1, MONDSP1, MONPNT2, MONPNT3) results.
3	Same as 2 plus the output of RIGID AIR, ELASTIC RESTRAINED, and INERTIAL, RIGID APPLIED and ELASTIC APPLIED components for the structural MONPNT1 results.
4	Same as 3 plus the output of RIGID AIR and ELASTIC RESTRAINED components for aerodynamic MONPNT1 results.

LFREQ, LFREQFL

Default = 0.0

See HFREQ, HFREQFL

If the MODESELECT Case Control command is used, it takes precedence over the parameters LMODES, LFREQ and HFREQ (or LMODESFL, LFREQFL and HFREQFL if MODESELECT refers to fluid modes). For the hierarchy of usage when the MODESELECT Case Control command is used in conjunction with these parameters, refer to the Remarks in the description of the MODESELECT Case Control command. See also the FLSFSEL Case Control command for an alternative selection.

LGDISP

LGDISP is a global parameter. For the case with multiple LGDISP definitions, Nastran will use the first definition for all the subcases and steps. Default = -1

- ≥ 0 The differential stiffness for structural elements is computed for the linear elements and added to the differential stiffness of the nonlinear elements.
- 1 All the nonlinear structural element types that have a large displacement capability in SOLs 106, 129, 153, 159, 600 and SOL 400 (see [Table 3-1](#), under “Geometric Nonlinear” in the *MSC Nastran Reference Guide*) will be assumed to have large displacement effects (updated element coordinates and follower forces). For simultaneous multi-physics coupling analysis defined through the SUBSTEP Case Control option in SOL 400, nonlinear heat transfer element types will be treated the same as LGDISP = -1 (i.e., original geometry for thermal stiffness, element fluxes and external loads).
- 1 No large displacement effects will be considered.
- 2 Follower force for structural elements effects will be ignored but large displacement effects will be considered. For simultaneous multi-physics coupling analysis defined through the SUBSTEP Case Control option in SOL 400, nonlinear heat transfer element types will be treated the same as LGDISP = -1 (i.e., original geometry for thermal stiffness, element fluxes and external loads).
- 11 Nonlinear structural elements will be treated the same as LGDISP = 1 in all supporting solution sequences. For simultaneous multi-physics coupling analysis defined through the SUBSTEP Case Control option in SOL 400, nonlinear heat transfer element types will be based on the current updated geometry (updated element coordinates for the thermal stiffness, element fluxes and external loads).
- 12 Nonlinear structural elements will be treated the same as LGDISP = 2 in all supporting solution sequences. For simultaneous multi-physics coupling analysis defined through the SUBSTEP Case Control option in SOL 400, nonlinear heat transfer element types will be based on updated current geometry (updated element coordinates for the thermal stiffness and element fluxes, original geometry for external loads).

LMFACT

LMFACT and PENFN are the scale factor and penalty function for the Lagrange rigid elements and the contact analysis. For Lagrange rigid elements, please see Case Control command, RIGID. The purpose of LMFACT and PENFN is to make the values of stiffness matrix of the Lagrange rigid elements and/or the

contact components about the same relative magnitude as those of the other elements in the model. Too small a value will produce inaccurate results and too large a value will produce numerical difficulties. The same value is usually assigned to both LMFACt and PENFN. Under special requirement, user may assign different values for LMFACt and PENFN. For example, if PENFN=0.0 and LMFACt ≠ 0.0 , then the solution method for the rigid elements becomes the pure Lagrange multiplier method instead of the augmented Lagrangian method. However, user must exercise caution if different values are assigned to LMFACt and PENFN. MSC Nastran will compute the appropriate default values for LMFACt and PENFN. The default value is 1.0e+5 for all solution sequences except SOL 400 and SOL 101 contact.

For SOL 400 and SOL 101 contact, MSC Nastran will compute the appropriate default values for LMFACt and PENFN. These defaults are currently calculated as 0.05% of the average stiffness of the diagonal terms of the stiffness matrix. This computed default is good in general. But when the material/element stiffnesses (e.g., Young's Modulus, Spring Stiffnesses) used in the model vary in a wide range, users may have to adjust the computed LMFACt and PENFN. Note that the computed LMFACt, PENFN are printed out in a .f06 file. It is generally recommended that for these cases, the LMFACt and PENFN should be reduced by several orders of magnitude to avoid numerical difficulty.

If in a coupled multi-physics analysis distinct values for LMFACt and PENFN are desired for each physics pass, the parameter definitions must be made inside the SUBSTEP Case Control command. If no values are entered each physics pass computes its own defaults.

LMODES, LMODESFL

Default = 0

LMODES and LMODESFL are the number of lowest modes to use in a modal formulation. In coupled fluid-structure analysis, LMODES specifies the lowest modes of the structural portion of the model and LMODESFL the modes of the fluid portion of the model. If LMODES (or LMODESFL) = 0, the retained modes are determined by the parameters LFREQ and HFREQ (or LFREQFL and HFREQFL).

In SOL 103, LMODES may be used to reduce the number of eigenvectors to be processed in data recovery which may significantly reduce the CPU and storage costs.

Note:

If the MODESELECT Case Control command is used, it takes precedence over the parameters LMODES, LFREQ and HFREQ (or LMODESFL, LFREQFL and HFREQFL if MODESELECT refers to fluid modes). For the hierarchy of usage when the MODESELECT Case Control command is used in conjunction with these parameters, refer to the Remarks in the description of the MODESELECT Case Control command. See also the FLSFSEL Case Control command for an alternative selection.

LOADU

Default = -1

See POST=0.

LOOPID

Default = 0

LOOPID defines the desired loop number for initial conditions in a restart of SOLs 106, 129, 153, and 159. By default in SOLs 106 and 153 the restart proceeds from the last loop ID of the subcase defined by SUBCASID or SUBID. In SOLs 106, and 153 PARAM,SUBID or SUBCASID may also be specified.

LSTRN

Replaced by the STRAIN Case Control command.

MACH

Default = 0.0

Mach number. If more than one Mach number was used to compute aerodynamic matrices, the one closest to MACH will be used in dynamic aeroelastic response analysis. The default causes the matrices computed at the lowest MACH number to be used.

MARALPHA

Default = 10, SOL 600 only.

Determines what type of coefficient of thermal expansion (CTE) is entered in the Nastran input file when CTE varies with temperature. For most Nastran solution sequences, such as SOL 106 and SOL 400, if CTE's vary with temperature, secant values need to be entered. This parameter specifies whether secant or actual values are entered at each temperature on curves using the TABLEM1 entry. In addition, it offers methods to convert the secant CTE's instantaneous values required by Marc. Please read MAT1 Remark 10 and MATHE Remark 8 for further details.

- 1 CTE's (vs temperature) are entered in the Nastran input as instantaneous values – no conversion is required (Warning if this option is used, the CTE's should only be used in SOL 600 and will produce incorrect results if used in SOL 106 or SOL 400).
- 0 CTE's (vs temperature) are secant values (as required by SOL 106, 129 and 400). They will be converted to instantaneous values using the formula immediately below for use in the Marc portion of SOL 600.

$$\alpha_i = A_i + \left(\frac{A_i - A_{i-1}}{T_i - T_{i-1}} \right) (T_i - T_{ref})$$

- 1 CTE's (vs temperature) are secant values and are converted to instantaneous values using the formula immediately below for use in the Marc portion of SOL 600.

$$\alpha_i = \frac{A_i(T_i - T_{ref}) - A_{i-1}(T_{i-1} - T_{ref})}{T_i - T_{i-1}}$$

- 2 CTE's (vs temperature) are secant values and are converted to instantaneous values using the average of the values computed by options 0 and 1.
- N Same as option 1 except N intermediate points are placed between each original CTE vs temperature value to increase accuracy. N must be 3 or greater. Testing has shown that for CTE's that change appreciably over the temperature range, N should be 10 or larger. The total number of points must not exceed 8000.

Remarks:

1. α is the actual (instantaneous) CTE
2. T is temperature T_{ref} is specified on MAT1 and other MAT entries.
3. At $T(1)\alpha_1 = A_1$
4. The above equations are applied at mid temperatures. The first temperature is retained. Starting with the second temperature $T_{iN} = 0.5 \cdot (T_i + T_{i-1})$ original. Since the new final temperature will be smaller than the original the user should ensure that the final average temperature is large enough to cover the applied temperature range of the analysis.
5. If the CTE's entered in the Nastran input file are secant values, it is recommended that MARALPHA=N with N set to a value of 20 or larger but less than 8000 divided by the number of points in the largest CTE vs temperature TABLEM1 entry.
6. Only option -1 was available prior to MD Nastran 2010 and MSC Nastran 2008.

MARAUTOC

Default = 0, SOL 600 only.

Determines whether NLAUTO entries for SOL 600,129 will override the default or not.

- 0 Do not override the default. (Default)
- 1 NLAUTO initial time step and final time will override the defaults.

Remark:

The default for the initial time step is to use the DT value from the entry or the first non-zero time value on the TABLED1 entry divided by 100.0 whichever is smaller. The default for the final time is NDT*DT from the entry or the last time point from the TABLED1 entry whichever is smaller.

MARBATCH

Default = 0, SOL 600 only.

Specifies whether Marc will be spawned from Nastran in the "batch" mode or not.

- 0 Marc will be spawned using batch=no. (Default)
- 1 Marc will be spawned using batch=yes.

Note: PARAM,MARBATCH,0 requires PARAM,MARCTEMP,1 (which is the default). This combination of parameters will place the Marc log file in the Nastran log file.

MARBK105

Default = 1, SOL 600 only.

This parameter controls whether linear buckling or nonlinear buckling eigenvalues are calculated for SOL 600,105.

- 1 Nonlinear eigenvalues are found. In other words, all loads are placed after Marc's END OPTION and default values are used for CONTROL and AUTOSTEP. This option simulates what happens with SOL 106 or SOL 400.
- 1 Linear eigenvalues are found. In other words, all loads are placed before Marc's END OPTION, a linear analysis is used to obtain the differential stiffness and eigenvalues are then calculated. This option simulates what happens with SOL 105. (Default)

MARBK106

Default = 1, SOL 600 only.

Controls whether linear buckling or nonlinear buckling eigenvalues are calculated for SOL 600,106.

- 1 Nonlinear eigenvalues are found. In other words, all loads are placed after Marc's END OPTION and default values are used for CONTROL and AUTO STEP. This option stimulates what happens with SOL 106 or SOL 400.
- 1 Linear eigenvalues are found. In other words, all loads are placed before Marc's END OPTION, a linear analysis is used to obtain the differential stiffness and eigenvalues are then calculated. This option simulates what happens with SOL 106.

MARC4401

Default = 0, SOL 600 only.

Determines whether recycling due to body to body contact occurs.

- 0 Recycling due to body to body contact can occur
- 1 Recycling due to body to body contact is prevented - will add feature,4401 to the Marc input

MARC7601

Default = 0, SOL 600 only.

Determines whether large deformation RBE3 element will be used.

- 0 Use small deformation RBE3
- 1 Use large deformation RBE3

Remarks:

1. MARC7601=0 sets FEATURE,7601 in Marc.
2. MARC7601=1 will not set FEATURE,7601.

MARCASUM

Default is -1 for nonlinear analysis and 1 for linear analysis. SOL 600 only.

Marc's assumed strain formulation is used for plane stress, plane strain and solid elements (Marc types 3, 11 and 7). The assumed strain formulation improves the bending behavior of these elements and uses an enriched set of interpolation functions. Assumed strain should be off for analyses with a significant amount of plasticity. In determining the type of analysis (linear or nonlinear) for defaults of this parameter, the SOL 600, ID Executive statement is used. If ID is 106 or 129, the analysis is considered to be nonlinear and the default is -1. If ID is any other value, the analysis is considered to be linear and the default is 1. For nonlinear analyses without plasticity, this parameter should be turned on for models with solid elements.

- 1 Assumed strain is not used.
- 1 Assumed strain is used.

MARCAUTO

Default = leave out parameter, SOL 600 only.

Determines which Marc's increment option is used.

- 1 NLPARM entries will be translated to Marc's AUTO STEP option. If contact is present, the number of steps (NINC) is less than 100, it will be reset to 100. Marc will adaptively reduce the number of steps if possible, however, this option forces the first step to be 1% of the total time. If the first step is too large, experience has shown that convergence problems may result. To start with a different initial time step, see options 999 or -999.
- 1 NLPARM entries will be translated to Marc's AUTO INCREMENT option. If contact is present, the number of steps is automatically set to 100. It has been found that certain difficult contact problems which fail using the AUTO STEP option run successfully using AUTO INCREMENT. This option is not available if the only "loading" is rigid contact or velocity control.
- 2 NLPARM entries will be translated to Marc's AUTO LOAD option with no adjustment in the number of steps. Use of the option is not recommended. This option is not available if the only "loading" is rigid contact or velocity control.

- 999 Marc's AUTO STEP option will be used with no adjustment in the number of steps whether or not contact is present. This option is not available if the only "loading" is rigid contact or velocity control.
- 999 Marc's AUTO INCREMENT option will be used with no adjustment in the number of steps whether or not contact is present. This option is not available if the only "loading" is rigid contact or velocity control.

See PARAM,MARCITER for a similar option. Do not use both MARCAUTO and MARCITER parameters.

MARCAZEL

Default = 0, SOL 600 only.

Allows a combination of axisymmetric and plane stress elements for 2D analyses. This analysis technique is sometimes used for approximate turbine disk/blade analysis.

- 0 The combination, if present in the input data will cause a Severe Warning and Marc will not be spawned.
- 1 Combination is allowed and all CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR elements will be considered to be plane stress and mate with CTRIA6 elements.

MARCBEAM

Default = -1, SOL 600 only.

- 1 All CBEAM elements which reference PBEAML will be mapped to Marc element type 14 for all cross if any CBEAM elements in the model reference MATS1 or MATEP entries. Full plasticity is available for all such cross section shapes with this option. CBEAM cross sections specified using PBEAM (where only A, I, J are available) will be mapped to Marc element 98 and remain elastic even though they might reference MATS1 or MATEP.
- 0 CBEAM will be mapped to Marc element type 14 for all cross sections specified with PBEAML. Full plasticity is available for all such cross section shapes with this option. CBEAM cross sections specified using PBEAM will be mapped to Marc element 98 and remain elastic even though they might reference MATS1 or MATEP.
- 1 All CBEAM elements will be mapped to Marc element type 98 and remain elastic regardless of whether the cross section is specified using PBEAM or PBEAML or whether they reference MATS1 or MATEP.

Remark:

If PBMARB6 or PBMMNUM6 beam properties are used, MACRBEAM must be set to zero which automatically activates the new forms of Marc element 98 that can include plasticity.

MARCBODY

Default=0, SOL 600 only.

Control the logic when the specified value of NBODIES entry on BCPARA bulk data card does not match the actual number of contact bodies in the model.

- 0 Analysis continuing with number of bodies actually in the model
- 1 Analysis continuing with number of bodies specified on NBODIES entry on BCPARA bulk data card. (This may produce exit 13 in Marc)

MARCBUSH

Default = -1.0 if parameter is not entered, no “small” stiffness will replace zero stiffness terms, SOL 600 only.

Determines whether “small” stiffness values will be used instead of zero for the stiffness values in various directions of CBUSHi elements.

- 1.0 No “small” stiffness terms will replace zero stiffness values in any direction.
- 0.0 Stiffness values of $0.01 * K_{max}$ will be added for any direction that is zero Marc input (applies to both translational and rotational directions)
- Value The value entered will be used to calculate stiffness=Value*Kmax to replace any zero stiffness values (applies to both translational and rotational directions)

MARCCBAR

Default = 0, SOL 600 only.

Specifies whether CBAR will be replaced by CBEAM for SOL 600.

- 0 CBAR is not replaced by CBEAM.
- 1 CBAR is replaced by CBEAM (PBAR is replaced by PBEAM, PBARL is replaced by PBEAML).

Note:

Use of this parameter is not usually required but might be beneficial in combination with PARAM,MSPEEDSE,1 to speed up translation of models with a large number of CBAR elements particularly when there are large number of PBAR entries or PBARL entries.

MARCCENT

Default = 0, SOL 600 only.

Controls where the element output is generated.

- | | |
|---|---|
| 0 | Element output from Marc will be generated for each integration point. |
| 1 | Element output from Marc will be generated at the center of each element only. This option saves disk space and computer time, but may not catch the maximum stresses or strains. Because the residual load calculation is not accurate, this should not be used in a nonlinear analysis. |

MARCCON2

Default = Program determines value, SOL 600 only.

- | | |
|-------|---|
| Value | If entered, the integer value entered is the second value on Marc's CONTACT second entry representing the maximum number of entities to be created for any contact surface. |
|-------|---|

MARCCON3

Default = Program determines value, SOL 600 only.

- | | |
|-------|---|
| Value | If entered, the integer value entered is the third value on Marc's CONTACT second entry representing the maximum number of nodes that lie on the periphery of any deformable contact surface. |
|-------|---|

MARCCPY

If MARCCPY is specified, Marc files will be copied to Nastran output files and/or deleted according to the option (0, 1, or 2) shown below.

MARCCPY Option	Copy Marc Output Files to Nastran Output Files	Delete Marc Input & Output Files
0 (Default)	No	No
1	Yes	Yes
2	Yes	No
3	No	Yes

If MARCCPY is 1 or 2, the out and log files will be copied as produced by Marc. If MARCCPY is -1 or -2 the actions as shown above for +1 or +2 will occur, and Marc-type test will be converted to Nastran-type text using an ASCII file named marcfilt.txt which must be located in the same directory where the Nastran input resides or in the same directory where the Nastran executable resides.

The following Marc files are potentially affected by the MARCCPY option:

Marc Output File	Nastran Output Copied to	MARCCPY
name.marc.out	name.f06	1, 2, -1, -2
name.marc.log	name.log	1, 2, -1, -2
name.marc.t16	not copied, will remain if produced	
name.op2, fort.11, or ftn11	not copied, will remain if produced	

MARCDEF

Default = 2, SOL 600 only.

- 0 SOL 600 default options for Marc will be set to values determined to be best for Nastran-type problems (for MARCDEF=0, Marc's SHELL SECT parameter will be set to 11 if the value of MARCDEF is zero).
- 1 Default values will be set to current Marc standard (Mentat) values.
- 2 Default values will be set to "improved" Marc default values agreed on by the Marc and Nastran development groups.

Default values affect the following Marc input data entries and fields:

MARCDEF Value	MARC Implicit Entry Type	Field	Value
0	control	2	10
0	Auto Step	5	0.01*max time
0	Auto Step	8	10
0	Auto Step	10	1
1	control	2	3
1	Auto Step	5	1.0E-5*max time
1	Auto Step	8	5
1	Auto Step	10	0
2	control	2	10
2	Auto Step	5	1.0E-3*max time
2	Auto Step	8	5
2	Auto Step	10	1

Note: For MARCDEF=0, the first three values were found to provide better convergence and the last (auto step 10) allows snap-through solution to converge correctly without having to use arc-length methods. This parameter can be set in the system-wide rc file as well as the user's rc file or the local rc file (same directory as the Nastran input data to provide the selected set of defaults for all runs if so desired. If the parameter is entered in the Nastran input data file, it will override any parameters set in any of the rc files.

MARCDILT

Default = 0, SOL 600 only.

If omitted, SOL 600 determines the value.

- | | |
|---|--|
| 0 | Constant dilatation is not used. |
| 1 | Constant dilatation formulation is used for solids, axisymmetric, and plane strain elements (advance nonlinear element types 7, 10, 11, 19 and 20) if the model includes any of these element types. For elastic-plastic and creep analysis this formulation is usually too stiff when constant dilatation is not used. MARCDILT=1 and MARCASUM=1 should not both be used. |

MARCDIS2

Default = Program determines value, SOL 600 only.

- | | |
|-------|--|
| Value | If entered, this integer value entered here is the second value on Marc's DIST LOADS ("parameter" Section 2 of Marc's Volume C Program Input) entry representing the maximum number of different lists of distributed loads. |
|-------|--|

MARCDIS3

Default = Program determines value, SOL 600 only.

- | | |
|-------|---|
| Value | If entered, the integer value entered here is the third value on Marc's DIST LOADS ("parameter" Section 2 of Marc's Volume C Program Input) entry representing the maximum number of elements in any particular distributed loads list. |
|-------|---|

MARCDIS4

Default = Program determines value, SOL 600 only.

- | | |
|-------|---|
| Value | If entered, the integer value entered here is the fourth value on Marc's DIST LOADS ("parameter" Section 2 of Marc's Volume C Program Input) entry representing the maximum number of nodes with point loads applied. |
|-------|---|

MARCDMIG,N

Default = 0, SOL 600 only.

If matrices or loads are entered using K2GG, M2GG, B2GG, K2PP, M2PP, B2PP, P2G in the Nastran Case Control Section, they will be translated to Marc as follows depending on the value of N:

- N=0 All DMIG's in the Nastran file (and include files) will be placed in the Marc input file whether used or not.
- N>0 All DMIG's in the Nastran file (and include files) will be placed on a new file named dmigxxxx.dmi where xxxx is the value of N. This new file will be "included" in Marc using a Marc include statement. For example, if N=100 the file name will be dmig100.dmi if N=25765 the file is dmig25765.dmi. N must not exceed a value of 999999.

Note:	This parameter is ignored for External Superelements (if the MESUPER Bulk Data entry is present).
--------------	---

MARCDUPE

SOL 600 only.

Controls whether SOL 600 will check for duplicate entries for most every type of bulk data entry. SOL 600 does not allow duplicate entries, but the portion of IFP that runs prior to spawning Marc does not usually check for duplicate entries.

- 1 Duplicate entries will be checked and exact duplicates are found, the job will fatal out.
- 2 In addition to option 1, entries will be checked and if duplicate ID's (field 2 for most entries or field 3 for loads) are found, the job will fatal out.
- 1 Duplicate entries and ID's will not be checked (this is desirable for certain models)

MARCDYND

Integer, Default = 0

Determines whether dynamic loads with "advanced table input" will be used in SOL 600.

- 0 Do not use advanced table input for dynamic loads
- 1 Use advanced table input for dynamic loads

If MARCDYND=1, PARAM,MARCTOTD,1 and PARAM,MARCTOTT,1 must also be set.

MARCEKND

Default = 0, SOL 600 only.

ID Selects the type of strain results to be placed in a Nastran op2 file (if a request for an op2 file is made). ID can take the following values:

 MARCEKND=0, Total strains will be processed

 MARCEKND=1, Plastic strains will be processed

 MARCEKND=2, Elastic strains will be processed

For creep analyses, creep strain is output if a request for strain output is made.

MARCEXIT

Default = 0, SOL 600 only.

0 If one of the COPYR options on the SOL 600 statement is specified, Nastran will process these options and then a DMAP exit will occur.
1 The COPYR options will be processes and Nastran will not exit.

MARCFEAT,N

SOL 600 only.

If entered will add FEATURE,N to the Marc input file in the Parameters Section.

N Feature to be added, for example PARAM,MARCFEAT,5102 will generate a heat transfer thermal contact file jid.marc.nthcnt

Only one PARAM,MARCFEAT may be entered.

MARCFILI

No Default, SOL 600 only.

Name Name a file name limited to 8 characters (16 characters if param* is used) used in conjunction with one of the CONTINUE options on the SOL 600 statement. For example, if CONTINUE=1 on the SOL 600 statement and PARAM,MARCFIL1,DMIG44 is entered, friction stiffness and possibly damping) matrices are created in DMIG format by Marc and placed on file DMIG44. The various CONTINUE options use the following MARCFILI entries:

Continue Option	MARCFILi	SOL Executed	K2GG/K2PP
1.	MARCFIL1	107	K2GG
2.	MARCFIL2	107	K2GG
3.	MARCFIL3	111	K2PP
4.	MARCFIL1	112	K2PP
5.	MARCFIL1	*	*
6.	MARCFIL1	110	K2GG
7.	MARCFIL1	103	K2GG

Remarks:

- For most continue options other than brake squeal models, the Marc portion of SOL 600 will produce files with very long names such as jid.marc.conmpc_0007. These file names are too long for a Nastran parameter field even if PARAM* is used. The solution is to specify a short name such as PARAM,MARCFIL1,ABCD.TXT
Inside ABCD.TXT place an include line for the actual file desired, such as
INCLUDE 'myjob.marc.conmpc_0007'
Where myjob would be replaced by the actual JID of the primary Nastran run without the bdf or dat extension.
- If INITCON=4 on the BCPARA entry is specified contact MPC's for each increment will be out on a file named jid.marc.conmpc_incr (if the increment is 7 it would be jid.marc.conmpc_0007).
- If the DMIGOUT Bulk Data entry is used, DMIG's will be produced in the form jid.marc_cglsti_incr. See the description of DMIGOUT for further details.
- For most cases, the difference between jid.marc.conmpc_last and jid.marc.conmpc.0001 is not large enough to affect the subsequent analysis significantly. To be sure that the last jid.marc.conmpc_xxxx is used specify the following
param,marcfil1,lastt
Nastran will search for all jid.marc.conmpc_* files and choose the one with the largest _xxxx. If no jid.marc.conmpc_* files exist, the job will terminate with an fatal error message.

MARCFRIC

Default = 0.0, SOL 600 only.

When the Case Control command, BCONTACT = ALL is specified, no other 3D contact data is required in the input file, except that the Coulomb coefficient of friction may be entered using the value of this parameter. Do not enter this entry if contact surfaces are specified in the Bulk Data.

MARCGAPD, D

Default is U0 of the PGAP, 2840 entry, SOL 600 only.

- D Depending on the value of PARAM,MARCGAPP, enter the gap closure distance for fixed direction gaps or the minimum distance between end points for the true distance gap. If $d > 0$, the two end points are never closer than a distance $|d|$ apart. If $d < 0$, the two end points are never farther apart than $|d|$.

MARCGAPN, ID

No Default, SOL 600 only.

- ID ID of gap element for which the immediately following PARAM,MARCGAPP and PARAM,MARCGAPD apply. Unlike most other parameters, several sequences of parameters MARCGAPN, MARCGAPP and MARCGAPD may be entered to specify values for all gap elements. If no MARCGFAPN is entered, the values entered for MARCGAPP and MARCGAPD will be used for all gaps in the model.

MARCGAPP

Default = 0, SOL 600 only.

- 0 Nastran gap elements will be translated to Marc fixed gap elements.
- 1 Nastran gap elements will be translated to Marc True Distance gaps.

MARCGAUS

Default if parameter is not entered = 1, SOL 600 only.

- 1 SOL 600 output stresses and strains will be at the Gauss points for solid elements and extrapolated to the corner points for plates/shells elements. Strains are handled the same way as stresses.
- 2 SOL 600 output stresses and strains will be at the center and at the grid points for solid elements (the maximum stress from any Gauss point is determined and compatible stresses for that Gauss point are placed at the center and at each of the grid points). Shell/plate stresses are at the center (top and bottom surfaces). The maximum Gauss point stress at each surface is found and a compatible set of stresses at that Gauss point are placed in the center of the surface. Strains are handled the same way as stresses.
- 3 Solid stresses/and strains are the same as option 1 and shell/plate stresses and strains are the same as option 2.

If this parameter is entered with values of zero or less or values greater than 3, it will be reset to 1.

MARCGLUE

Default = 0 if parameter is not entered, SOL 600 only.

If MARCGLUE is set to 1, all contact surfaces will be glued whether or not IGLUE=1 is specified on the BCTABLE entry or not.

- 0 IGLUE on BCTABLE entries specifies whether or not glued contact is used
- 1 IGLUE on BCTABLE entries will be ignored and glued contact will be used for all contact surfaces.

Remark:

For this parameter to function BCTABLE entries must be entered (do not set BCONTACT=ALL).

MARCGRAV

Integer, Default = 0

Determines whether PARAM,MNASTLDS,777 will automatically be set if multiple GRAV entries are present in any subcase. This is determined in a simplified fashion by comparing the number of GRAV entries to the number of subcases to save computer time.

- 0 Set PARAM,MNASTLDS,777 for multiple GRAV entries.
- 1 Do not set PARAM,MNASTLDS,777 regardless of the number of GRAV and SUBCASE entries.

MARCHOST

No Default, SOL 600 only.

Determines the name of a hostfile to be used with SOL 600 parallel runs. If this parameter is missing, no host file is used and the parallel run will run on one machine. That machine may have several processors and as many processors as specified on the PARAMARC Bulk Data entry will be used. If PARAM,MARCHOST,Name is specified, the hostfile must be generated by the user in a format acceptable to Marc (see the *Marc and Marc Mentat Installation and Operations Guide*). Each line of the hostfile normally lists how many processors are used on each machine. If PARAM*,MARCHOST is entered, the name is limited to 16 characters (all lower case).

MARCIAMN

Default = 1, SOL 600 only.

- 0 Nastran is directed to spawn Marc (as specified by the SOL 600 Executive Control statement or PARAM,MARCRUN), using a full version of Marc. Standard Marc licensing is required.
- 1 A special version of Marc is spawned by Nastran. This version of Marc may have certain features that are not available in the full version. Marc will be spawned from Nastran with the additional command line switch - iam nanl. The licensing for both Nastran and Marc reflect this situation. This option applies only to Marc version 2003 or later. If PARAM,MARCVERS points to a Marc version earlier than 2003, MARCIAMN will be set to zero and a full version of Marc is required.

The parameter may be set in the system-wide rc, the user rc file or as an environmental variable using NASM_IAMN=0 or 1 (similar to the way values on the SOL 600 statement are set).

MARCINTC

Default = 2.

Option to ignore or fatal SOL 600 if any CINTC Bulk Data entries are found. This option will also ignore or issue a warning for GMBNDC entries.

- 0 Fatal job if any CINTC entries are found (also issue warning messages if any GMBNDC entries are found).
- 1 Ignore all CINTC and GMBNDC entries in SOL 600.
- 2 Generate the MPC's for CINTC/GMBNDC and combine them with standard MCP entries (if they exist), then use them in the SOL 600 analysis (SOL 600 only).

Remark:

MARCINTC=2 spawns a secondary SOL 100 Nastran job to generate MPC's for the CINTC's. Licenses from the primary job are released, then reclaimed after the secondary job terminates. The secondary job stops prior to decomp and uses the standard SOL 101 licenses.

MARCINTF

Default = 1.0D-6.

Threshold value below which MPC coefficients generated by CINTC/GMBNDC are not considered. This parameter is ignored unless PARAM,MARCINTC,2 has been entered.

MARCITER

Default = 0, SOL 600 only.

Used to control fixed time stepping in SOL 600.

- 0 Fixed time steps or auto time steps will be controlled by PARAM,MARCAUTO.
- N Allows fixed time steps to be used without needing to set the maximum and minimum times to nearly the same value (using Marc's AUTO STEP option). This parameter triggers true fixed time stepping with the other advantages AUTO STEP has over methods such as AUTO LOAD. For example, it uses better numerical damping. If this parameter is entered with a positive integer (N), a value of 2 is placed in the AUTO STEP field 9 and N is placed in field 7.
- 1 This option is similar to PARAM,MARCITER,N (fixed time stepping will be used) except that the time comes from the NLPARAM or entry. This option is not available if the only "loading" is rigid contact or velocity control. See PARAM,MARCAUTO for a similar option. Do not use both MARCAUTO and MARCITER parameters.

MARCL001

Default = -1, SOL 600 only.

Determines whether Marc's POINT LOAD (without tables) 2nd datablock, 3rd field will be honored or not. If this value is set to 1 multiple loads at the same dof in the same subcase will usually be summed (however, see item d.).

- 1 The 2nd datablock 3rd field will be set to 1 (see Remark)
- 1 The 2nd datablock 3rd field will not be set to 1. If it is necessary to sum the loads, this must be accomplished in one of several ways:
 - a. Summed in the GUI or other method of data input
 - b. By setting PARAM,MNASTLDS,1
 - c. By setting PARAM,MARCTOTT,1
 - d. For certain simple multiple force/moment cases, SOL 600 can sum them and PARAM,MNASTLDS,1 must be specified to prevent loads from being specified multiple times. However, it may be difficult to determine whether the loads are complex or not, so the use of either option c or b is recommended.

Remarks:

1. For releases prior to MD Nastran 2010 and MSC Nastran 2009, this field was set to 1 to automatically sum any forces or moments that might have been entered more than once by the GUI (see PARAM,MNASTLDS for a discussion of ways this can happen). This field, when set to one, allows loads entered more than once at the same grid ID in the same subcase to be summed. However, experience shows that this field does not work in certain circumstances, particularly when "total loads" using PARAM,MARCTOTL are requested. PARAM,NASTLDS has been added beginning with MD Nastran 2010 and MSC Nastran 2009 to handle the cases where the 2nd datablock 3rd field does not work as expected.
2. To maintain backward compatibility with previous runs, set the following parameters:
PARAM,MARCL001,1
PARAM,MNASTLDS,0
PARAM,MARCTOTT,0
PARAM,MARCTOTL,0
3. If PARAM,MARCTOTL,1 is entered, PARAM,MARCL001,1 will be set unless PARAM,MARCL001,-1 is entered.
4. This parameter may be set in RC files.

MARCLOWE

Default = 0, SOL 600 only.

Used in conjunction with superelement matrices created by Marc.

- 0 Standard modulus values for all materials will be used.
- 1 All modulus of elasticity values will be changed to 1.0E-9 for the second Nastran run (when Nastran spawns another Nastran run using the SOL 600 continue option. This option is sometimes necessary for cases where Marc creates a superelement or substructure stiffness matrix but does not create a mass matrix. In this case, the second Nastran run will create the mass matrix using standard elements, density and other concentrated and distributed masses but the stiffness created by Nastran will be very low. Essentially the entire stiffness of the model will come from the stiffness matrices created by Marc.

MARCLUMP

Default = 0, SOL 600 only.

- 0 Consistent mass will be used for SOL 600 transient dynamics or eigenvalue problems with rotational masses (if applicable). (Default)
- 1 Lumped mass will be used for SOL 600 transient dynamics or eigenvalue problems with rotational masses (if applicable).
- 2 Consistent mass will be used for SOL 600 transient dynamics or eigenvalue problems without rotational masses.
- 3 Lumped mass will be used for SOL 600 transient dynamics or eigenvalue problems without rotational masses.

Remark:

This parameter is only used with dynamic analysis.

MARCMAT2

Default = -1.0, SOL 600 only.

Used if g33 = 0.0 on MAT2 entries. Marc will diverge if g33 = 0.0 for MAT2 entries. If the value entered is positive, the value is a multiplier of g11 and g22 to calculate g33 as follows:

$$g_{33} = \text{marcmat2}^*(g_{11} + g_{22})$$

MARCMAT3

Default = 1, SOL 600 only.

Used if Nastran has generated MAT2 from PCOMP and the MID of MAT2 is 30000001 is greater corresponding to MID3 for PSHELL.

If the value entered is 0, this entry is ignored regardless of the MAT2 MID value.

If the value entered is 1 the MAT2 entry will be mapped to Marc's ANISOTROPIC entry such that all Cij are zero except the following:

C55=g11
 C56=g12
 C66=g22

If the value entered is 2 the MAT2 entry will be mapped to Marc's ANISOTROPIC entry such that all C_{ij} are zero except the following:

C44=g11
 C45=g12
 C55=g22
 C66=g22

If the value entered is 11 the MAT2 entry will be mapped to Marc's ANISOTROPIC entry such that all C_{ij} are zero except the following:

C55=g11
 C56=g12
 C66=g22

If the value entered is 12 the MAT2 entry will be mapped to Marc's ANISOTROPIC entry such that all C_{ij} are zero except the following:

C44=g11
 C45=g12
 C55=g22
 C66=g22

Note: This entry is ignored unless the MAT2 MID is greater than 30000000.

MARCMATT

Default = -1 if parameter is not entered, SOL 600 only.

Determines if Marc input file will be created with materials using the table-driven formats or not.

- 1 Table-driven formats for materials will not be used. (Default)
- 1 Table-driven formats for materials will be used

Remark:

This parameter can be set in RC files.

MARCMEM, Value

Default = Program determines value, SOL 600 only.

- | | |
|-------|--|
| Value | If entered, the integer value entered here is the second field on Marc's SIZING entry (MAXALL) and is the main memory specification for memory in Marc. This value is entered in MW (the program multiplies it by 1,000,000). For example, if a value of 350 is entered, the number of 350000000 will be placed in the second field of the SIZING entry. |
|-------|--|

MARCMID3

Default = 0, SOL 600 only.

Controls whether MID3 will be set to the same value as MID2 when the Marc PSHELL option is used (designated by PARAM,MRPSHELL,1 or when the SMEAR option is used on the SOL 600 Executive Control statement.)

- 0 MID3 will not be changed (if zero or blank, it will remain zero or blank). (Default)
- 1 MID3 will be set to MID2. This improves the singularity ratio in some problems without appreciably changing the results particularly when orthotropic material properties are used. It is not necessary if PARAM,MMAT2ANI,11 or a similar option is used to specify anisotropic material properties.

MARCMNF

Default = 0, SOL 600 only.

Controls creation of an Adams MNF file by Marc for eigenvalue analysis when Marc is spawned from SOL 600.

- 0 MNF file will not be created. (Default)
- 1 MNF file will be created (for Marc 2003, the MNF file is located in the .t19 file, so PARAM,MARCT19,1 must also be specified).

Note: It is not necessary to use this parameter if the MDMIOUT Bulk Data entry is used to request an MSC Adams MNF file.

MARCMPCC

Default = 0, SOL 600 only.

Determines whether SOL 600 converts MPCs to stiff beams or not.

- 0 MPC's are not converted to stiff beams. (Default)
- 1 MPC's are converted to stiff beams.

Note: PARAM,MARCMPCC,1 should not be used if RBE's are converted to MPC's using PARAM,MARCRBE2 or PARAM,MARCRBE3.

MARCMPII

Integer, Default = 2 for DDM parallel executions, SOL 600 only.

Determines whether the MPI service on PC systems will remain running after the job finishes.

- 1 The MPI service will remain running after SOL 600 finishes
- 1 The Intel MPI service, ismpd.exe will be uninstalled before SOL 600 terminates.
- 2 The Intel MPI service, ismpd.exe will be started before each Marc execution and terminated at the end of each Marc execution from within the Nastran portion of SOL 600.
- 771 MSMPI rather than Intel MPI is requested (starting and/or stopping of the ismpd.exe Intel MPI service is ignored)

Remarks:

1. If SOL 600 terminates abnormally the service may remain running regardless of the value of this parameter.
2. Leaving the MPI service running will speed up the execution of multiple jobs.
3. This parameter may be entered in RC files.
4. Although ismpd.exe is started by the Marc script, it sometimes fails to start properly unless started outside the script. The user can either do this in his own script or set MARCMPII=2 to have Nastran do it. Either of these options improves reliability of the Intel MPI starting.

MARCND99

Default-see below. SOL 600 only.

Determines whether a set in the Marc input file to output all nodal quantities will be generated or not. If MARCND99=1, all Case Control nodal output requests must have the PLOT qualifier or the job may fail.

- 1 A set named ND999999 will be generated to output all nodes for at least one type of nodal output. This is the default of all Case Control nodal requests do not have (plot).
- 1 The set will not be generated. If all nodal Case Control requests have (plot) such as DISP(PLOT)=ALL, ACCEL(PLOT)=ALL, etc. the default is 1 even if the parameter is not entered.

MARCNOER

Default = 0, SOL 600 only.

Determines action to take when unsupported features are encountered.

- 0 The internal Marc translator will stop and generate FATAL ERRORS if unsupported features in Marc or in the internal translator are encountered.
- 1 If unsupported features are encountered, they are ignored, no FATAL ERROR messages are issued and if requested, Marc will be executed.

MARCOFFT

Default = -1, SOL 600 only.

Controls whether Nodal Temperatures are specified at the original or offset grid point or both grid points for beams and shells with offsets or connected by RBE2's (see note 2) when PARAM,MAROFSET is set to 0,2,3. For the default MAROFSET=1 where Marc handles the offsets and RBE2's are not added, this parameter is not applicable and will be reset to -1 internally. Applies only if MAROFSET=0.

- 0 Temperatures are applied both at the original grid point and at the offset grid point.
- 1 Temperatures are applied at the original grid point only.
- 2 Temperatures are applied at the offset grid point only.
- 1 Temperature loading is not altered in any way from the Nastran input.

Note:

1. Processing time can increase significantly if beam or shell offsets are present and param,marcofft is zero or greater unless PARAM,MOFFCORE,1 is used.
2. If nodes are connected by RBE2's and a temperature is applied at an independent node, the temperature will be applied at the independent node and the first dependent node of the rbe2 if MARCOFFT=0.
3. If MAROFSET is 0, 2 or 3 and a node has applied temperature and is also a part of a standard RBE2 with many grids, the job may abort or the results can be wrong.

MARCONTF

No Default, SOL 600 only.

- | | |
|------|--|
| Name | Name of a file name limited to 8 characters (16 characters if param* is used) used in conjunction with one of the CONTINUE options on the SOL 600 statement. If entered this file will be used as the input file for the second Nastran execution (after Marc has finished). If specified, this file will be used instead of automatically creating a file named jid.nast.dat from the original jid.dat input. This option allows more versatility in achieving exactly what is desired in the Nastran continuation run input at the expense of additional input data preparation. |
|------|--|

MARCOOCC

Default = 0, SOL 600 only.

- 0 Standard Marc memory management will be used. If the memory is sufficient the decomposition will be in core. If the memory requirements are too large, an out of core solution will take place.
- 1 An out of core solution will be forced if solvers 0, 2, 4 or 8 are used. This option triggers Marc parameter, OOC (without any other characters).

- 2 An out of core solution will be forced if solver 0, 2, 4 or 8 are used - available starting with Marc 2005 r2. This option generates Marc parameter OOC,0,1.
- 3 Same as option2 except Marc parameter version,4902 will be added. This option triggers Marc parameter OOC,0,1 plus version,4902.

MARCOPT

Default = 9, SOL 600 only.

Determines which bandwidth optimizer is to be used.

- 0 Marc does not optimize the bandwidth.
- 2 Cuthill-McKee bandwidth optimization is used
- 5 External user-supplied bandwidth optimization is used
- 9 Sloan bandwidth optimization is used. (Default)
- 10 Minimum degree bandwidth optimization is used (only available with the sparse solver)
- 11 Metis nested dissection algorithm (only available with multifrontal direct sparse solver)
- 9999 Set MARCOPT to -9999 if the OPTIMIZE entry is not wanted in the Marc file for example with use by the iterative solver.

MARCOSET

Default = 0, SOL 600 only.

Determines whether SOL 600 set names will be standard sets or “open sets” for nodes and elements. The standard Marc sets are:

```
DEFINE, ELEMENT, SET, PR00001
DEFINE, NODE, SET, ND001
```

The Marc open sets (OSET) are:

```
DEFINE, ELEMENT, OSET, PR00001
DEFINE, NODE, OSET, ND001
```

- 1 Standard sets are defined.
- 0 Standard sets are defined for small models and open sets are defined for large models.
- 1 Open sets are defined.

Note: For Parallel (DDM) analyses, it is sometimes necessary to set MARCOSET=1.

MARCOTIM

Default = 0, (SOL 600 only and is mapped to Marc's POST 2nd line 9th field.

Determines if Marc is to be processed at selective or at all output times.

- 0 or 1 Marc output data will be processed by Nastran at all converged output times. This option is similar to INTOUT=YES on the NLPARM entry.
- 2 Marc output data will be processed by Marc only at times near 1.0, 2.0, 3.0, etc. For this option, no additional output times will be available in the Marc.t16 and/or.t19 output files. This option is similar to INTOUT=NO on the NLPARM entry.
- N Marc output data will be processed by Marc every Nth increment for the.t16 and/or.t19 output files. (N > 2)

MARCOUTR

Default = 1 if Marc single file parallel input is used, Default = 0 if Marc multiple file inputs are used, see Note, SOL 600 only.

Determines how Marc t16 file results will be handled for SOL 600 parallel processing.

- 0 Multiple t16 files, one for each domain will be produced.
- 1 A single t16 file will be produced by Marc. This option requires Marc 2005 or later versions and the parallel run made using the "single file" input (PARAMARC KIND=0). (Default)

Note: Whether single file or multiple Marc inputs are used for parallel processing is determined by the PARAMARC Bulk Data entry.

MARCPARR

Default = 0, SOL 600 only.

Controls options for splitting an Marc file into parts for DDM.

- 0 All Marc files will be created during this run. (Default)
- 1 Nastran will be stopped after the single-processor file has been created and before DDM files are created. If desired, all files may be copied to a backup directory for use with MARCPARR=2.
- 2 The parallel files will be created starting with the single processor file created using the MARCPARR=1 option.
- 3 Same as MARCPARR=2 except the debug option MARCBUG=1 is turned on.

MARCPENT

Default = 0, SOL 600 only.

Option to specify if CPENTA is mapped to Marc brick element with degenerate nodes or actual penta elements (see Remarks).

- 0 Map CPENTA to Marc brick elements with degenerate nodes. (Default)
- 1 Marc CPENTA to actual Marc penta elements.

Remarks:

1. Option zero is the default to maintain backwards compatibility.
2. The t16 to op2 version is not available for option 1. That means no op2, xdb, f06 or punch file is available for option 1. Postprocessing must occur using the t16 or t19 file if option 1 is used.
3. Speed options such as invoked using param,mspeedse are available using option 1 only.

MARCPINN

Default = 0, SOL 600 only.

- 0 Pin flags will be included by created new nodes and appropriate MPC's by the translator in Nastran. (Default)
- 1 Pin flags will be ignored and the translator will continue.
- 2 A severe warning will be issued and Marc will not run.
- 3 The new Marc PIN CODE option will be used. feature,6901 will also be used so that extra nodes are not introduced.

Remark:

This parameter can be set in RC files.

MARCPLAS, n

Default = 3 if there is plasticity in the model. SOL 600 only.

This parameter effects the value of Marc's PLASTICITY parameter. The value of n can range from 0 to 6 (0 is the same as 3).

- 1 Additive decomposition using the mean normal method; small strain formulation
- 2 Additive decomposition using the radial return method; small strain formulation.
- 3 Additive decomposition using the mean normal method; large strain formulation using the updated Lagrange procedure.
- 4 Additive decomposition using the radial return method; large strain formulation during the updated Lagrange procedure.
- 5 Multiplicative decomposition (FeFp) using the radial return method and the three field variational principle; large strain formulation using the updated Lagrange procedure.

- 6 Advance nonlinear elements type 3 and 26 (plane stress), 18 and 30 (membrane) using multiplicative decomposition with the radial return method; large strain formulation using the updated Lagrange procedure.
- 1 Ensures that the Marc PLASTICITY parameter will not be used.

MARCPOS

SOL 600 only.

Determines whether to terminate Marc if a non-positive definite matrix is attempted.

- 0 The run will terminate if a non-positive definite matrix decomposition is encountered.
- 1 Non-positive definite matrices will be solved.

MARCPOST

Default, SOL 600 only.

Determines the format for the .t16 and .t19 files.

- 1 The .t16 and .t19 files will be created using the Marc default for the version of Marc that is executed.
- 0 Same as MARCPOST=9.
- 1 The .t16 and .t19 files will be created using Marc K2 formats.
- 3 The .t16 and .t19 files will be created using Marc K3 formats.
- 4 The .t16 and .t19 files will be created using Marc K4 formats.
- 5 The .t16 and .t19 files will be created using Marc K5 formats.
- 6 The .t16 and .t19 files will be created using Marc K6 formats.
- 7 The .t16 and .t19 files will be created using Marc K7 formats.
- 8 The .t16 and .t19 files will be created using Marc K8 formats.
- 9 The .t16 and .t19 files will be created using Marc 2000 formats.
- 10 The .t16 and .t19 files will be created using Marc 2001 formats.
- 11 The .t16 and .t19 files will be created using Marc 2003 formats.
- 12 The .t16 and .t19 files will be created using Marc 2005 formats.
- 13 The .t16 and .t19 files will be created using Marc 2005 r3 formats.

Note:

It is suggested that a small test case be executed and tested with your postprocessor to determine what version is necessary for your postprocessor.

MARCP99

Default-see below. SOL 600 only.

Determines whether a set in the Marc input file to output all elemental quantities will be generated or not. If MARCP99=1, all Case Control element output requests must have the PLOT qualifier or the job may fail.

- 1 A set named PR999999 will be generated to output all elements for at least one type of element output. This is the default if all elemental Case Control requests do not have (plot).
- 1 The set will not be generated. If all elemental Case Control requests have (plot) such as STRESS(PLOT)=ALL, STRAIN(PLOT)=ALL, etc. The default is 1 even if the parameter is not entered.

MARCPRN

SOL 600 only.

Controls the amount of contact information printed.

- 0 Detailed contact information is not printed.
- 1 Detailed contact information is printed (this is equivalent to Marc parameter PRINT,2,8).
- 2 Somewhat less detailed contact information is printed (this is equivalent to Marc parameter PRINT,2). Print constraint matrices associated with MPC's, RBAR, RBE2, RBE3 and the formable to deformable contact.
- 5 Marc print option. PRINT,5 will be used. Prints messages when changes in contact status occur.
- 25 Marc print options. PRINT,2,5 will be used.
- 258 Marc print options. PRINT,2,5,8 will be used. In addition to 2 and 5, also prints the displacement and reaction forces in the local coordinate system associated with formable to rigid contact.

For other print options, use the MARCIN Bulk Data entry.

MARCPRNG

Default = 0, SOL 600 only.

Determines whether geometry is printed in the Marc .out file.

- 0 Most geometry printing is suppressed.
- 1 All geometry is printed.

MARCPRNR

Default = 0, SOL 600 only.

Determines whether nodal stress and strain output is printed in the Marc .out file.

- 0 Nodal, stress and strain printing is suppressed.
- 1 Nodal, stress and strain printing will occur if Case Control options request it specifically or by default.

MARCPROG

No Default, SOL 600 only.

- prg “prg” is the name of a program to be executed instead of Marc. The program may be any program desired and must already be fully compiled on the computer system being run. prg is limited to 8 characters, however 16 characters can be used if PARAM* is entered. “prg” must be a compiled program, not a script or batch file. The name of the program must be in lower case (if not, it will be converted to lower case).

MARCRACC

Default = 0, SOL 600 only.

SOL 600 does not normally support RACC on the RFORCE entry. The default is to fatal a job when RACC is zero or blank. This parameter may be used to set it to zero or to obtain approximate results.

- 0 If MARCRACC is zero and RACC is nonzero, the job will terminate with an appropriate message (Default).
- 1 If MARCRACC is one and RACC is nonzero, RACC will be set to zero internally and the job will continue.
- 2 If MARCRACC is two and RACC is nonzero, the following will occur:
for brake squeal, coriolis loading and values (C1, C2, C3) will be placed in Marc ROTATION A 3rd datablock fields 4-6 as follows:

$$RR=\sqrt{R1^{**2}+R2^{**2}+R3^{**2}}$$

$$C1=RACC*R1/RR$$

$$C2=RACC*R3/RR$$

$$C3=RACC*R3/RR$$

Where RACC, R1, R2, R3 are described on the RFORCE entry.

For standard structural analysis if MARCRACC=2, the value of A in RFORCE field 5 will be altered using the following equation:

$$A_{\text{new}} = \sqrt{A^{**2} + RACC}$$

Marc's ROTATION A entry will be the same as if RACC=0.0 was entered unless A is zero, in which case the third line will have all zero entries. Results should be considered as an approximation.

Remarks:

1. See PARAM,MARCRCID for related SOL 600 RFORCE uses.

2. MARCRACC=2 only applies for brake squeal and only if the BRKSQ entry is used for releases prior to MD Nastran R2.1 and MSC Nastran 2007.

MARCRBAL

Default = 0, SOL 600 only.

This parameter is used only for eigenvalue analysis in the Nastran-Marc interface where natural frequencies or buckling modes need to be calculated using the deformed geometry from a nonlinear analysis. The parameter is only necessary if the last nonlinear increment created a non-positive definite matrix. When MARCRBAL=1 is set, the system will be rebalanced and a positive-definite matrix is assured. Do not use this parameter unless it is known that a non-positive definite system occurs just prior to eigenvalue analysis.

MARCRBAR

Default = 0, SOL 600 only.

Determines how RBAR is treated.

- | | |
|---|--|
| 0 | RBARs will be simulated using Marc's Servo Link. This option is best only for small incremental deformation and rotations. (Default) |
| 1 | RBARs will be converted to RBE2 with GN=GA, CM=123456, GM1=GB. The MARCRBE2 option specified will then be used to translate the converted RBARs to Marc. |

MARCRBE2

Default = 3, SOL 600 only.

- | | |
|----|--|
| ID | Determines the type of RBE2 used. The default should be used (parameter not entered) with models comprised of solid elements or a mixture of solid elements and other types of elements. |
| 0 | RBE2s will be simulated using Marc's Servo Link. This option is best only for small incremental deformations and rotations. |
| 1 | RBE2s will be simulated using Marc's TYING type 80 for translation and Servo Link for rotations. This option is capable of larger incremental deformations but requires small rotational increments. The MARCRBE2=1 option is only available if all 6 DOF's are specified in the CM field (4) of the RBE2 entry. |
| 2 | RBE2s will be simulated using the new RBE2 element introduced into Marc Version 2003 for a 2D analysis. |
| 3 | RBE2s will be simulated using the new RBE2 element introduced into Marc Version 2003 for a 3D analysis. |

Note: If RBAR, RROD or RTRPLT elements are found in the model and if MARCRBE2=2 or 3, these elements will be converted to equivalent RBE2's and used with the new Marc RBE2 element during the Marc execution (thus providing higher accuracy for large deformations and/or rotations).

The default should be used (parameter not entered) with models comprised of solid elements or a mixture of solid elements and other types of elements.

MARCRBE3

Default = 3 SOL 600 only.

Determines the type of RBE3 used.

- 0 RBE3s will be simulated using Marc's Servo Link. This option is best only for small incremental deformations and rotations (see option 4 for a similar alternative).
- 2 RBE3s will be simulated using the new Marc RBE3 element introduced into Marc Version 2003 for a 2D analysis.
- 3 RBE3s will be simulated using the new Marc RBE3 element introduced into Marc Version 2003 for a 3D analysis. (Default)
- 4 Same as MARCRBE3=0 except that all MPC's due to RBE3 will be placed ahead of all other MPC's. This option might improve the Marc solution for versions where Marc has not implemented AUTOMSET logic (those versions prior to MSC Nastran 2005 r3).

MARCREVR

Default = 0, SOL 600 only.

Specifies that all rigid surfaces need to be modified.

- 0 Rigid contact surfaces are correct as entered and no changes are made by the translator. (Default)
- 1 All rigid surfaces are entered backwards and will be reversed.

MARCREVRX

SOL 600 only.

Determines whether coordinates for NURBS2D (BCBODY) will be reversed or not if CQUADX, CTRIA6, and/or CTRIAX6 elements exist.

- 0 Reverse the coordinates (Y becomes X, X becomes Y).
- 1 In addition to reversing the coordinates (option 0) all points are also reversed. For example, if there are 4 points (1,2,3,4) they are reversed to (4,3,2,1). (Default)

- 1 Do not reverse the coordinates.
- 2 Reverse the coordinates for CQUADX and CTRIAZ abut not for CTRIAZ6.

MARCRIGD

Default = 0, SOL 600 only.

The parameter should only be entered if PARAM,MARMPCHK and/or PARAM,AUTOMSET options fail during a Marc execution.

- 0 All RBEi will not be converted to stiff beams or plates. (Default)
- 1 All RBEi will be converted to stiff beams or plates for the Marc input file using a stiffness scale factor from parameter MARCRSCL. This parameter allows such elements to have large rotations for versions of Marc which do not include large rotations of rigid elements.

Remarks:

1. This option may not be used if RBAR, RROD or RTRPLT elements are in the model. See PARAM,MSTFBEAM for an alternative.
2. See PARAM,MARCSCLR to specify a scale factor for the “default” properties of these stiff beams.

MARCSAME

Default = 0, SOL 600 only.

Determines whether SOL 600 runs with multiple subcase having the same LOAD ID or loads (see note below) in more than one subcase will be processed or not. SOL 600 will usually run under such circumstances but may get the wrong results.

Important:	Do not use this parameter if the loading contains enforced displacements or the results may be incorrect.
-------------------	---

- 0 The job will be aborted before Marc is spawned with a “Severe Warning” message. (Default)
- 1 The job will run to completion (if there are no other errors) and a standard Warning message will be issued.

It is recommended that if the same loads are to be used in multiple subcases that each subcase have a different LOAD ID. A typical file setup for SOL 600 should be setup in the following manner:

```
SOL 600,NLSTATIC PATH=1 STOP=1
CEND
DISP=ALL
STRESS=ALL
SPC=123
TEMP(INIT)=33
PARAM,MARCSAME,1
SUBCASE 1
LOAD=100
```

```
SUBCASE 2
LOAD=200
TEMP (LOAD)=300
BEGIN BULK
LOAD, 100, 1., 1.0, 1000, 1.0, 2000
LOAD, 200, 1., 1.0, 1000, 1.0, 2000
PLOAD4, 1000, 10, 20.0
PLOAD4, 2000, 20, 25.0
(Other Bulk Data entries)
ENDDATA
```

Note:

In the above example if the Bulk Data LOAD entries are changed to:

```
LOAD,100,1.,1.0,1000,1.0,2000
LOAD, 200,0.9,1.0,1000,1.0,2000
```

The loads are the same before the overall scale factor (1.0 and 0.9, respectively) are applied and PARAM,MARCSAME,1 is needed. However if the entries were as follows, it is not needed because the loads before the overall scale factor is applied are different.

```
LOAD,100,1.,1.0,1000,1.0,2000
LOAD,200,1.0,0.9,1000,1.0,2000
```

MARCSCLR

SOL 600 only.

Sets property values for stiff beams when PARAM,MARCRIGD or PARAM,MSTFBEM are used in SOL 600.

The value of this parameter scales the “default” properties of stiff beams or plates if parameter MARCRIGD=1. The “default” (unscaled) values for the stiff beams are A = 10, I = 100 (both directions) J = 200, shear area = 5 and plate thickness of 1.5. Linear scaling is used for all areas and thickness, all inertia terms are multiplied by the square of the Value entered.

MARCSETS

Integer, Default = 0, SOL 600 only.

Controls the type of sets used for the Marc input file element and node sets.

- | | |
|---|---|
| 0 | Sets will have the format DEFINE, ELEMENT, ... or DEFINE, NODE, ... |
| 1 | Sets will have the format DEFINE, ELSQ, ... or DEFINE, NDSQ, ... |

Remarks:

1. PARAM,MARCSETS,1 can speed up certain large Marc jobs by a considerable amount.
2. The parameter may be set in RC files.

MARCSETT

Default = 0, SOL 600 only.

- 0 The current environment is not printed. (Default)
- 1 The current environment is printed in the .f06 file. A user program named eodenv.f must be compiled, linked and placed in the input file directory. The contents of eodevn.f resembles the following:

```
Program eodenv
call system("set")
stop
end
```

MARCSINC

Default = 0, SOL 600 only.

This parameter controls how often a spline file is written if the spline option (analytical contact for deformable bodies) is requested. If this parameter is not entered or if it is 0 or -1, then a file is not written. If N is greater or equal to 1, then every nth time step is written. Spline files have the extension *.mfd which may be processed by MSC Mentat.

MARCSIZ3, Value

Default = Program determines value, SOL 600 only.

- Value If entered, the integer value entered here is the third value on Marc's SIZING entry representing the maximum number of elements.

MARCSIZ4, Value

Default = Program determines value, SOL 600 only.

- Value If entered, the integer value entered here is the fourth value on Marc's SIZING entry representing the maximum number of grid points.

MARCSIZ5, Value

Default = Program determines value, SOL 600 only.

- Value If entered, the integer value entered here is the fifth value on Marc's SIZING entry representing the maximum number of constrained degrees-of-freedom.

MARCSIZ6, Value

Default = Program determines value, SOL 600 only.

Value If entered, the integer value entered here is the sixth value on Marc's SIZING entry representing the maximum number of elements in the largest list of distributed loads (the internal Marc to Nastran translator generates these one at a time, so this value is normally 1).

MARCSLHT

Default = 5, SOL 600 only.

Number of layers through the shell thickness used to integrate shell and beam elements. For linear behavior, N=1 is sufficient. For most plasticity problems, N=5 is adequate. For extremely nonlinear plasticity problems N=11 should be used. SOL 600 requires that N be 5 or larger. If N is entered with a positive value less than 5, SOL 600 will set it to 5. To use values smaller than 5, enter N as a negative number. The absolute value will be used, however the job may fail or results may be incorrect if the model has plasticity.

Note:	Use of PARAM,MARCDEF can effect the value of Marc's SHELL SECT parameter if PARAM,MARCSLHT is not entered. To eliminate SHELL SECT from the Marc file set N to -9999.
--------------	---

MARCSOLV

Default = 8, SOL 600 only.

Determines the solver to use for Marc.

- | | |
|----|--|
| 0 | The profile solver will be used (solver 8 should normally be used instead) |
| 2 | The Sparse Iterative Solver will be used. |
| 4 | The Direct Sparse Solver will be used. |
| 6 | A hardware-provided solver will be used. |
| 8 | A sparse solver similar to the one used by Nastran will be used (Default) |
| 9 | The CASI element based iterative solver will be used. |
| 10 | The mixed direct/iterative solver will be used. |
| 11 | Pardiso direct solver (see param,mrthread) |
| 12 | MUMPS parallel direct solver (see param,mrthread) |

Note:	If any NLSTRAT entries are entered, the solver type must be specified using the IOLSVER option of NLSTRAT rather than this parameter.
--------------	---

MARCSPCC

Default = 0 if no SPCD's are entered or 2 if there are SPCD's in the input.

Determines how SOL 600 will proceed if non-zero value enforced displacements are found in SPC entries for multiple subcase models.

- 0 A fatal error will be issued and the user will be requested to change all non-zero value enforced displacements to SPCD's.
- 1 The job will continue without any changes (wrong results could be produced).
- 2 All displacements on SPC entries will be set to blank and the job will continue (will usually produce the correct results if SPCD's with the correct values are specified).

Remark:

Non-zero enforced displacements should always be specified using SPCD (not SPC) for SOL 600. It is not necessary to specify SPC's for degrees of freedom where SPCD's are applied for SOL 600, but it will not do harm unless non-zero enforced displacements are specified on the SPC entries (either in addition to being specified on SPCD's or without having any SPCD's). If there is only one subcases, the non-zero enforced displacements may be specified on SPCD's and/or SPC's, however for multiple subcases wrong results can result even if the displacements specified on the SPC's and SPCD's are the same. The results will be even further off if the displacements are different. The remedy is to ensure that the SPCD's reflect the correct applied displacements, and then either remove the applied displacements from all SPC's or to set MARCSPCC=2.

MARCSTIFF, Time

Default = 1.0, SOL 600 only.

- Time This parameter specifies what time matrices entered using PARAM,MARCFILi will be used in a Nastran solution. The file may contain matrices at several times, but only the matrices specified by the parameter will be used. This parameter is not usually used, MRMTXNAM,NAME is used instead.

MARCSTOP

SOL 600 only.

- 0 Normal Marc execution when spawned from Nastran.
- 1 Marc will exit (with exit code 7) after phase 0 (corresponds to Marc parameter stop).

MARCSUMY

Default = 0, SOL 600 only.

Determines if the summary of maximum values is to be printed.

- 0 A summary of maximum displacements, stresses and strains will be printed in the Marc output file. (Default)
- 1 The summary of maximum values is not output.

MARCT16

Default = 2, SOL 600 only.

Controls generation of a Marc t16 file. MARCT16=0, Marc does not generate a .t16 output file.

- 1 or 0 Does not generate a t16 file.
Parameter omitted generates a t16 file. All entries are controlled by the MARCOUT Bulk Data entry, or if MARCOUT is not specified, by the default shown in option 2 below.
- 1 Generates a .t16 output file with the following post codes:
11 11,1 11,N 12 12,1 12,N 13 13,1 13,N 14 14,1 14,N
15, 15,1 15,N 16 16,1 16,N 17 17,1 17,N 18 18,1 18,N
7 7,1 7,N 27 27,1 27,N 301 301,1 301,N 321 321,1 321,N
341 341,1 341,N 401 401,1 401,N
Nodal: 1, 2, 3, 4, 5, 6, 35, 36, 37, 38, 39, 40, 46, 48, 51, 52
- 2 Generates a .t16 output file with the following post codes. (Default)
301, 301,1 301,N 341 341,1 341,N 47
Nodal: 1, 2, 3, 4, 5, 6, 34, 35, 36, 37, 38, 39, 46, 48
- 3 Generates a .t16 output file with the following post codes (op2, xdb, f06, punch results cannot be made using this option):
301, 341 47
Nodal: 1, 2, 5, 6, 34, 35, 36, ,37, 38, 39
- 4 Generates a .t16 output file with the following post codes (op2, xdb, f06, punch results cannot be made using this option):
301 341 47
Nodal: 1

Remarks:

1. This entry is used as an easy way to control which results are placed on Marc's op2 file. All entries can be overridden using the MARCOUT Bulk Data entry. For MARCOUT values of 1 and larger, MARCOUT should be omitted for the input file. MARCOUT should not be used if PARAM,MARCT16 is entered.
2. The frequency of output is controlled by the NLPARM or Bulk Data entries (variables INTOUT and NO respectively).
3. Items such as 341,1 and 341,N designate stresses at the bottom and top surfaces (for applicable elements). Items such as 341 designate stress at mid-thickness.

4. When stress 341 is specified for models with composite elements, the value will be changed to 391 to obtain stresses in the fiber (layer) direction.
5. Consult Marc Volume C documentation for the meaning of the above blocks. Option 1 provides most of the structural output anyone might want, option 2 provides total strain, Cauchy stress, displacement and contact information at the top, center and bottom of surfaces. Option 3 provides the information of option 2 but only at the center (not at the top and bottom).
6. It is necessary to generate a t16 file in order to produce op2, xdb, f06 or punch results.
7. Op2, xdb, punch and f06 results can only be created using option 1 and 2 although option 0 can also be used if the selected outputs are the same as option 1 or 2.
8. If a Bulk Data VCCT entry is included in the model, nodal post codes 57-64 and 74 will be added to any of the previous defaults if MARCOUT is not entered in the model.

MARCT19

Default = 0, SOL 600 only.

- | | |
|---|--|
| 0 | Marc does not generate a .t19 output file. (Default) |
| 1 | Generates a .t19 output file. |

MARCTABL

Default = 0, SOL 600 only.

Determines if contact table is to be generated.

- | | |
|---|--|
| 0 | Contact tables will be generated for the main Marc input (phase 0) and for each subcase if specified by the user (Default). When MARCTABL=0, each subcase may have a BC CONTACT Case Control command and a matching BCTABLE ID entry. In addition, Marc's "Phase 0" entry is supported by entering a BCTABLE with an ID of zero (or 1,000,000). Each separate BCTABLE will reference the BCBODY entries defined which, in turn, reference BSURF entries. (Default) |
| 1 | No contact tables will be generated and all contact bodies (if any) will be placed in the main input data section. Contact will thus be the same for all subcases. When MARCTABL=1, there must only be one BCTABLE entry in the file. There must only be one BC CONTACT command in the Case Control and it must be above all subcases. The BC CONTACT and BCTABLE entry must have the same ID. The BCTABLE entry can reference several BCBODY entries which, in turn, reference BSURF entries. |

MARCTEDF

Character*8, no Default, SOL 600 only.

Enter the Marc nthcnt file name without extension. Use this option only if PARAM,MARCTEDN,1 is entered (the file name without extension is limited to 8 characters). The characters ".nthcnt" will automatically be appended to the name specified.

MARCTEDN

Default = 0, SOL 600 only.

Determines whether a thermal contact analysis will use the Marc jid.nthcnt file generated by this run or use one input by the user.

- 0 Analysis uses the Marc jid.marc.nthcnt file generated in this run. (Default)
- 1 Analysis uses a Marc nthcnt file generated by the user or in a previous run.

Note: If MARCTEDN=1, PARAM,MARCTEDF below must be entered to specify the file name.

MARCTEMP

Default = 1, SOL 600 only.

- 0 The scratch files produced by Marc will be in the same directory as the Nastran input file.
- 1 The scratch files produced by Marc will be in the same directory as the Nastran scratch files. (Default)

Note: The Marc scratch files cannot be split.

MARCTETT

Integer, Default = 0, SOL 600 only.

Determines how CTETRA elements will be mapped to marc.

- 0 4-node or 10-node Nastran CTETRA elements will be mapped to the Marc tet-4 or tet-10 elements.
- 11 4-node or 10-node Nastran CTETRA elements will be mapped to Marc solid element type 7. for tet-10 elements, the mid-side nodes are neglected. the 4 CTETRA nodes (N1,N2,N3,N4) are mapped to the 8 solid shell nodes as follows N1,N2,N3,N4,N4,N4,N4.
- 12 Same as MARCTETT=11 except debug is turned on (for the tetra conversion only)

Remark:

MARCTETT should not be greater than zero for heat transfer or Herrmann elements.

MARCTIEC

Default = 1, SOL 600 only.

- 1 Transient Time Integration Error Check for Marc's AUTO STEP method. A value of 1 turns the check on. (Default)
- 0 A value of 0 turns the check off. Turn the check off to match Marc results for version prior to Marc 2003 r1.

MARCTOL

Default = 0, SOL 600 only.

Determines the method of convergence tolerance.

If parameter MARCTOL=0, convergence tolerances are based on residuals (loads). If parameters MARCTOL and MARCTVL are not entered, the tolerances are determined by and/or NLPARM Bulk Data entries. However, parameters MARCTOL and MARCTVL provide extra control over these convergence tolerances particularly in the case where or NLPARM specify more than one convergence type (such as load and energy).

- 1 Convergence tolerances are based on displacement.
- 2 Convergence tolerances are based on strain energy.
- 4 Convergence is achieved when either residual or displacement satisfies the criteria.
- 5 Convergence is achieved when both residual and displacement satisfies the criteria.

MARCTOTD

Default = 0, SOL 600 only

Determines whether for SOL 600 dynamic analyses will use full table association.

- 0 Full table vs time association is not used. (Default)
- 1 Full table vs time association will be used

Remarks:

1. param,marctott,1 must also be set if MARCTOTD=1
2. This parameter may be set in RC files.

MARCTOTL

Default = 0, SOL 600 only

Determines whether total or incremental loads are used in a SOL 600 static nonlinear analysis.

- 0 Incremental loads are used. (Default)
- 1 Total loads are used

Remarks:

1. Warning - If MARCTOTL=1, MRFOLLOW3=1 will be set if possible (see PARAM,MRFOLLOW3)
2. This parameter may be set in RC files.

MARCTOTT

Default = 0, SOL 600 only

Determines whether total loads, including pressures, gravity, spcd, etc., with associated tables are used in a SOL 600 static or dynamic analysis (Remark 3.)

- | | |
|---|---|
| 0 | Total loads with tables are not used. (Default) |
| 1 | Total loads with tables are used |

Remarks:

1. Warning - If MARCTOTT=1, MRFOLLOW3=1 will be set if possible (see PARAM,MRFOLLOW3)
2. This parameter may be set in RC files.
3. For full table association, dynamic analyses require PARAM,MARCTOTD,1 to also be set.

MARCTUBE

Default = 0, SOL 600 only.

Determines whether CTUBE maps to Marc element 31 or 98 (Default = 0 if this parameter is omitted).

- | | |
|---|---|
| 0 | CTUBE elements map to Marc tube element 31 (see Remark) (Default) |
| 1 | CTUBE elements map to Marc element 98 (elastic beam) |

Remarks:

Marc element 31 does not support thermal loading or creep. If the model contains thermal Loading or creep (even if not applicable to the CTUBE elements, option 1 is always used.

MARCTVL, Value

SOL 600 only.

- | | |
|-------|--|
| Value | If parameter MARCTVL is entered, it must have a real value. The value entered is the convergence tolerance used by Marc (see PARAM, MARCTOL). If parameters MARCTOL and MARCTVL are not entered, the tolerances are determined by and/or NLPARM Bulk Data entries. However, parameters MARCTOL and MARCTVL provide extra control over these convergence tolerances particularly in the case where or NLPARM specify more than one convergence type (such as load and energy). |
|-------|--|

MARCUSUB, chr

No Default, SOL 600 only.

chr chr is the name of a “user subroutine” to be included in the Marc run. chr is limited to 8 characters without the .f extension. This file must be located in the same directory as the Nastran input data. The subroutine name must have all lowercase letters. chr should be in lower case to prevent confusion. (Nastran will convert the file name to uppercase, but it will be reconverted to lower case when Marc is spawned.) Any user subroutine available to Marc may be specified. Multiple user subroutines must be combined into one file.

Restriction: The computer must have a Fortran compiler and Linker and the Fortran compiler must be the same as used to create the original Marc executable (see the Marc installation manual). The Marc input file does not as yet call out user subroutines, so manual editing of the Marc input file may be necessary in some cases to invoke them. Existing regular Marc subroutines can be modified and handled in the same manner if available to you.

Note: If more than one user subroutine is required, all should be combined into one file before execution.

MARCVERS

SOL 600 only.

This parameter has been replaced by the following parameters:

param,mrconver
param,mrcontab
param,mrsprver

MARCWDIS

Default = 1, SOL 600 only.

Determines whether Marc parameter section DIST LOADS Is written or not. If any of the three values for DIST LOADS are entered (see PARAM,MARCDIS2 PARAM,MARCDIS3 PARAM,MARCDIS4) it will be written. If PARAM,MARCWDIS,1 is entered, it will be written. If PARAM,MARCWDIS,-1 is entered, it will not be written.

Caution: Nastran cannot estimate these values very well and produces overly conservative numbers that sometimes leads to failure of the Marc run due to lack of memory. We suggest that the user should use this parameter sparingly and enter the MARCDIS2, MARCDIS3 and MARCDIS4 values for best Marc memory usage.

MARCWELD

Default = 0 if this parameter is omitted, SOL 600 only.

Determines how CWELD/PWELD elements will be translated to Marc.

- 0 Add extra nodes and elements to simulate the weld.
- 1 Use Marc CWELD/PWELD formulation (available starting with MD Nastran R2 and MSC Nastran 2007).

This parameter may be set in RC files.

MARELSTO

Default = 0, SOL 600 only.

Determines whether Marc's parameter ELSTO will be created.

- 1 ELSTO will not be created.
- 0 ELSTO will be created only for large models as determined by Nastran. The value of ELSTO will be 40960 except for parallel runs (where a PARAMARC entry exists in the input file) in which case ELSTO will not be created. (Default)
- >0 ELSTO will be created with the value specified used as the Marc ELSTO parameter whether or not the run uses parallel processing.

MARGPFEL

Default = 0, SOL 600 only.

Used to determine if Marc GRID FORCE output will occur by element, by node, or both ways.

- 1 Output is by element only.
- 0 Output is by node only. (Default)
- 1 Output is by both element and node.

Notes:

1. This parameter effects the contents of the Marc jid.marc.grd file. Option 0 or 1 is required for GPFORCE output in op2, xdb, punch, and/or f06 files.
2. This parameter can be set in RC files.

MARFACEA

Default = 1, SOL 600 only.

Face number for "A" side of weld if welds are made of solid elements. Not used if welds involve plates or shells.

MARFACEB

Default = 1, SOL 600 only.

Face number for “B” side of weld if welds are made of solid elements. Not used if welds involve plates or shells.

MARFATAL

Default = 1, SOL 600 only.

Determines whether non-existent grid id's for BCBODY entries will cause fatal errors or not.

- | | |
|---|---|
| 0 | Non-existent grid ID in BCBODY entries will only cause warning messages. The grid ID will be reset to zero and the job will continue. The results should be checked carefully. In some cases this could result in an entire contact body being skipped. |
| 1 | Non-existent grid ID in BCBODY entries will cause FATAL ERRORS. (Default) |

MARGPFOR

Default = 0, SOL 600 only.

Used to determine whether GPFORCE is active for SOL 600. A Case Control request GPFORCE must be specified to obtain grid point forces (see note 4). The grid point forces are output from Marc on a file named jid.marc.grd. The t16op2 program reads this file, puts the data on the f11 file along with displacements, stresses, etc. After t16op2 finishes, the f11 file is brought into DBALL, from which DMAP generated on the fly can produce op2, xdb, punch, or f06 output.

- | | |
|----|---|
| -1 | GPFORCE is ignored (Default). |
| 0 | GPFORCE is output for the last time in the last subcase only. |
| 1 | GPFORCE is output at all times given in the .sts file. |
| N | GPFORCE is output for times 1, 1+N, 1+2N, etc. |

Notes:

1. This parameter can be set in RC files.
2. Output is available only in the new Marc jid.marc.gid file.
3. The Case Control GPFORCE request must be above all subcases, or the same within all subcases.
4. The jid.marc.grd file can become very large if option 0 or N is not used.

MARHEATM

Default = 0, SOL 600 only.

Determines whether a file named heatm.rc is necessary to run the second phase of SOL 600 heat transfer initial contact job.

- 0 heatm.rc is not required. Defaults will be used. The defaults are scr=yes batch=no mem=80mw
- 1 A heatm.rc file will be supplied by the user in the same directory as the original Nastran input file. The heatm.rc can contain any information used by other rc files except that batch=no. If the original Nastran input file is named jid.dat (or jid.bdf) and out=jid is specified, the final output will be in files such as jid.f06, jid.op2, jid.xdb. If out=jid is not specified the final output will be in files such as jid.nast.f06, jid.nast.op2, jid.nast.xdb.

MARHTPRT

SOL 600 only.

Controls heat transfer output in the Marc .out file.

- 0 Do not print any output except for summary table.
- 1 Print the nodal temperatures.
- 2 Print all possible nodal heat transfer output.

MARIBOOC

Default = 0, SOL 600 only.

- 0 Incremental backup data will be stored in memory. (Default)
- 1 For large problems, incremental backup data will be stored on disk. This option triggers Marc's parameter IBOOC.

MARIPROJ

Default = 0, SOL 600 only.

Flag to determine if auxiliary nodes of a CWELD will be projected on the model or not. (If this parameter is not entered)

- 0 Nodes are not projected on to the model.
- 1 Nodes are projected on to the model.

MARLDCMB

Default = -1, SOL 600 only.

Determines whether extraneous loads in the input file will be filtered out at an early stage to save computer time. This applies only to SOL 600, ID where ID=1, 3, 6, 101, 105, 106.

- 1 Extra loads will not be filtered out.
- 2 Extra Bulk Data LOAD entries will be filtered out early in SOL 600 to save computer time.

Note:

If PARAM,MARLDRMV,-1 is not entered, extra FORCE, MOMENT and PLOAD4 entries not used in the present analysis will also be filtered out early in SOL 600. Computer time saving can be appreciable for certain models with many extra loads.

MARLDRMV

Default = 1, SOL 600 only.

Determines whether extraneous FORCE, MOMENT, and/or PLOAD4 entries in the input file will be filtered out at an early stage to save computer time. This applies only to SOL 600, ID where ID=1, 3, 6, 101, 103, 105, 106.

- 1 Extra loads will not be filtered out.
- 1 Extra Bulk Data FORCE, MOMENT, and/or PLOAD4 entries will be filtered out early in SOL 600 to save computer time. (Default)

Note:

This option will be ignored (same as PARAM,MARLDRMV,-1) if PARAM,MARLDCMB,-1 is entered.

MARMPCHK

Default = -1, SOL 600 only.

Determines whether Marc parameter MPC-CHECK is written.

- 1 MPC-CHECK is not written. (Default)
- 1 Apply the MPCs in the default order:
 1. MPCs obtained from SERVO LINK option.
 2. MPCs obtained from INSET option.
 3. MPCs obtained from TYING, RBE2, or RBE3 options (the actual order follows from the order of these options in the model definition block of the data file.)
 4. MPCs obtained from CYCLIC SYMMETRY option.
 5. MPCs obtained from CONTACT option.

Print a warning message if a tied degree of freedom is being used by a subsequent MPC.
- 2 Same as 1, but instead of warning, a fatal error message is printed and the analysis will stop with exit 2001.

- 3 Try to rearrange the MPCs in such a way that a tied degree of freedom will not be used in a subsequent MPC. If this reordering cannot successfully be completed, print a fatal error message and stop the analysis with exit 2011.
- 999 Skip MPC-CHECK entirely, prevent Marc's error message "Exceed the max trial to order RBE2/3". This option should be used in conjunction with PARAM,AUTOMSET,YES only. (Marc feature,3901 is set)

Note: This parameter can be set in RC files.

MARMPCID

Integer, Default = 1, SOL 600 only

Determines the ID of the MPC Case Control command for SOL 600 jobs using a CONTINUE=101 to CONTINUE=159 (see SOL 600 Executive Control statement, CONTINUE option).

- 0 or 1 A line MPC=1 will be inserted in the Case Control (remark)
- N A line MPC=N will be inserted in the Case Control (N must be between 2 and 9999)

Remark:

When models with contact and if BCPARA,INITCON,4 is specified, MPC's for each increment will be produced and placed in files named jid.marc.conmpc_xxxx where xxxx ranges from 0001 to the number of increments in the solution. The ID of these MPC's is usually equal to one. If for some reason it is not one, the actual value may be specified by this parameter.

MARMTLCK

Default = 0, SOL 600 only.

Determines whether a check of various property-material combinations for SOL 600 will be made or not.

- 0 The checks will not be made. (Default)
- 1 Check will be made. These take extra computer processing time and for most models are not required. The user should turn on these check if he is in doubt if any property-type of material combinations entered into the model may be in error. Current check made are for the following illegal combination.
PSOLID/MATHE (model should be PLSOLID/MATHE)
PSOLID/MATHP (model should be PLSOLID/MATHP)

MARNOCID

Default = 0, SOL 600 only.

SOL 600 does not support MCID defined by cylindrical or spherical coordinate systems. This parameter determines whether MCID defined by cylindrical or spherical coordinate systems will be ignored or “fatalized out” for shell and solid elements.

- 0 The run will be “fatalized out” if MCID defined by cylindrical or spherical coordinate systems are found for shell or solid elements. (Default)
- 1 MCID defined by cylindrical or spherical coordinate systems for shell or solid elements will be ignored.

MARNOSET, Name

No Default, SOL 600 only.

- Name Character. If entered, this parameter will not write out a set with the specified name. This is useful, when portion of the model specify sets that are not actually used in the present analysis. Up to 20 of these can be specified.

MAROFSET

Default = 1, SOL 600 only.

Determines how beam and shell offsets are applied.

- 0 Extra grids and rigid elements will be created to model the offsets.
- 1 Marc will automatically handle offsets for beam and shell elements. No extra grids or elements will be created. The offsets will be found in Marc’s GEOMETRY data. (Default)
- 2 Marc will automatically handle offsets for beam elements only.
- 3 Marc will automatically handle offsets for shell elements only.

Note: If MAROFSET is 1 or 2, the beam orientation can be specified using the CBAR/CBEAM “BIT” flag. It is suggested that only combinations, GGG or BGG be used.

MARPLANE

Default = 0, SOL 600 only.

For composite structures described using PCOMP, together with CQUAD4, CQUADR, etc., it is not possible to tell whether a standard 3D shell or a plane strain shell has been modeled. If MARPLANE is set to 1, such composite models will be assumed to be plane strain (as if a PLPLANE property had been entered rather than PCOMP).

MARPROCS

Determines the number of domains to be used in a DDM parallel execution.

- 1 Use 1 processor.
- 2 Use 2 processors.
- N Use N processors.

Remarks:

1. This parameter is not intended to be used by parallel jobs using several different machines.
2. This parameter and PARAMARC may not both be entered in the same run.
3. This parameter and PARAM,MRTHREAD may both be used in the same job.
4. The parameter may be used with most of the solver types except for the MUMPS solver (type 12).

MARRBAR2

Default = 1, SOL 600 only.

If MARCRBAR is set to 1, all CMA fields will be changed to 123456.

- 0 CMA field will remain as coded by user, Marc run may fail.
- 1 CMA field will be set to 123456. (Default)

MARROUTT

Default = -1, SOL 600 only.

Determines whether an inconsistent set of outputs between the Marc t16 file (selected using MARCOUT) and standard Nastran output selected using Case Control requests (and param,post) is allowed or not.

- 1 Inconsistent output is not allowed, if the outputs are inconsistent, the job will fatal early before spawning Marc. (Default)
- 1 Inconsistent output is allowed. However, if the output requests are inconsistent, the job may fail during the t16op2 conversion after Marc has finished.

Remark:

Nastran outputs in the op2, xdb, punch or f06 files are obtained by converting the Marc output from the t16 file. If output is not available in the t16 file, or if it is in the wrong form, errors will occur during the t16op2 conversion. For most problems, if op2, xdb, f06 and/or punch output is required, it is best not to enter MARCOUT. In most cases MARCOUT should only be used if post-processing using the Marc t16 file is to be done.

MARSHRII

Integer ≥ 0 , Default = 0, SOL 600 only.

Controls how the properties of bars added to the edges of CSHEAR elements when f1 or f2 of PSHEAR is nonzero.

- 0 CBAR elements at the edges of CSHEAR elements (if any) will only have area (I1, I2, J are all 0.0). See PSHEAR Remark 2 as to how the area of these bars is calculated.
- > 0 The value specified will be used to calculate I1, I2 and J from the area as
 $I1=I2=A/MARSHRII$. $J = 2.0 * I1$

Remark:

This parameter must be entered in the Case Control above any subcases.

MARTET10

Integer, Default = 10, SOL 600 only

Controls how to treat badly shaped 10-node tetra elements in SOL 600 if renumbering the element does not correct problems.

- 1 Stop the analysis if all possible re-numbering options do not correct the bad element. See Remark 4.
- 0 Continue the analysis with the best re-numbering option and let Marc possibly fail with inside out messages.
- 1 If all possible re-numbering options fail, change the bad tet 10 element to tet 4. See Remarks 1 and 5.
- 2 If all possible renumbering options fail, deactivate the bad tet 10 element. (See Remarks 2, 3, and 5.)
- 3 Change the coordinates of bad tet 10 midside nodes to lie at the midpoints of the vertices. (See Remark 6.)

Remarks:

1. Changing bad tet 10 elements to tet 4 may ignore continuity on the edges of adjacent tet 10 elements unless they also get changed to tet 4 elements. If many tet 10 elements are changed to tet 4 elements, the accuracy of the stresses could be inaccurate. In addition, if the elements are used in contact, the same contact body can't have mixed elements with midside and without midside nodes.
2. Deactivating bad tet 10 elements creates holes in the model. If only a few bad elements are deactivated in a fine-grid model, the overall behavior will usually be nearly the same as if the elements were good and remained in the model, however the stresses near the deactivated elements may be too large. If many bad elements are deactivated, all analysis results will probably be inaccurate.
3. Setting PARAM,MARTET10,2 will deactivate all types of elements for which Marc gets inside out elements (not just tet 10 elements). This option maps to Marc's 10-DEACT parameter.

4. This parameter is available starting with MD Nastran 2010. Prior to this release, only option 0 was available.
5. Use options 1 or 2 with extreme care and only when all other alternatives including remeshing using the gui have been exhausted.
6. If martet10=3, any midside nodes whose coordinates are changed and lie on the boundary of the structure will redefine the exterior of the structure by a small amount. Also, if the element is used in contact and the stress-free option is invoked, the coordinates may be changed again and the elements may become badly shaped again.

MARTETIN

Integer, Default = 0, SOL 600 only.

Controls whether additional information messages are output to the .f06 file or not when param,martet10 is set to a positive value.

- | | |
|---|---|
| 0 | Do not output additional tet10 information messages. |
| 1 | Output additional tot10 information messages regarding checking of all tet10 elements. (The .f06 file could be very large using this option.) |

MARUPDAT

Default = 1, SOL 600 only.

If this parameter is omitted, the Updated Lagrange method will be used if plasticity is involved. Ensure that all elements in the analysis are capable of using the Updated Lagrange method. If not, enter PARAM,MARUPDAT,-1. The Updated Lagrange Method is more accurate for many problems and also runs faster for some problems.

- | | |
|----|---|
| -1 | The Total Lagrange solution procedure will be used when Marc is executed from Nastran. |
| 1 | The Updated Lagrange solution procedure will be used when Marc is executed from Nastran. This corresponds to Marc parameter update. (Default) |
| 2 | The updated Lagrange solution with large rotations for beam elements. This corresponds to Marc parameter (see Marc VOL C documentation for details): update,0,2 |
| 3 | The following Marc parameter will be set, in which LARGE DISP need not be specified elsewhere (see Marc VOL C documentation for details): update,0,2,1 |
| 4 | The following Marc VOL C parameter will be set, in which case LARGE DISP need not be specified elsewhere. (see Marc documentation for details): update,0,,1 |
| 5 | The following Marc parameter will be set (see Marc VOL C documentation for details): update,0,1 |

MARVFCUT

Default = 0.0001, SOL 600 only.

Controls the fraction of the maximum view factor that is to be used as a cutoff. View factors calculated below this cutoff are ignored. (Used in SOL 600 radiation heat transfer only.)

MAUTOSPC

Default = -1, SOL 600 only.

Determines whether Marc AUTOSPC will be added.

-1	Do not add AUTOSPC. (Default)
1	Add AUTOSPC to Marc's parameter's section. It will remain throughout the run.
2000	Turn on AUTOSPC by adding the integer 1 to Marc's model definition SOLVER option
2001	Turn on AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 1
2002	Turn on AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 2
...	
2999	Turn on AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 999
3000	Turn off AUTOSPC by adding the integer 1 to Marc's model definition SOLVER option.
3001	Turn off AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 1.
3002	Turn off AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 2.
...	
3999	Turn off AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 999.

MAXAPL

Default = 1000

MAXAPL specifies the maximum number of applied loads to be used as a basis for residual vector calculations. If the number of applied loads exceeds MAXAPL, User Warning Message 9166 is issued, and residual vector augmentation is disabled. Note that the computing requirements for a large number of residual vectors can be prohibitive. See also, the RESVEC Case Control command.

MAXDAMP

Default = 1000

See [BEIGRED](#).

MAXIREVV

Default = 1, SOL 600 only.

Determines whether to reverse axisymmetric coordinates ($x \rightarrow y$ $y \rightarrow x$) and reorder nodes in the elements or not.

- 1 Do not reverse coordinates and renumber elements
- 1 Reverse coordinates and renumber grid ID's for each element. (Default)

Remark:

Experience has demonstrated that certain models require the -1 option and others require the +1 option. The majority require the +1 option. If one of the options produces inside out messages in the first increment or other error messages, switch to the other option.

MAXLP

Default = 5

MAXLP specifies the maximum number of iterations for element relaxation and material point subincrement processes in SOLs 129 and 159. MAXLP is 10 in SOLs 106 and 153 and cannot be changed by the user.

MAXRATIO

Default = 1.E7

The ratios of terms on the diagonal of the stiffness matrix to the corresponding terms on the diagonal of the triangular factor are computed. If, for any row, this ratio is greater than MAXRATIO, the matrix will be considered to be nearly singular (having mechanisms). If any diagonal terms of the factor are negative, the stiffness matrix is considered implausible (non-positive definite). The ratios greater than MAXRATIO and less than zero and their associated external grid identities will be printed out. The program will then take appropriate action as directed by the parameter BAYOUT.

By default, in the superelement solution sequences the program will terminate processing for that superelement. A negative value for BAYOUT directs the program to continue processing the superelement. Although forcing the program to continue with near-singularities is a useful modeling checkout technique, it may lead to solutions of poor quality or fatal messages later in the run. It is recommended that the default values be used for production runs. A related parameter is ERROR.

The value -1 of BAYOUT causes the program to continue processing with near singularities and a zero value will cause the program to exit if near singularities are detected.

In SOLs 101 through 200 when PARAM,CHECKOUT,YES is specified, PARAM,MAXRATIO sets the tolerance for detecting multipoint constraint equations with poor linear independence. (See [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.)

BAYOUT and MAXRATIO may be specified in the Case Control Section in order to provide different values between superelements but not between different boundary conditions.

MBENDCAP

Default = 1, SOL 600 only.

Determines how PBEND internal pressure will be treated.

- 1 Stress stiffening and axial loading due to pressure is ignored
- 0 Stress stiffening and axial loading occurs (this usually happens with end caps)
- 1 Only stress stiffening occurs. (Default)
- 2 Stress stiffening and axial loading occurs for CBEND elements with any nodes having SPC or SPC1 and stress stiffening occurs for all other CBEND elements regardless whether the SPC/SPC1 is referenced above the subcase level or within a subcase.

MCNLPARM

Default = 1, SOL 600 only.

Determines whether to make all field 5 (KMETHOD) consistent if more than one NLPARM entry occurs in the input stream. If mixed KMETHOD fields are provided for different NLPARM entries, SOL 600 can sometimes encounter difficulties.

- 1 KMETHOD will not be changed to be consistent.
- 1 KMETHOD will be changed to be consistent with that used with the NLPARM entry with the lowest ID. (Default)

Remark:

This parameter may be set in RC files.

MCON2D3D

Integer, Default = -1, SOL 600 only.

Controls whether a mixture of 2D and 3D contact will be allowed in a SOL 600 model. For most models a mixture of 2D and 3D contact will fail but there are a few cases where it will run properly. The default will stop the run should a mixture of 2D and 3D contact as specified in field 3 of the first BCBODY line is found. PARAM,MCONF2D3D,1 will allow the job to continue but the job may fail later in the run.

- 0 Stop job with a FATAL ERRO if a mixture of 2D and 3D contact is found.
- 1 Allow job to continue if a mixture of 2D and 3D contact is found.

MCORDUPD

Default = 0, SOL 600 only.

Determines if the coordinates will be updated if one of the CONTINUE options is specified on the SOL 600 Executive Control statement.

- 0 Original coordinates are used for the continue option. (Default)
- 1 Coordinates are revised to reflect the final position after the nonlinear analysis. These updated coordinates are used in the continue option.

Remark:

If mcordupd=1, parampmarct19,1 is automatically set and jid.marc.t19 will be generated (jid.marc.t16 may also be generated if desired).

MCSSHLC

Default = 1, SOL 600 only.

Case Control parameter that determines whether to check that the thickness direction of CSSHL elements is in the direction pointing from grids 1-2-3-4 to 5-6-7-8 or not.

- 1 Do not make the check.
- 1 Make the check. (Default)
- 2 Make the check only for 6-sided elements (looking like chexa) and ignore for 5-sided elements (looking like cpenta).

Remark:

This parameter must be set in the Case Control above the first subcase.

MCSSHORR

Default = 1, SOL 600 only.

Case Control parameter that sets the orientation of the thickness direction of CEHXA converted to solid shells.

- 1 The smallest CEHXA direction will be determined and the element will be renumbered such that the thickness of the renumbered elements points from grids 1-2-3-4 to 5-6-7-8
- 1 The thickness direction will be in the grid 1 to 5 direction (thickness points from grids 1-2-3-4 to 5-6-7-8). (Default)
- 2 The thickness direction will be in the grid 1 to 2 direction (grid numbering that originally was 1,2,3,4,5,6,7,8 will be changed to 1,5,6,2,4,8,7,3)
- 3 The thickness direction will be in the grid 1 to 4 direction (grid numbering that originally was 1,2,3,4,5,6,7,8 will be changed to 1,4,8,5,2,3,7,6)
- 11 The thickness direction will be along the side closest to the basic X-direction
- 12 The thickness direction will be along the side closest to the basic Y direction
- 13 The thickness direction will be along the side closest to the basic Z direction.

Remarks:

1. This parameter must be set in the Case Control above the first subcase.
2. This entry is only used in conjunction with the CSSHLM entry only.
3. This entry should only be used if all elements converted to solid shells are CHEXA, none should be CPENTA.
4. See the CSSHL entry for a description of grids 1 to 8.
5. For options other than mcsshorr=1 the material properties and material orientation direction must be orientated properly.

MDAREAMD

Default = 1, SOL 600 only.

Option to modify or not modify all DAREA entries which are not associated with any other loads (DAREA entries that supply the actual load).

- 1 Do not modify any DAREA entries.
- 1 Modify all DAREA entries that are not associated with any other load entries and supply the actual loading. (Default)

MDEFSEPP

Integer, Default = see below

This SOL 600 parameter allows an easy way to set the defaults for contact separation control. Since this parameter may be placed in RC files, the user can set default separation controls appropriate to the type of problems normally run and apply the same values to all models without having to enter them in the bulk data for each run. These values are normally entered on the BCPARA entry. The form of MDEFSEPP is as follows:

PARAM,MDEFSEPP,AABBBBCD

where AABBBBCD is a packed integer field comprised of the following 4 controls:

	Variable	Standard Default	Values that can be Entered	Max Digits	Min Digits
AA	NODSEP	5	0 to 99	2	0
BBBB	MAXSEP	9999	0 to 9999	4	4
C	ICSEP	0	0 to 3	1	1
D	IBSEP	0	0 to 4	1	1

Remarks:

1. All values of AA, BBBB, C, D override the standard defaults.
2. NODSEP, MAXSEP, ICSEP, IBSEP entered on BCPARA override values on this parameter.
3. This parameter can be set in RC files.
4. Max digits is the maximum number of digits that can be entered.
Min digits is the minimum number of digits that must be entered if this parameter is used.
5. A common setting for moderately difficult separation problems is MDEFSEPP=5001004

AA=NODSEP=5

BBBB=MAXSEP=10

C=ICSEP=0

D=IBSEP=4

6. A common setting for more difficult separation problems is MDEFSEPP=2999934

AA=NODSEP=5
BBBB=MAXSEP=9999
C=ICSEP=3
D=IBSEP=4

MDK4OPT

Default = 1

Performance parameter used in MDACMS for the calculation of K4. If MDK4OPT>0 the most common value of GE for structure elements will be internally applied to PARAM,G to speed up the calculation of K4. Set MDK4OPT to 0 to disable this feature. The traditional K4 calculation will be used instead.

MDOPT14

Default = NO

Option 14 of the MDENZO functional module modifies degree of freedom (DOF) based domain decomposition such that DOF belonging to any one grid point are retained in a single domain. This action (PARAM,MDOPT14,YES) lends stability to the resulting solution at the possible expense of some efficiency. Setting MDOPT14 to NO bypasses this operation so that DOF may be distributed to any domain without restriction. This parameter applies to Matrix Domain ACMS.

MDOTM

Default = AUTO

The default selects the most efficient method for data recovery in DOMAINsolver ACMS (PARTOPT=DOF). There are two methods: (1) Output transformation matrix (OTM) method is selected if the number of modes multiplied by PARAM,MDOTMFAC (Default = 20) exceeds the number of degrees-of-freedom at which displacements are required in order to complete data recovery (DISP, STRESS, etc.) This method also requires PARAM,SPARSEPH,YES which is the default. This method may be forced with PARAM,MDOTM,YES. (2) If the number of modes multiplied by PARAM,MDOTMFAC (Default = 20) does not exceed the number of degrees-of-freedom at which displacements are required in order to complete data recovery condition is not satisfied then the non-OTM method is selected. This method may be forced with PARAM,MDOTM,NO.

MDOTMFAC

Default = 20

See MDOTM.

MDREDOPT

Default = -1

This parameter is used in ACMS for matrix reduction operations. It affects the way in which matrices are reduced to modal coordinates. Quantities affected are damping, loads, and acoustic coupling. By default, the ACMS-generated Q-size matrices are reduced to H-size (modal coordinates). This is the most efficient method. If MDREDOPT=1, a triple matrix multiplication is used.

MDUMLOAD

Default = 0, SOL 600 only.

- 0 For subcases with no applied loads and enforced displacement or velocity of rigid contact bodies, a small magnitude dummy load will be created. Without these dummy loads, Nastran becomes confused and does not produce the correct contact information. If there is only one subcase, the dummy loads are not necessary. (Default)
- 1 Dummy loads will not be produced for any subcase and rigid contact with enforced motion may be incorrectly described.

MECHFIL

Default = 1.E-6

Criterion for discarding massless mechanism modes with small generalized mass. A smaller value will result in more marginal constraint modes being retained. This parameter is used for PARAM,MMMETH,OLD. See also [MECHFIX, 898](#), [MMMETH, 908](#), and [MMFIL, 908](#).

MECHFIX

Default = AUTO

Control for fixing the massless mechanism problem. This capability is provided automatically for the default of this parameter, listed above. This capability is executed only when the eigensolution does not provide answers because of symptoms consistent with the presence of massless mechanisms. If MECHFIX is set to YES, the constraint modes are removed before attempting an eigensolution. When set to NO, the capability is blocked, the eigensolution will fail, and User Fatal Message 9119 will be printed in the F06 file. See also [MMFIL, 908](#) and [MMMETH, 908](#).

MECHPRT

Default = NO

For SOL 103 only, if massless mechanisms are found the constraint modes are printed with a format similar to eigenvectors when this parameter is set to YES. They are labeled CONSTRAINT MODES, and are numbered sequentially. Grid points with only zero values in a mode are not printed. This parameter should be used when performing initial checkout of a model and a goal is to remove all massless mechanisms before starting production analysis. The number of each “mode” matches the corresponding GID,C pair in the high ratio message. If there are many (thousands) of such modes the output file will be large. There is no method to plot these shapes at present.

MESH

Default = NO

If MESH=YES is specified, then a summary of the shading status and the subelement mesh for each CHBDYi element referencing a VIEW Bulk Data entry is printed.

METHCMRS

Default = 0

In dynamic analysis (SOLs 103, 107, 108, 109, 110, 111, 112, 145, 146, and 200), METHCMRS specifies the set identification number of an EIGR or EIGRL entry to be used in the calculation of the normal modes on the v-set of the residual structure.

By default, the residual structure v-set normal modes will be computed based on the METHOD Case Control command selection as long as q-set is present.

MEXTRNOD

Default = 0, SOL 600 only.

Determines whether extra grids will be added to SOL 600 parallel analyses.

- 0 Extra grids will not be added. (Default)
- 1 Extra grids will be added so that all grids from 1 to the highest grid are defined this was necessary for certain version of Marc prior to the 2005 version. All extra grids that are added have coordinates of 0.0 in all three directions.

MEXTSEE,N

Default = 0, SOL 600 only, Case Control Parameter

Determines whether SOL 600 external superelement residual runs contain loads in the main input file or not.

- 0 External Superelement residual runs will not contain any loads. (Default)
- 1 External Superelement residual runs will contain all loads.

MFASTCMP

Default = 1, SOL 600 only.

Determines default composite shell integration method.

- 1 Standard integration method is used. Failure, plasticity, thermal loads, etc. are all allowed. (Default)
- 2 A “fast integration” through the thickness technique is used which ignores thermal strains, plasticity, and temperature-dependent material properties (which should not be entered in the model).
- 3 A “fast integration” through the thickness technique is used which only ignores plasticity. Thermal strains and temperature-dependent materials are allowed.

Remarks:

1. To override the default integration method, use Bulk Data entry PCOMPF.

2. If option 2 is used and temperature loading is present in the model, the option will automatically be re-set to 3.

MFEA5701

Default = 1, SOL 600 only.

Determines whether feature 5701 is written to the Marc file. When this feature is set to 1, it disables default rotation checking set to 0.001, which was initially set for early versions of Marc when RBE2 and similar elements were added because the 0.001 value proved to be responsible for convergence problems. For models that need rotation checking, you can enter the value using the NLSTRAT variable RLROTT. We recommend always using MFEA5701,1 to turn off the default 0.001 checking value in Marc whether or not the value is entered using NLSTRAT RLROTT.

- 0 Rigid rotation checking of 0.001 is turned on.
- 1 Rigid rotation checking of 0.001 is turned off. (Default)

MFORCOR1

Default = 1, SOL 600 only.

Option to correct forces entered twice (at the same node) in multiple subcases. This is not commonly found in the input files.

- 0 Do not correct the forces.
- 1 Correct the forces. (Default)

Remark:

This parameter can be set in RC files.

MFORDUPE

Default = 0, SOL 600 only.

Controls how duplicate forces encountered for the same load case are handled in SOL 600.

- 0 Forces will be translated to Marc as encountered even if duplicates are present. In the case of duplicates (more than one set of forces for the same node in the same subcase), all forces found will be translated directly to the Marc input. (Default)
- 1 If duplicates exist, only the last set of forces will be translated.

Note: To prevent confusion, it is suggested that duplicate forces not be used.

MFSKIPPP

Integer, Default = 0, SOL 600 only.

Controls memory near the end of SOL 600 translations will be freed or not.

- 0 Memory is freed.
- 1 Memory is not freed (can save computer time, and, in some cases may be necessary for some models.)

MGAPINIT

Integer, Default = 0, SOL 600 only.

Controls whether CGAP elements are initially open or closed (during Marc increment zero).

- 0 Gaps are initially open.
- 1 Gaps are initially closed.

MGLUETOL

Real, Default = 1.0E-6, SOL 600 only

Determines the tolerance for the PERMGLUP option of the SOL 600 Executive Control statement for situation where the primary method (same as SOL 101 with permanent glue) fails to find any glued contact possibilities (for example, for edge contact will usually find no glued contact using SOL 101). If not glued contact is found using the primary method a secondary method, which can sometimes take a significant amount of computer time for large models is used. The second method evaluates all grids that have the same coordinates as the grids within the tolerance specified by this parameter. They will be tied together unless they have SPC's, MPC's or are grids attached to rigid elements.

MHEATSHL

Default = 0, SOL 600 only.

Determines whether a membrane or thick shell element formulation is used for heat transfer. This parameter can be overridden by individual PSHELL entries. In the current release, the membrane elements should not be used for thermal contact analysis.

- 1 All "quad" elements in the model will have membrane capability regardless of MIDi values on the PSHELL entries.
- 0 All "quad" elements in the model will have membrane capability. (Default)
- 1 All "quad" elements in the model will have thick shell capability (shell sect, 1).
- 2 Shells use 2 dof per node, linear variation of temperature through thickness (shell sect, 2).
- 3 Shells use 3 dof per node, quadratic variation of temperature through thickness (shell sect, 3).
- >3 Shells use $2*n+1$ dof per node (where ni is the value of MHEATSHL specified). (shell sect, $2*(n-2)+1$)

Notes:

1. Membrane capability in heat transfer means that the temperature is constant throughout the thickness. The MHEATSHL=0 option can be overridden by entering a non-blank value for MID2, MID3 and/or MID4 on an applicable PSHELL entry in which case the MHEATSHL=1 option will be used if no MHEATSHL parameter is entered.
2. Post options will be added to be the same as the value for Marc's shell sect. For example, if shell sect, 3 is generated, post codes 9,1 9,2 and 9,3 will also be generated.

MHEATUNT

Default = 2, SOL 600 only.

Specifies the units for heat transfer using SOL 600.

- 0 SI mm units used
- 1 SI m units used
- 2 US units used. (Default)

Note: This parameter is used by Marc's ISOTROPIC (heat transfer) third datablock, fourth field.

MHEMIPIX

Default = 500, SOL 600 only.

Controls the number of pixels used in radiation heat transfer for SOL 600 using the hemi-cube method.

MHOU BOLT

Default = 0, SOL 600 only.

- 0 SOL 600 transient dynamics will use the single step Houbolt numerical integration method. (Default)
- 1 SOL 600 transient dynamics will use the Newmark Beta numerical integration method.
- 2 SOL 600 transient dynamics will use the standard Houbolt numerical integration method.
- 7 SOL 600 transient dynamics will use the generalized alpha (Hilber-Hughes Taylor) numerical integration method.

MHRED

Default = NO

MHRED=YES, selects the c-set and r-set component mode reduction method suggested by Dr. Arya Majed and Ed Henkel. See the *MSC Nastran V2004 Release Guide* for details.

MICRO

Default = 10, SOL 700 only

Defines the accuracy of the initial conditions in Eulerian elements, when using the geometrical shape definition.

Format:

PARAM,MICRO,VALUE

Example:

PARAM,MICRO,15

VALUE Micro-zoning parameter. (Integer > 0)

Remarks:

1. MICRO is the number of micro zones into which an element is subdivided during initial condition generation.
2. The default MICRO = 10 results in material fractions as accurate as 0.001. If a higher accuracy is required, a greater value for MICRO can be used, but the CPU time for the generation increases rapidly.
3. Micro zoning is only used when the initial conditions of the Eulerian material are specified on a TICEUL1 entry.

MINCLDSB

Integer, Default = 0, SOL 600 only

Determines whether the SB value in field 4 of MATF will be included or not in the stiffness formulation of the model (Marc COMPOSITE datablock 3 field 4):

- 0 SB will not be included.
1 SB will be included.

Remarks:

1. For releases prior to MD Nastran 2010, SB was not included.
2. If SB is included, Marc parameter TSHEAR will also be included.

MINIGOA

Default = NO

Allows for the reduction in the amount of disk space used when using superelements. When this parameter is set to YES, the transformation matrix GOA will contain terms only for the degrees-of-freedom in the U5 (USET, USET1, SEUSET, SEUSET1) set. This can allow for a significant reduction in the amount of disk

space used by the database. The limitation of using this approach is that data recovery will be available only for these degrees-of-freedom and elements connected to them.

MINRECCC, N

Default = 0, SOL 600 only.

- N Integer. Minimum number of iterations per load step. This is the same as MINREC on the NLSTRAT entry. If no other NLSTRAT values are entered, it is easier to enter this parameter. The value can range from 0 to 9. For certain problems, the value should be 2 or greater or accuracy will be poor.

Minimum number of iterations per load step. This is the same as MINREC on the NLSTRAT entry. If no other NLSTRAT values are entered, it is easier to enter this parameter. The value can range from 0 to 9. For certain problems, the value should be 2 or greater or accuracy will be poor.

MINSOUTT

Default = 0, SOL 600 only

Determines elements that deform so much that they go inside-out in an analysis will be deactivated.

- 0 Issue an error message and terminate the job if an element goes inside-out. (Default)
1 Deactivate any element that goes inside-out and continue the analysis. This option is particularly useful for models which allow failure (for example those using MATF).

Remark:

param,minsoutt,1 maps to Marc's parameter IO-DEACT.

MINVASHF

Default = 1.0, SOL 600 only

Inverse Power "auto shift" value. A new shift point (in frequency squared) is determined as the highest frequency squared plus this entry times the difference between the highest and next highest distinct frequency squared.

MINVCITR

Default = 40, SOL 600 only

Inverse Power method, number of iterations.

MINVCSHF

Default = 0.0, SOL 600 only

Inverse Power shift frequency in Hz.)

MINVCTOL

Default = 1.0E-5, SOL 600 only

Inverse Power convergence tolerance.

MINVFMAX

no Default, use MINVNMOD, SOL 600 only

Real. Inverse Power max frequency to extract in Hz.

MINVNMOD

Default = 5, SOL 600 only

Integer. Inverse Power max number of modes to extract.

MLDALLOW

Integer, Default = 0

Determines whether forces (FORCE, MOMENT, PLOAD4, etc.) are allowed in SOL 600 if their ID is the same as the value N of any Case Control LOAD=N entry.

- | | |
|---|--|
| 0 | Forces with the same ID are not allowed. |
| 1 | Forces with the same ID are allowed. |

Remarks:

1. MLDALLOW=0 is the only option available for most other Nastran solution sequences.
2. MLDALLOW=1 was the only option available in SOL 600 prior to MD Nastran 2010.
3. MLDALLOW=1 cannot be used with PARAM,MNASTLDS if MNASTLDS>0.
4. This parameter can be set in RC files.

MLSTRAIN

Default = -1, SOL 600 only.

Corresponds to Marc's "LARGE STRAIN" parameter and optionally allows large rotation of beams and shells.

- | | |
|----|--|
| -1 | Small strain, small rotation analysis is used. (Default) |
| 1 | Large strain and large rotation analysis of beams and shells is implemented in a way to automatically select the best options for a large strain analysis based upon the element type (see the following table). |
| 2 | Use large strain and large rotation analyses of beams and shells for hyperelasticity and multiplicative plasticity with radial return mapping |
| 11 | Same as option 1 except beams and shells are limited to small rotations (see Remark 1.) |
| 12 | Same as option 2 except beams and shells are limited to small rotations (see Remark 1.) |

Element Type/ Material Model	1-Dimensional	Plane Stress or Membranes or Shell Elements	Plane Strain or Axisymmetric, or 3- Dimensional Displacement Form	Plane Strain or Axisymmetric, or 3- Dimensional Hermann Form
Conventional elastic-plastic	Updated Lagrange additive plasticity; no finite strain	Updated Lagrange additive plasticity; includes finite strain	Updated Lagrange additive plasticity; includes finite strain utilized constant strain	Updated Lagrange multiplicative plasticity; includes finite strain
Mooney, Ogden, Gent, or Arruda- Boyce	Total Lagrange	Total Lagrange	Updated Lagrange	Updated Lagrange
Foam	Total Lagrange	Total Lagrange	Updated Lagrange	Updated Lagrange; incompressibility neglected

Remarks:

- Starting with MD Nastran 2010, if MLSTRAIN=1 or 2, feature,2103 will also be added to provide the best large rotation solution of beam and shell models using updated Lagrange. To prevent feature,2103 from being used MLSTRAIN can be set to 11 or 12 instead of 1 or 2 respectively if large strain small rotation is desired.
- Starting with MD Nastran 2010, if hyperelastic materials are found in the model, PARAM,MLSTRAIN,12 will be set automatically unless a different value of MLSTRAIN has been set by the user.

MLSTRAN2

Integer, Default = 0, SOL 600 only

Determines how the LARGE STRAIN option is written in the Marc input file for SOL 600.

- | | |
|---|---|
| 0 | LARGE STRAIN, (other options if applicable) |
| 1 | LARGE STRA, (other options if applicable) |

Remark:

Since SOL 600 writes the Marc input file using large-field free-format input either option will work properly. If the input file is modified by hand and this entry is changed to fixed-format, the LARGE STRA option must be used.

MMAT2ANI

Default = 2, SOL 600 only.

Determines how MAT2 will be mapped to Marc.

- | | |
|---|---|
| 0 | MAT2 will be mapped to Marc's ORTHOTROPIC option. |
|---|---|

- 1 MAT2 will be mapped to Marc's ANISOTROPIC option.
- 2 MAT2 will be mapped as explained in the following note. (Default)

Note: The default, MMAT2ANI=2, maps MAT2 to Marc's ORTHOTROPIC option if G13 and G23 are both zero or blank and to Marc's ANISOTROPIC option if G13 and/or G23 are non-zero.

MMBOLTUS

Integer, Default = -1, SOL 600 only

Controls how the top and bottom nodes are placed in the Marc "tying 69" input when MBOLTUS is used in a SOL 600 model.

- 1 Same as pre-MD Nastran 2010 versions tying 69 node order is top, bottom, control
- 1 Reversed - tying 69 node order is bottom, top control

Remarks:

1. If it appears that a bolt preload may be backwards for bolts modeled using MBOLTUS, this parameter may be entered to reverse the mpc equations internally generated in Marc by tying 69 instead of reversing the loads.
2. For the MBOLTUS entry to be consistent with the SOL 400 BOLT entry PARAM,MMBOLTUS,1 is usually required.

MMEMDETT

Default = -1, SOL 600 only.

Determines the Nastran portion of SOL 600 determines how much usable memory is available just before spawning Marc and informs Marc through the -ml flag. If Marc needs more memory than this value, it will go out of core.

- 1 Do not determine the amount of usable memory. (Default)
- 1 Determine the amount of usable memory and set the -ml Marc command line option to that value

Remarks:

1. Marc can determine this memory, however the computations determine the amount of installed memory and do not take into account memory used by the operating system or by other user processes.
2. This parameter may be entered in RC files.

MMFIL

Default = 1.E-10

Filter value used to distinguish between massless mechanism modes and rigid body modes. A smaller value may discard rigid body modes. The default value has been effective on all problems solved to date. This parameter is used for PARAM,MMMETH,NEW and PARAM,MMMETH,BETA. See also [MMMETH, 908](#).

MMMETH

Default = NEW

Selects method for searching for massless mechanisms. The default value is generally the most reliable method. Use PARAM,MMMETH,BETA to invoke a new procedure that can sometimes find massless mechanisms when the default does not find any. Specify PARAM,MMMETH,OLD to use the original procedure. See also [MECHFIX, 898](#).

MNASTLDS

Integer, Default = See Remark 1., SOL 600 only.

Option to determine complex forces using OLOADS datablock or the foes, moments, pressures, etc. in the Nastran input file. Positive values of MNASTLDS may also significantly speed up the SOL 600 translation and, in the case of RFORCE, the marc portion of the run will also usually be faster.

- | | |
|---------|--|
| -1 or 0 | Loads for Marc are translated or calculated directly from the forces, moments, pressures, etc. in the Nastran input file. (Remark 1.) |
| 1 | Complex loads and moments are obtained from the OLOADS datablock, pressures, rforce and other distributed loads are translated from the values found in the Nastran input. (Remark 3.) |
| 4 | Same as MNASTLDS=1 if the model has Bulk Data LOAD entries with more than one term (L2, L3, ... on the Bulk Data LOAD entry is defined), otherwise same as mnastlds=0. |
| 99 | Same as MNASTLDS=1 regardless of whether complex loads or Bulk Data LOAD entries with more than one term exist in the mode. |
| 777 | Same as MNASTLDS=99 except that all type of distributed loads (pressures, rforce, etc.) will be converted to force and obtain from the OLOADS datablock. (Remark 5.) |

Remarks:

1. Default=99 if any GRID entry has field 7 defined. Default=777 if there are multiple subcases and RFORCE or PLOAD4 with continuation lines exist in the model or if any FORCE entry with nonzero (or blank) field 4 is defined, otherwise the default is 4.
2. MNASTLDS=-1 (or 0) was the only option prior to MD Nastran 2010.
3. Complex loads are defined as those forces or moments whose grid appears more than once when the Case Control and Bulk Data LOAD entries are evaluated. For example consider the following situation:

```
....  
SUBCASE 1  
LOAD = 10  
SUBCASE 2  
LOAD = 20  
BEGIN BULK  
LOAD, 10, 1.0, 2.0, 1, 3.0,2  
LOAD, 20, 1.0, 4.0, 1, 5.0,2  
FORCE,1,51, ,1.0,0.0,0.0,100.0  
FORCE,2,51, ,1.0,0.0,0.0,200.0
```

Although the force at grid 51 occurs for different load ID's when combined using the LOAD entries it actually occurs twice in each subcase. This is the simplest form of a complex load and can be handled by either MNASTLDS option. However, some models will have much more complex situation, even having different entries for the Fx, Fy and Fz directions. In such circumstances, particularly if multiple subcases are involved MNASTLDS > 0 is recommended.

4. This parameter may be set in RC files.
5. When MNASTLDS=777 pressures (and other forms of distributed loads) are converted to nodal forces and the follower force effects will be lost. However, for multiple subcases the follower force effects are not available in SOL 600 unless the loading is the same between subcases except for a scale factor.

MODACC

Default = -1

- | | |
|----------|--|
| 0 | Selects the mode acceleration method for data recovery in dynamic analysis. See Formulation of Dynamic Equations in SubDMAP GMA in the <i>MSC Nastran Reference Guide</i> for further discussion. If PARAM,SPARSEDR,NO is specified, then PARAM,DDRMM,-1 must also be specified. |
| 1 | Is the same as MODACC = 0 except if SUPPORTi entries are present then the displacements are corrected for rigid body motion and the local flexibility at the SUPPORTi degrees-of-freedom. |
| ≥ 0 | Is not recommended for use in hydroelastic problems. |

MODEL

Default = 0

This parameter also allows several models to be stored in the same graphics database created by PARAM,POST,0.

MOFFCORE

Default = 1, SOL 600 only.

Determines how memory for PARAM,MARCOFFT above is to be allocated (for increased speed).

- 1 Additional memory is not allocated.
- 1 Additional memory is allocated if available. (Default)

MOP2TITL

Default = 1, SOL 600 only.

Determines how titles are placed on the 146 word record for op2 output records generated by SOL 600.

- 1 Standard titles will not be placed, however titles of the form "CQUAD4 STRESS FROM MARC", and similar titles for other element types, strains, displacements, etc. will be written. This option is useful for certain postprocessors.
- 1 Standard Nastran title, subtitles will be placed on op2 files generated by SOL 600. This option is useful for postprocessors that require SOL 600 op2 data to be exactly in the same format as that generated by other Nastran solution sequences. (Default)

Note: The SOL 600 op2 file follows that of SOL 109 as closely as possible.

MOUTRCOR

Integer, Default = 0, SOL 600 only

Determines whether SOL 600 OUTR options are disabled or not if any CORD2i references another coordinate system.

- 0 Disable OUTR options
- 1 Do not disable OUTR options

Remarks:

1. In some cases, OUTR results are inaccurate if a CORD2i references another CORD.
2. CORD2i entries are not checked to see if they are actually used in the model.

MP1SET

Character "F" or "G" (default, see below)

Controls the integration set used in the calculation of MONPNT1 results. For the value of "F", the integrated values will reflect any redistribution of loads caused by rigid elements and MPCs. This is particularly useful in the static aeroelastic solution sequences (SOL 144) when RBE3s are used to distribute mass, applied loads and splining. For the value of "G", no redistribution occurs. The default is a function of the solution sequence.

SOL 144 - Default = "F", option "G" is not available.

All other sequences - Default = "G", option "F" can be specified by the user.

MPCX

Default = 0

See OLDSEQ.

MPERMPRT

SOL 600 only.

Determines whether to print permanent glue MPC's in the f06 file.

0 Do not print permanent glue MPC's

1 Print permanent glue MPC's

MPTUNIT

Default=0

MPTUNIT is used in conjunction with an FMS ASSIGN statement to specify the unit number for the storage of monitor points results in OUTPUT2 format in SOL 146. Default=0 indicates that no monitor point data are to be stored

MQUATERN

Integer, Default = 2, SOL 600 only

Controls whether quaternions will be used for SOL 600 models with large rotation.

-1 Do not use quaternion "feature" for large rotation

1 Use quaternion feature,2103 which allows large rotations for total Lagrange or updated Lagrange

2 Use quaternion feature,2105 which allows large rotations only if updated Lagrange is used

Remarks:

1. The use of MQUATERN = -1 allows backwards compatibility such that most models will obtain the same results using MD Nastran R3 as were obtained using MD Nastran R3.
2. The use of MQUATERN = 1 is believed to yield the most accurate large rotation results for most models.
3. With PARAM,MQUATERN,1 large rotation may be used both with updated Lagrange and total Lagrange.
4. mquatern = -1 does not set Marc feature 2103 (feature 2105 is not set).
5. mquatern = 1 sets Marc feature 2103

MRADUNIT

SOL 600 only.

Controls the units used in radiation heat transfer for SOL 600.

- 1 Degrees Celsius
- 2 Degrees Kelvin (default if parameter not entered)
- 3 Degrees Fahrenheit

Note: Degrees Rankin are not available.

MRAFFLOR, N

Default = N0, SOL 600 only.

- N Integer. If N=0, a new AF_flowmat file containing temperature-dependent stress-strain curves will be generated during the current Nastran execution and also used in the spawned Marc run. If N=1, an existing AF_flowmat file will be used. The name of the file is always determined by the value of PARAM,MRAFFLOW, but PARAM,MRAFFLOT determines if other characters are added.

MRAFFLOT, N

Default = N0, SOL 600 only.

- N Integer. If N=0, the file name as specified using PARAM,MRAFFLOW,Name will be used with no changes except that all characters will be in lower case and the extension “.mat” will be added. If N=1, the characters “asm_” will be added at the beginning of Name, the first character of Name will be upper case (the other characters of Name will be lower case) and the extension “.mat” will be added. This will make the AF_flowmat file name compatible with many names in Marc’s AF_flowmat directory.

MRAFFLOW, Name

No Default, SOL 600 only.

- Name Character. Name of a file containing temperature dependent stress versus plastic strain curves in Marc’s AF_flowmat format. This file can be generated from the current Nastran run using TABLEST and TABLES1 entries or a pre-existing file can be used depending on the value of PARAM,MRAFFLOR. The extension “.mat” will be added to Name. If this is a new file, it will be saved in the directory from which the Nastran execution is submitted. If a pre-existing file is to be used, it can either be located in the directory where the Nastran execution is submitted or in the Marc AF_flowmat directory.

MRALIAS ID (MALIAS02, MALIAS03, etc.)

No Default, SOL 600 only. This parameter is not usually used.

The purpose of the parameter is to map the advance nonlinear element type selected by the internal Marc translator in Nastran to a different type. For example, if the element type 75 for CQUAD4 is normally used, a mapping to advance nonlinear element type 139 could be made. ID is a 6 digit number. The left 3 digits are the element type normally selected by the translator and the right 3 digits are the element type to be mapped. In the above example, element type 75 is to be mapped to 139. The user would enter ID=075139. If element type 165 is to be mapped to element type 1 (which is not a real case), ID=165001. Consult Marc Volume B for a list of elements and their meaning. The user is responsible for ensuring that the mapping selected is proper. There is a limit of 18 aliases that may be entered in any model. Since Nastran can only accept one parameter with a given name, the second alias should be named PARAM,MALIAS02 and the third PARAM,MALIAS03, etc. All original element types mapped must actually exist in the model.

Remark:

These entries should only be used if the Marc GEOMETRY entries are identical for the original and new element types.

Note: If you use one of the MRALIAS parameters, certain “parameters” in the Marc file may no longer be correct. For example, an element originally capable of using the updated Lagrange method may be aliased to one that must use the total Lagrange method. Such conditions are not checked by the translator when you use alias and you will need to make modifications to the Marc input file yourself to reflect them. To resolve this use PARAM,MLSTRAIN.

The Bulk Data entry, ALIASM, is available and subsequent version are more powerful.

MRALLOCG

No Default, SOL 600 only.

Integer. The value entered here is the amount of memory (MB) allocated for general Marc memory when Marc is spawned from Nastran. It specifies the initial allocation of “general memory”. This is used for storing element stiffness matrices and part or all of the matrix solver workspace among other things. Please note that element data like stresses and strains are not part of the general memory. Solvers 6, 8, and 9 use the main part of the workspace in separate memory. Initial allocation of the general memory can be used for avoiding reallocation (increase of the workspace). For parallel processing the amount specified is the total for the job. It is divided by the number of domains used.

MRALLOCS

No Default, SOL 600 only.

Integer. The value entered here is the amount of memory (MB) allocated for Marc solver memory when Marc is spawned from Nastran. It specifies the initial allocation of memory for solver 8. By giving a value that is more than the maximum used during the run, one avoids that the solver workspace is increased (reallocated). This can be particularly useful for large contact jobs, where additional memory may be allocated due to

contact. If the given workspace is less than what is needed, it is automatically increased. This option is only for use with solver type 8. No check is done to see if solver type 8 is used in the job. For parallel processing the amount specified is the total for the job. It is divided by the number of domains used.

MRBE3SNG

SOL 600 only.

Option to check the singularity of RBE3's in SOL 600. The real value entered is the singularity threshold allowed. If MRBE3SNG is entered as a positive value, all RBE3's with poor singularity values above the value entered will be output in the jid.marc.out file as warning messages. If MRBE3SNG is entered as a negative value, all RBE3's with poor singularity values above the absolute value entered will be output in the jid.marc.out file as error messages and the job will abort if they are found in increment zero, but if they are found after increment zero the messages are warning messages and the job will continue.

Note: A "good" RBE3 element using the original geometry in increment zero could become singular as the structure deforms in subsequent increments.

MRBEAMB

Default = 0, SOL 600 only.

- 0 Write equivalent radius for all beams (see PARAM,BEAMBEA) whether beam-beam contact is anticipated or not. The equivalent radius is the 7th field of Marc's GEOMETRY values for beam type elements. (Default)
- 1 Do not write equivalent radius (7th field is blank). This might be necessary for versions of Marc earlier than 2003.

MRBEPARM, IJK

Default = 0, SOL 600 only.

IJK provides settings for Marc's RBE parameter. If PARAM,MRBEPARM parameter is entered, Marc's RBE parameter will be set using this IJK. If PARAM<MRBEAMPM is not entered, PARAM,MARCRBE2 or PARAM,MARCRBE3 can be used to set Marc's RBE parameter.

IJK is a combination of three variables. For example 311. Descriptions for individual entries are as follows:

- I Enter 3 or 6 to control the number of degrees-of-freedom. For the dependent grid (reference grid) of each rbe2 or rbe3. The independent grids can have 3 or 6 dof and can be different than what is specified by I.
- J Enter 1 to use large displacement formulation of rbe2. Enter 3 to deactivate automatic convergence test for rbe2.
- K Enter 1 to use large displacement formulation of rbe3. Enter 2 to activate non-normalized rotation constraint coefficient for rbe3.

Note: If IJK values other than specified above are entered, IJK will be set to zero and the parameter will not be used. This parameter should not be entered unless there are rbe2's or rbe3's in the model and they are to be used as such in Marc (rather than mpc or stiff beams).

MRBDYCVT

Default = 0, SOL 600 only.

Determines if CHBDYG is converted to CHBDYE for SOL 600 heat transfer.

- 0 Do not convert. (Default)
- 1 Attempt to convert CHBDYG to CHBDYE. All grids specified by all CHBDYG entries must reference actual conduction elements (CQUAD4, CHEXE, CTETRA, etc.) If even one CHBDYG does not reference an existing conduction element, this parameter will be re-set to zero.

Note: If MRBDYCVT=0, CHBDYG will normally result in point heat transfer loads rather than distributed heat transfer loads.

MRBIGMEM

Integer, Default = 0, SOL 600 only.

- N If N=0, memory allocations during loads translation phases are sized for computers with limited memory and swap space (paging space). Some large problems and/or unusual problems may not run. If this happens, use a newer modern computer with lots of memory and disk space (and lots of swap space) and set N=1. Larger memory allocations will then be available. This parameter is not usually required unless the available memory is extremely small.

MRBUKMTH

Default = 2, SOL 600 only

- 1 Buckling modes will be computed using the Inverse Power method.
- 2 Buckling modes will be computed using the Lanczos method. Matrices must be positive-definite for this option. (Default)
- 3 Use Lanczos if EIGRL is specified, Inverse Power if EIGB is specified.

Note: MRBUKMTH should be specified if EIGR or EIGRL is used for buckling unless SOL 600,105 is the solution sequence. An alternative is to use ANALYSIS=BUCK. In SOL 600, it is not possible to compute natural frequencies and buckling modes in the same run. If ANALYSIS=BUCK is specified anywhere in the Case Control and if PARAM,MRBUKMTH is omitted, param,mrbukmth=3 will be set automatically.

MRC2DADD

Default = 0, SOL 600 only.

Allows an offset to be added to all coordinates for 2D analyses so that X and Y will always be positive.

- 0 Offsets will not be added. (Default)
- 1 Offsets will be determined so that all Marc X and Y coordinates are positive (will exceed 0.1).

MRCKBODY

Integer, Default = 1, SOL 600 only.

Determines whether each BCBODY will be checked to see if the 2nd line is present when more than 2 lines exist. If the 2nd line is missing for BCBODY entries with more than 2 lines, errors can occur.

- 1 BCBODY entries are not checked.
- 1 BCBODY entries are checked. If the 2nd line is not present for BCBODY entries having more than 2 lines, a 2nd line will be added internally.

Remark:

This parameter can be set in RC files.

MRCKLOAD

Integer, Default = 0, SOL 600 only.

Controls whether each subcase has LOAD or DLOAD specified. SOL 600 will usually not run correctly unless each subcase has some type of load (dummy or real). If a subcase only has a change in SPC, MPC, or contact to obtain the correct results a dummy load is usually required.

- 0 No check is made to see if each subcase has a Case Control LOAD or DLOAD entry.
- 1 A check is made to see if each subcase has a Case Control LOAD or DLOAD entry and if any subcase does not have such entry, a fatal error will be issued.

Remarks:

1. This parameter must be entered in the Case Control above any subcases.
2. This parameter may be set in RC files.

MRCNADD

Integer, Default = 1, SOL 600 only.

Controls whether PARAM,MRCNVER,11 will be added automatically if any BCTABLE with FBSH, BKGL and/or SEGS sub-headers is found. Also controls whether a BCTABLE with ID=0 will be added if one is not present in the input file.

- 1 Do not change value of PARAM,MRCNVER and do not add a BCTABLE with ID=0 if it is missing.
- 1 Change PARAM,MRCNVER to 11 if any sub-headers FBSH, BKGL and/or SEGS are found. Also add BCTABLE=0 if there are other BCTABLEs and none with ID=0 exists.
- 2 Change PARAM,MRCNVER to 11 if any sub-headers FBSH, BKGL and/or SEGS are found but do not add BCTABLE=0 if there are other BCTABLEs and none with ID=0 exists.
- 3 Do not change PARAM,MRCNVER to 11 if any sub-headers FBSH, BKGL and/or SEGS are found. Add BCTABLE=0 if there are other BCTABLEs and none with ID=0 exists.

Remarks:

1. PARAM,MRCNVER,11 is necessary if BCTABLE sub-headers FBSH, BKGL and/or SEGS exist in the model.
2. BCTABLE with ID=0 is sometimes necessary for certain contact problems. It never hurts to have one as it will be ignored for those contact problems where it is not needed.
3. This parameter may be used in MD Nastran 2010 and subsequent versions. The behavior of previous versions was the same as PARAM,MRCNADD,-1.
4. This parameter may be set in RC files.
5. This parameter, if used, must be placed in the Case Control above any subcases.

MRCNRES

Default = 0, SOL 600 only.

For restart runs, Marc will produce “continuous” results files (t16/t19) which contain the results of the original run(s) as well as the results of the restart run. If Nastran postprocessing is requested to generate op2, xdb, etc. files, they will also contain the results from prior runs as well as the restart run.

MRCNRES=1, output results files will contain the results of the restart run only.

MRCNTAB

Default = 0, SOL 600 only.

Determines whether CONTACT and CONTACT TABLE for SOL 600 use table-driven form or not.

- 0 Do not use table-driven form. (Default)
- 11 Use table-driven form of CONTACT and CONTACT TABLE

Remarks:

1. If MRCONTAB=11 then MRCONVER must also be set to 11
2. This parameter should is not necessary for SOL 600 heat transfer analysis and should not be set

MRCONUSE

Integer, Default = 0, SOL 600 only

Determines which increment for which contact MPC's will be used for SOL 600 jobs using a CONTINUE=101 to CONTINUE=159 (see the SOL 600 Executive Control statement, CONTINUE option).

- 0 The last increment will be used (remark).
N Increment N will be used.

Remark:

When models with contact and if BCPARA<INITCON,4 is specified, MPC's for each increment will be produced and placed in files named jid.marc.conmpc_xxxx where xxxx ranges from 0001 to the number of increments in the solution. This parameter allows the user to specify which increment will be used in the continuation run spawned by the primary Nastran run after Marc finished. N must be less than or equal to the number of actual increments.

MRCONVER

Default = 0, SOL 600 only.

Determines version of Marc to use for CONTACT and CONTACT TABLE for structural analysis.

- 0 Uses enhanced Marc version 9 (Marc 2001 with added fields).
11 Users Marc version 11 (Marc 2005).

Remarks:

1. This parameter only affects contact and only options 0 and 11 are available. It is not needed unless BCTABLE with FBSH and/or BKGL is entered.
2. This parameter may be set in RC files.
3. If PARAM,MRCONVER,11 is entered it is usually also necessary to enter PARAM,MRSRVER,11.

MRCORDS

Default = 0, SOL 600 only.

Determines whether Marc COORD SYSTEM will be added if any CORD1i or CORD2i entries are in the model and if CBUSH elements, orientation vectors or other items requiring coordinate system transformation are present in the model or if PARAM,MRPLOAD4,2 is entered which signifies that PLOAD4 with continuation entries with CID, N1, N2, N3 are to be taken into account in the model.

- 0 Add Marc COORD SYSTEM. (Default)
 1 Do not add MARC COORD SYSTEM

MRCQUAD4

Default = 75, SOL 600 only.

Controls the “Default” advance nonlinear element type for CQUAD4 elements in SOL 600.

- 75 Advance nonlinear element type 75 is used. (Default)
 139 Advance nonlinear element type 139 is used.

Note: Element 75 is capable of thick shell behavior and element type 139 is applicable to thin shells.

MRCTRIA3

Default = 75, SOL 600 only.

Controls the “Default” advance nonlinear element type for CTRIA3 elements in SOL 600.

- 75 Advance nonlinear element type 75 with a duplicate node is used. (Default)
 138 Advance nonlinear element type 138 with 3 nodes is used.

Note: Element 75 is capable of thick shell behavior and compatible with the default element 75 used for CQUAD4, but since one node is degenerate (repeated) stresses may not be accurate. Element 138 is accurate for thin shell behavior but may not be accurate for thick shell behavior. Currently there is no triangle thick shell in Marc.

MRCWANGL

Default = 0.0, SOL 600 only.

Angle in degrees over which to rotate the cross-section about the beam axis to obtain its final orientation.

MRDELTIT

Default = 3, SOL 600 only.

Determines how delta time is set for each “step” of a SOL 600 transient nonlinear analysis.

- 0 DTI=Ti/N
 1 DTI=Ttot/N
 2 DTI=min(Ttot/N, Ti/2.0)

- 3 DTi=min(Ttot/N, Ti/10.0) (Default)
- 4 DTi=min(Ttot/N, Ti/100.0)
- 5 DTi=min(Ti/N, Ti/2.0)
- 6 DTi=min(Ti/N, Ti/10.0)
- 7 DTi=min(Ti/N, Ti/100.0)

1. SOL 600 transient dynamics is run by taking each pair of points on the applicable TABLED1 entry for the applied forcing function as a step. For example, if the following TABLED1 is given, the time steps will be as shown below:

TABLED1, 1
 +, 0.0, 0.0, 1.0E-3, .2 4.0E-3, .3, 6.0e-3, -.3
 +, 1.0E-2, 0.0

Step	Final Time	Delta Step Time (Ti)
1	1.0E-3	1.0E-3
2	4.0E-3	3.0E-3
3	6.0E-3	2.0E-3
4	1.0E-2	4.0E-3

If the entry for this model is (N=100, Ttot=1.0E-2)

DTi is the initial delta time (AUTO STEP (2,1) for the particular step, the six selections of MRDELTTT would give for the DTi values shown in the table below. Note that option 3 gives at least 10 points per step and the old option 0 is probably too conservative.

Step	Final Time	Time Step (Ti)	DTi for MRDELTTT							
			0	1	2	3	4	5	6	7
1	1.0E-3	1.0E-3	1.0E-5	1.0E-4	1.0E-4	1.0E-4	1.0E-5	1.0E-5	1.0E-5	1.0E-5
2	4.0E-3	3.0E-3	3.0E-5	1.0E-4	1.0E-4	1.0E-4	3.0E-5	3.0E-5	3.0E-5	3.0E-5
3	6.0E-3	2.0E-3	2.0E-5	1.0E-4	1.0E-4	1.0E-4	2.0E-5	2.0E-5	2.0E-5	2.0E-5
4	1.0E-2	4.0E-3	4.0E-5	1.0E-4	1.0E-4	1.0E-4	4.0E-5	4.0E-5	4.0E-5	4.0E-5

2. If Dti is set using NLAUTO variable TINIT, this parameter will be ignored.

MRDISCMB

Default = 0 without gravity, =1 with gravity, SOL 600 only.

Determines whether distributed loads, such as pressure, are combined in the Marc input file into one list if the magnitude of the distributed loads are the same. This parameter applies to MSC Nastran 2005 only. For previous MSC Nastran versions, MRDISCMB was 1. Option 0 save Marc memory and processing time.

- 0 Distributed loads with the same magnitude are combined.
- 1 All distributed loads are input individually. Must be used if multiple subcases with the same loadings are present in the model.

Caution: This parameter should be used with caution. For multiple load cases, MRDISCMB=0 may produce the wrong results particularly if gravity loading is present. MRDISCMB=1 must be used if multiple subcases with the same loadings are present in the model.

MRDUPMAT

Default = 1 without gravity loading and -1 with gravity loading, SOL 600 only.

Controls whether to create duplicate materials for elements used by both the Marc pshell (PARAM,MRPSHELL,1 or the smear option on the SOL 600, ID statement) or not. If the user knows in advance that no materials are used by shells and any other types of elements, this step can be skipped to save considerable computer time.

- 1 Check for materials used by shells and other types of elements and create duplicate materials if any are used by both.
- 1 Do not check or create duplicate materials.

MRDYNOLD

Default = 1, SOL 600 only.

Determines whether dynamic loads created by SOL 600 are the same as in MD Nastran R3 (and MSC Nastran 2008) and prior releases or uses a new calculation method.

- 1 Dynamic loads are the same as in MD Nastran R3 (and MSC Nastran 2008) and prior releases.
- 1 Dynamic loads are “total loads” and may be more accurate for multiple subcases. (Default)

MREIGMTH

Default = 1, SOL 600 only.

- 1 Eigenvalue analysis will be done in Marc using the Lanczos method. (Default)
- 0 Eigenvalue analysis will be done in Marc using the inverse power sweep with double eigenvalue extraction.
- 3 Eigenvalue analysis will be done in Marc using the inverse power sweep with single eigenvalue extraction.

MREL1103

Default = 0, SOL 600 only.

- 0 Maps CQUAD4 to Marc's element type 11 for plane strain problems. (Default)
- 3 Maps CQUAD4 to Marc's element type 3 for plane stress problems.

MRELRB

Default = 0, SOL 600 only.

- 0 If BCMOVE with the release option is specified, IDRDBODY (see the BCMOVE entry) will refer to the actual IDs of BCBODY entries. (Default)
- 1 The release option of BCMOVE is used, IDRDBODY (see the BCMOVE entry) will be in the order of occurrence of the BCBODY entries in the sorted Nastran Bulk Data file. For example, if there are two BCBODY entries with ID 12 and 22, the MRELRB=1 option means that you should specify IDRDBODY on the BCMOVE entry using values of 1 and 2. If MRELRB=0, the IDRDBODY values should be 12 and 22.

MRELSPCD

Integer, Default = 0, SOL 600.

Controls whether SPCD for the second or subsequent subcase represents total (absolute) displacements or incremental displacements (relative to the displacement at the end of the previous subcase). SPCD for the first subcase is always total (absolute).

- 0 SPCD represents relative displacements (same as SPCR) for subcases 2 and above unless PARAM,MARCTOTT,1 has been set, in which case the SPCD's represent absolute displacements.
- 1 SPCD represents absolute displacements for subcases 2 and above (relative displacements for subcases 2 and above must be entered using SPCR).

Remarks:

1. This parameter may be set in RC files.
2. PARAM,MRELSPCD,1 will automatically set PARAM,MARCTOTT,1 which means total loads using table-driven input will be used in the analysis. This is the only way total (absolute) displacements for subcases 2 and above can be used in Marc.
3. The behavior of SPCD prior to the MD Nastran 2010 release was to consider SPCD as relative displacements for subcases 2 and above.

MRENUEL

Default = -1 if parameter is not entered and MRENUEL is not entered on the SOL 600 statement, SOL 600 only.

It is best if MRENUEL is specified in the SOL statement. Some models will not have memory allocated properly if this parameter is placed in the bulk data.

- 1 No renumbering will occur (suggested for models with largest element number less than approximately 20000). (Default)
- 1 All elements will be renumbered and the new numbers will be used in the Marc analysis (see Remark 4.)
- 2 All elements will be renumbered, the new numbers are used internally during translation however the original element numbers will be used in the Marc input file and Marc analysis.

Remarks:

1. This parameter can be set in RC files or on the SOL 600 statement.
2. To use models where the maximum element number is greater than 9,999,999 this parameter must be set on the SOL 600 statement (see MRENUEL on the SOL 600 statement).
3. MRENUEL should not be set both on the SOL 600 statement and as a parameter.
4. For MRENUEL=1 an equivalence list of original and re-numbered element numbers is output on file elenum.txt.

MRENUGRD

Default = -1 if parameter is not entered and MRENUGRD is not entered on the SOL 600 statement, SOL 600 only.

It is best if MRENUGRD is specified in the SOL statement. Some models will not have memory allocated properly if this parameter is placed in the bulk data.

- 1 No renumbering will occur (suggested for models with grid IDs less than approximately 20000). (Default)
- 1 All grid IDs will be renumbered and the new numbers will be used in the Marc analysis (see Remark 4.)
- 2 All grid IDs will be renumbered, the new numbers are used internally during translation however the original element numbers will be used in the Marc input file and Marc analysis.

Remarks:

1. This parameter can be set in RC files or on the SOL 600 statement.
2. To use models where the grid ID is greater than 9,999,999 this parameter must be set on the SOL 600 statement (see MRENUGRD on the SOL 600 entry).
3. MRENUGRD should not be set both on the SOL 600 statement and as a parameter.

4. For MRENUMGRD=1 an equivalence list of original and re-numbered grid IDs is output on file gridnum.txt.

MRENUMBR

Default = 0, SOL 600 only.

Determines if both grid and element IDs for SOL 600 will be renumbered or not.

- 0 No renumbering will occur (suggested for models with largest grid ID less than approximately 20000). (Default)
- 1 All grid and element IDs will be renumbered and the new numbers will be used in the Marc analysis. An equivalence list will be output on file grdid.txt
- 2 All grid and element IDs will be renumbered internally during translation, however the original element numbers will be used in the Marc input file and Marc analysis.

Remarks:

1. This parameter can be set in RC files or on the SOL 600 statement.
2. To use models where the grid ID is greater than 9,999,999 this parameter must be set on the SOL 600 entry (see MRENUMBR on the SOL 600 statement).
3. MRENUMBR should not be set both on the SOL 600 statement and as a parameter.
4. For MRENUMBR=1 an equivalence list of original and re-numbered grid IDs is output on file gridnum.txt.

MRESTALL

Default = 1, SOL 600 only.

Controls rotational restraints for solid element-only models.

- 0 SPCs for DOFs 4-6 will be ignored if entered in the Nastran file
- 1 SPCs for DOFs 4-6 will be included if entered in the Nastran file. (Default)

Note: This option might produce an input-data error in the Marc run but is sometimes required if RBEs or other special items are included in the model.

MRESULTS

Default = 0, SOL 600 only.

If MRESULTS is set to 3, postprocessing of a previously-generated Marc t16 file to a results-only op2 file (normally called an f11 file) will be accomplished. OUTR=f11 and STOP=3 should also be set on the SOL 600 command line. This capability is available starting with MSC Nastran 2004.1.0. If MRESULTS=0 the t16 file from the current job will be processed if requested by OUTR options on the SOL 600 statement.

MREVPLST

Default = -1, SOL 600 only.

Determines whether 2D plain stress triangular element node numbers will be reversed or not.

-1 or 0	Do not reverse the node numbers
1	Reverse the node numbers

MRFINITE

No Default, SOL 600 only.

Controls Marc's FINITE parameter. If entered, with an integer value of 1, Marc's FINITE option will be employed. If this option is entered, parameters MRFOLLOW1, MRFOLLOW3 and MARUPDAT should also be entered. Other parameters to be considered are MARCDILT, MARCASUM and LGDISP.

MRFOLLOW

Default = 0, SOL 600 only.

- 0 FORCE1, FORCE2, MOMENT1, MOMENT2 will act as non-follower forces. (Default)
- 1 Follower forces entered using FORCE1, FORCE2, MOMENT1, MOMENT2 will be mapped to Marc's new follower force option.
- 1 Follower forces will be turned off even if requested to be on using other options. This is sometimes necessary for multiple load cases where pressures are applied to different elements in the different load cases.

MRFOLL02

Default = 0, SOL 600 only.

Controls whether Marc's follow for, 2 is used when multiple subcases are present.

- 0 Marc's follow for, 2 will not be used when multiple subcases are present. (Default)
- 1 Marc's follow for, 2 will be used when multiple subcases are present

MRFOLLOW1

No Default, SOL 600 only.

Controls the second field of Marc's follow for, N, M, L parameter (in this case, value N). If entered, options 1, 2, 3 and -1 are currently available. Enter:

- 0 If follower force due to distributed loads (pressure) is not to be considered.
- 1 If follower force stiffness due to distributed loads is not required.
- 2 If follower force stiffness due to distributed loads is to be included.

- 3 If the follower force for distributed loads is based upon the displacement at the beginning of the increment, as opposed to the last iteration.
- 1 If the undeformed geometry is required but total values of distributed loads are to be used (not presently available)

MRFOLLOW3

Default = 0 for statics and 1 for dynamics.

Controls the 3rd field of Marc's follow for, N, M, L parameter (in this case, value M). This parameter is normally set internally by SOL 600 and should not normally be set by users, please see the following remarks.

Remarks:

1. Starting with MD Nastran 2010 and MSC Nastran 2009, MRFOLLOW3 must be 1 for SOL 600 dynamics and if entered as zero for dynamics will be reset to 1. If the user enters dynamic loads using MARCIN they should be in the form of total loads rather than incremental loads.
2. Starting with MD Nastran 2010 and MSC Nastran 2009, MRFOLLOW3 must be 1 for SOL 600 statics if PARAM,MARCTOTL,1 is entered. IF PARAM,MRFOLLOW3 is entered as zero when PARAM,MARCTOTL,1 is entered, it will be reset to 1 if possible.
3. Starting with MD Nastran 2010 and MSC Nastran 2009, MRFOLLOW3 must be 1 for SOL 600 if PARAM,MARCTOTT,1 is entered. If PARAM,MRFOLLOW3 is entered as zero when PARAM,MARCTOTT,1 is entered, it will be reset to 1 if possible.

MRFOLLOW4

SOL 600 only.

Controls the 4th field of Marc's follow for N, M, L parameter (in this case, value L). This parameter is normally set internally by SOL 600 and should not normally be set by users, please see the following remark.

- 0 If follower force for point loads is not required. (Default)
- 1 If follower force for point loads is to be considered.

Remark:

To activate follower forces (concentrated loads) PARAM,MRFOLLOW,1 must be entered as well as the forces using FORCE1, FORCE2, MOMENT1 and/or MOMENT2. PARAM,MRFOLLOW4 should also be set to one.

MRGAPUSE

Default = 0, SOL 600 only.

Determines whether Nastran CGAP elements will be approximated as Marc gap elements. The default is to fatal SOL 600 analyses if CGAP elements are found in the model.

- 0 Do not translate Nastran models using Nastran CGAP elements.
- 1 Translate Nastran models using CGAP elements. Marc gap elements are quite different than Nastran elements and usually can't be translated. In a few limited cases the Nastran and Marc gap elements are equivalent. It is up to the user to determine whether the gap Nastran elements can be used with SOL 600 or not. It is suggested that the user read Marc Volume A and C and run small test models to access each particular use of gaps. If gaps can be used, set PARAM,MRGAPUSE,1 and re-run the analysis. See also [MARCAPP](#), [MARGAPN](#), and [MARGAPD](#).

MRHERRMN

Default = 1, SOL 600 only.

This entry controls whether extra grid created for such items as hyperelastic Herrmann elements, CWELD, et.c are output or not in the op2, f06, punch and/or xdb files. When Herrmann grids are output, the displacement value is actually pressure which might be confusing when looking at an f06 file.

- 1 Nodal output for extra grids is not provided.
- 1 Nodal output for extra grids is provided when the above files are requested. (Default)

MRHYPMPID

Default = 0, SOL 600 only.

- 0 Hyperelastic models with mid-side nodes will be translated to Marc including the mid-side nodes. Such models might run quite slowly. (Default)
- 1 Hyperelastic models with mid-side nodes will be translated to Marc leaving out the mid-side nodes. These models will normally run faster, but the displacements of the mid-side nodes will be zero and thus plots might look strange unless the model without mid-side nodes is read into the postprocessor using the .marc.dat or .t16 files.

MRITTYPE,

Default = 0, SOL 600 only.

Type of "constraint" used to connect the auxiliary nodes in all CWELDs.

- 0 RBE3 constraints will be used. (Default)
- 44 Kirchhoff constraints will be used
- 80 Combined RBE2 and RBE3 constraints will be used.

MRMAT8A3, Value

Default = 1.0, SOL 600 only.

Value If solid composites are modeled using MAT8, the third coefficient of thermal expansion, ALPHA3 is not defined. ALPHA3 is calculated as ALPHA3=value * ALPHA1.

MRMAT8E3, Value

Default = 0.1, SOL 600 only.

Value If solid composites are modeled using MAT8, the third modulus, E3 is not defined. E3 is calculated as E3=value * E1.

MRMAT8N1, Value

Default = 0.5, SOL 600 only.

Value If solid composites are modeled using MAT8, NU31 is not defined. NU31 will be calculated as NU31=value * NU12.

MRMAT8N3, Value

Default = 1.0, SOL 600 only.

Value If solid composites are modeled using MAT8, the NU23 is not defined. NU23 is calculated as NU23=value * NU12.

MRMAXISZ

Default = Value in Marc include file in tools directory. SOL 600 only.

If this value is entered, the integer value will be used on the command line to run Marc as -maxsize N where N is the integer entered.

MRMAXNUM

Default = Value in Marc include file in tools directory. SOL 600 only.

If this value is entered, the integer value will be used on the command line to run Marc as -maxnum N where N is the integer entered.

MRMEMSUM

Default = 0, SOL 600 only.

If MRMEMSUM=1, a summary of memory used by the internal Nastran-to-Marc translator will be printed in the f06 file. Each line will contain four numbers (all are in 4-byte words). The first number is the current memory request, the second the current memory (in addition to standard MD Nastran open core), the third is the memory limit with zero meaning no memory limit, and the fourth is the high water memory used so far by the Nastran-to-Marc translator and/or the t160p2 results conversion.

This parameter must be entered in the Case Control.

MRMTXKGG, Name

No Default, SOL 600 only.

Name Character. The NAME will be used for K2GG entries if entered. This parameter is similar to MRMTXNAM. Either MRMTXKGG or MRMTXNAM can usually be entered. The proper Case Control K2GG=NAME or K2PP=NAME will be selected automatically by SOL 600 as follows: If the continue option involves dynamic response analysis, K2PP will be used. If the continue option involves eigenvalue extraction or static analysis, K2GG will be used.

MRMTXNAM, Name

No Default, SOL 600 only.

Name Character. The NAME (field 2 of the DMIG entry) that will be used for DMIG values in a spawned Nastran execution. For example, a common name used frequently is K2XX. The Case Control command K2PP=Name will be added (in this example K2PP=K2XX will be added) at the end of the Case Control of the spawned job. DMIG entries with other names may exist on the file, but only those with NAME will be used in the spawned execution. Either MRMTXKGG or MRMTXNAM can usually be entered. The proper Case Control K2GG=NAME or K2PP=NAME will be selected automatically by SOL 600 as follows: If the continue option involves dynamic response analysis, K2PP will be used. If the continue option involves eigenvalue extraction or static analysis, K2GG will be used.

MRNOCOMP

Default = 1, SOL 600 only.

Determines whether n-layer composite will be created if CORDM is defined on the PSOLID entry. This allows output in material coordinate systems.

- 1 Composite are not created (automatically activated for brake squeal). (Default)
- 1 1-layer composites are created.
- N N-layer composites are created.

MRNOCOR

Default = 0, SOL 600 only.

0	Marc parameters ELASTICITY, PLASTICITY, UPDATE, LARGE DISP, CONSTANT DILATATION will be automatically adjusted as recommended by the Marc developers.
1	the above parameters will be adjusted as determined to be the most consistent in correlation between Nastran and Marc results for similar problems. This parameter should be entered for buckling problems without plasticity. PARAM,LGDISP,1 should also be entered.

MRNOECHO

Default = 0, SOL 600 only.

Determines whether various outputs are placed in the Marc file.

- 0 Output is not suppressed (unless other entries are made to suppress it). (Default)
- 1 Suppress echo of nodes and element lists
- 2 Suppress echo of boundary conditions
- 3 Suppress echo of nurbs data

Note: Enter any combination to suppress whatever is desired. To suppress all items, enter 123.

MRORINTS

Default = 99, SOL 600 only.

Controls orientation type for all solid elements in model (see Marc Volume C documentation, ORIENTATION option).

- 1 Edge 1-2
- 2 Edge 2-3
- 3 Edge 3-4
- 4 Edge 3-1
- 5 Edge 4-1
- 6 XY Plane
- 7 YZ Plane
- 8 ZX Plane
- 9 XU Plane
- 10 YU Plane



- 11 ZU Plane
- 12 UU Plane
- 13 UORIENT Define transformation matrix with orient.f user subroutine
- 14 3D ANIO
- 91 Composite orientations will be Edge 1-2 for shells and 3D ANSO for solids and all non-composites will use option 99. This option provides compatibility with earlier versions and requires that all referenced coordinate systems by composite elements by rectangular.
- 99 All orientations are specified by coordinate systems (CORD2R, CORD2C, etc.). (Default)
- 101-112 Shell elements will have options 1-12, solid element will have option 14. (See Note 2)

Notes:

- 1. For solid composites, it is necessary to use the PSOLID entry as well as the PCOMP entry. The PSOLID entry should normally have an entry in field 4 which specifies a CORDi entry to use for the material alignment direction.
- 2. If option 101 is picked, shells will have edge 1-2 and solids will have 3D ANISO. If option 102 is picked, shells will have edge 2-3 and solids will have 3D ANISO, etc.

MROUTLAY

Default = -9999, SOL 600 only.

Indicates which shell or composite layers are to be output using the MARCOUT Bulk Data entry or by default. Options are as follows:

- 9999 All layers will be output. (Default)
- N Layers 1 and abs(N) will be output. (N must not be -1.)
- 1 If the number of layers is less than 21, same as -N option. If the number of layers is greater than 21, layers 1, 11, 21, 31, ..., N will be output.
- 0 Top and Bottom layers only will be output.
- N Layers 1 through N will be output.
- 9999 Stresses at the element center only will be output (top and bottom are not output). This option produces output with the assumption that the element has constant stress and strain throughout the element.

Remarks:

1. Options -9999, -N, -1 and N also produce output at the element center.
2. MROUTLAY = 0 is the same as MROUTLAY = -1.
3. Do not enter this parameter if Bulk Data entry MLAYOUT is used.

MRPARALL

Default = 0, SOL 600 only.

Determines whether parallel processing for SOL 600 will be forced even if Nastran detects that it might fail prior to spawning Marc.

- 0 Spawn Marc in single-processor mode if Nastran detects Marc may fail in DDM mode.
(Default)
- 1 Spawn Marc in DDM mode even if Nastran detects Marc may fail. Leave any COORD SYS entries in the Marc input file. If COORD SYS occur in the input file a DDM Marc job may fail depending on the version of Marc being used. Please read the following remark carefully before using this option.
- 2 Spawn Marc in DDM mode even if Nastran detects Marc may fail. Remove all COORD SYS entries from the Marc input file. Please read the following remark carefully before using this option.

Remark:

Known situations where Marc does not presently work in parallel are as follows:

When local coordinate systems are specified for the following:

Field 7 of any GRID entry.

Material coordinate systems (for example field 8 of CQUAD4).

If CBUSH uses coordinate systems.

For the following cases, this situation can be avoided and PARAM,MRPARALL does not need to be entered:

If coordinate systems are used only for input (field 3 of the GRID entry) or just entered in the Nastran input but not actually used, enter PARAM,MRCORDS,1 to allow parallel processing to proceed.

If coordinate systems are only used for shell orientation (such as field 8 of CQUAD4) and if all such coordinate systems are rectangular the following two parameters may be used to allow parallel processing.

PARAM,MRCORDS,1

PARAM,MRORINTS,1

MRPBUSHT

Default = 0, SOL 600 only.

Determines whether the requirement that when PBUSHT and contact occur in the same model, BCTABLE must be specified for each subcase.

- 0 This requirement is enforced and the job will abort if BCTABLE's are not entered
- 1 This requirement will be ignored and the job will proceed. Wrong results may occur and/or the job may terminate later in the run.

MRPIDCHK

Integer, Default = 1, SOL 600 only.

Controls whether a check for duplicate PIDs (property IDs) will be made in SOL 600.

- 1 Do not make a check for duplicate PIDs.
- 1 Check for duplicate PIDs for structural elements - if any are found fatal the job.

Remarks:

1. This parameter may be set in RC files.
2. This parameter may be used in the MD Nastran 2010 and subsequent versions.
3. Duplicate PIDs will usually cause SOL 600 to fail or obtain the wrong results if the duplicates occur for structural elements. It is usually acceptable to have a duplicate PID for a spring or damper and for a structural element.
4. For versions prior to MD Nastran 2010, SOL 600 behaved as if PARAM,MRPIDCHK,-1 was set.

MRPELAST

Default = -1, SOL 600 only.

Determines whether PELAST will be skipped or cause the job to abort for SOL 600. SOL 600 does not support PELAST. PBUSHT along with CBUSH and PBUSH should be used instead.

- 1 SOL 600 jobs with PELAST that are referenced by any CELAS will abort.
- 1 PELAST entries will be skipped (ignored).

MRPLOAD4

Default = 2, SOL 600 only.

Determines whether continuation entry for PLOAD4 will cause the run to stop or if the continuation line(s) are to be ignored.

- 0 The job will stop and a “Severe Warning” message will be issued.
- 1 The job will continue and the continuation lines will be ignored. A warning message will be issued for the first few such entries.
- 2 CID, N1, N2, N3 and SOLR will be used. If SORL=LINE, the direction must be specified using the CID, N1, N2, N3 fields for SOL 600. LDIR is ignored. (Default)
- 3 If any CSSHL elements are used, the pressures at the corners will be averaged and applied to the entire element for all element types in the model.

Remarks:

1. MRPLOAD4 should not be set to 2 for axisymmetric or plain strain analyses or if parallel processing is used unless the “single file Marc input file” option is used. In some cases, but not all, if MRPLOAD4=2 is set it will automatically be changed to zero one of these conditions exist.
2. See parameter MRPLOD4R for a companion control.
3. If any CSSHL elements are used, the pressures at the corners will be averaged and applied to the entire element for all element types in the model.

MRPLOD4R

Integer, Default = 0, SOL 600 only.

Determines how PLOAD4 pressures are treated in Marc when PARAM,MRPLOAD4,2 is set.

- | | |
|---|--|
| 0 | Pressures are treated as constant over the element if the pressure over the element varies, the average is applied. |
| 1 | The Marc PLOAD4 style of loads is used which allows the pressures at the four corners of a quad surface to be different. |

Remark:

Versions of Marc including MDR3 and Marc 2008R1 and versions previous to these versions produce incorrect results for some models using MRPLOD4R=1 if total loads (specified by PARAM,MARCTOTL,1 or MARCTOTT,1) are used even if the pressure is the same at the four corners of each quad element.

MRPLSUPD

Default = 1, SOL 600 only.

Determines whether Marc’s PLASTICITY,3 will be used or not for fast integrated composites or smeared composites. Wrong results may be obtained if PLASTICITY,3 is used for these types of analyses even if the plasticity is in non-composite portions of the model (Default = 1 if this parameter is omitted).

- | | |
|----|---|
| -1 | Use PLASTICITY,3 if it is necessary for non-composite regions of the model |
| 1 | Replace PLASTICITY,3 with UPDATE (FINITE and CONSTATN DIALITATION are not used. (Default) |

Remark:

SOL 600 determines if smeared composites are used by the presence or lack of the SMEAR word on the SOL 600,ID entry. Fast composites are determine if PARAM,MFASTCMP is set to 2 or 3 or if any PCOMPF Bulk Data entries are present in the model.

MRPOISCK

Integer, Default = 2, SOL 600 only

Controls whether to check if a “bad” Poisson ratio has been entered in SOL 600 for MAT1 entries.

- 0 Do not check or correct “bad” Poisson ratios.
- 1 Check if Poisson ratio is negative or > 0.5, if so change it to 0.333.
- 2 Check if Poisson ratio is negative or > 0.5, if so change MAT1 to MATORT with all three E values, the same as E for MAT1, all three G values the same as G for MAT1 and
 $v_{12}=v_{23}=v_{31}=0.333$

Remarks:

1. Current Marc versions do not allow isotropic materials with Poisson ratios having negative values or values greater than 0.5. Previous Marc versions may have allowed this.
2. “Bad” Poisson ratios may occur if the user specifies E and G and leaves the Poisson field on MAT1 blank.
3. If mrposick=1, the value of G will change. The value of E remains as originally specified.
4. If mrposick=0, the job will error out in the Marc phase of SOL 600.
5. This parameter is available starting the MD Nastran 2010.

MRPREFER

Default = 1, SOL 600 only.

Determines to output SOL 600 stresses on the t16 file in the standard Marc coordinate system for the element or the “preferred” (layer) coordinate system when the model contains composite elements.

- 0 Stresses are output in the standard coordinate for the element
- 1 Stresses are output in the “preferred” (layer) coordinate system. (Default)

Remarks:

1. The standard coordinate system for solids is usually the basic coordinate system. The standard coordinate system for shells and beams is usually the element coordinate. See Marc documentation volumes A and B for further details.
2. Some versions of Patran cannot plot stresses for MRPREFER=1.

MRPRSFAC

Default = 0.0, SOL 600 only.

This parameter is primarily used by Versions of SOL 600 prior to MD Nastran R2 and MSC Nastran 2007 which could not support different pressures at the different corners, pressures applied in directions that are not normal to a surface or edge loads. This parameter is not necessary starting with the MD Nastran R2 and MSC Nastran 2007 Version 5 if PARAM,MRPLLOAD4,2 is set.

Factor by which pressure loads are scaled for SOL 600. Prior to MD Nastran 2006 R1 and MSC Nastran 2006, Marc was not capable of handling different pressure applied to different corners of a surface. In other words, different values of p1, p2, p3, p4 on the PLOAD4 entry could not be handled. Approximations can be made by assuming the pressure is uniform over the surface if the surface is small enough. For SOL 600, the pressure (P) used by Marc is calculated using the following:

If p1 is not blank (or zero) regardless of the value of mrprsfac
 $P = p1$

If mrprsfac is positive and p1 is blank or zero and one or more of p2, p3, or p4 are not zero or blank,
 $P=mrprsfac*(p1+p2+p3+p4)$ (This default if p2 or p3 or p4 are not zero or blank)

If mrprsfac is zero or negative,

If none of p1, p2, p3, p4 are blank or zero $P=0.25*(p1+p2+p3+p4)$

Otherwise,

$P=(+/-)pp$ where pp is the value largest absolute value of p1, p2, p3, p4 and P will have the proper associated sign.

Remarks:

1. If MRPLOAD4=2, MRPRSFAC is ignored and p1,p2,p3,p4 are used as entered.
2. If MRPLOAD4 is not 2, the default for mrprsfac varies from element to element. For each element it is 1.0 divided by the number of Pi defined. For example, if only one of p1,p2,p3,p4 is defined, the default mrprsfac is 1.0. If two of p1,p2,p3,p4 are defined, the default for mrprsfac is 0.5. If three are defined the default is 0.33333 and if all four are defined the default is 0.25. When PARAM,MRPRSFAC is entered, the value is the same for all elements with pressure specified by PLOAD4.

Note: The default for mrpsfac varies from element to element. For each element it is 1.0 divided by the number of Pi defined. For example, if only one of p1,p2,p3,p4 is defined, the default mrprsfac is 1.0. If two of p1,p2,p3,p4 are defined, the default for mrpsfac is 0.5. If three are defined the default is 0.33333 and if all four are defined the default is 0.25. When PARAM,MRPRSFAC is entered, the value is the same for all elements with pressure specified by pload4.

MRPSHCMP

Integer, Default = 0, SOL 600 only

If MRPSHELL=1 and there is a Marc PSHELL and composite for the same elements, this option controls which will be used.

- 0 Use Marc composite and ignore Marc pshell. (Default)
- 1 Use Marc pshell and ignore Marc composite.
- 2 Do not make any corrections (this works for some models and fails for others)

MRPSHELL

Default = 0, SOL 600 only.

Used to control shell property specifications for Marc in SOL 600.

- 0 Classical GEOMETRY options will be used for all shell elements. (Default)
 1 The Marc PSHELL option will be used, (can only be used with CQUAD4, CQUAD8, CTRIA3, and CTRIA6).

Important: If MRPSHELL=1, shell elements will use Marc's new PSHELL option. No materials used by PSHELL may be referenced by other types of elements that this option has used. For commonly used elements, duplicate materials will automatically be created by SOL 600 to satisfy this criteria, however this can take considerable computer time. If the user knows that no materials are used by both PSHELL and other properties, he can set PARAM,MRDUPMAT,-1 to bypass the checking and creation for duplicate materials. To speed up this process, set PARAM,MSPEEDPS,1. The Marc PSHELL formulation is more stable if MID3=MID2. To set MID3=MID2, set PARAM,MARCMID3,1.

MRRBE3TR

Default = 0, SOL 600 only.

Controls whether all translations or all translation and rotations will be used for the reference degrees of freedom or whether fewer than 3 are allowed.

- 0 Fewer then 3 reference degrees-of-freedom may be used. (Default)
 3 All RBE3's with REF C (field 5 of the RBE3 entry) not having 123 will have all translations for the Marc REFC (field 5 of the Marc RBE3 entry) set to 123.
 6 All RBE3's with REF C (field 5 of the Nastran RBE3 entry) not having 123456 will have all translations and rotationf or the Marc REFC (field 5 of the Marc RBE3 entry) set to 123456.

Remarks:

1. Marc frequently gives a singular tying message if there are not at least degrees of freedom 123 for REFC. This sometimes results in poor singularity ratios and incorrect results for nonlinear analyses. It also can prevent eigenvalues from being calculated.
2. It is recommended that option 3 or 6 be used for all SOL 600 analyses unless the user is certain that REFC can have fewer then three dof's for the particular analysis.

MRRCFFILE, RCF

No Default, SOL 600 only.

- RCF Character. Name of RCF file name (limited to 8 characters) used in conjunction with another MSC Nastran run spawned from an original MSC Nastran run as specified by the CONTINUE option on the SOL 600 command. The RCF file may contain any information required (such as scratch=yes, exe=, etc.) as discussed in [Executing MSC Nastran, 2](#) of this guide. This rcf file does not have to use the same options as the primary rcf file and should normally set batch=no as one of the options.

MRRELNOD

Default = -1, SOL 600 only.

Controls enforced displacements using SPCD.

- 0 For multiple subcases with SPCDs, the SPCDs from the previous subcase will be released at the start of the current subcase.
- 1 SPC's and SPCD's from the previous subcase, not specified again in the current subcase, will be released gradually during the current subcase.
- 1 SPC's and SPCD's from the previous subcase, not specified again in the current subcase, will be subtracted out. For example, if the previous subcase applied a SPCD of 0.1 to a particular dof, the current subcase will apply -1.0. This will bring the displacement of that dof to zero for the current subcase.

MRRSTOP2

Default = 0, SOL 600 only

Integer, Normally op2, xdb, punch and f06 output is not available for SOL 600 restart analyses. Setting this parameter to 1 will allow the program to attempt to create one or more of these files. Only in limited cases will the job be successful.

- 0 Do not attempt to create .op2, xdb, punch, f06 output for restart runs (default if parameter is not entered). (Default)
- 1 Attempt to create .op2, xdb, punch, f06 output as specified by other options.

MRSCMOD

Default = 0.0, SOL 600 only.

Solution scaling factor for linear buckling analysis (SOLs 600, 105) using the Lanczos method. If the applied load in the first subcase is too large, the Lanczos procedure may fail. This number may be used to scale the solution for numerical reasons. The collapse load will be output based upon the total load applied.

Remark:

This parameter applies to rigid surfaces described using 4-node patches only.

MRSETNA1, N

Default = Program calculated, SOL 600 only.

- N If this parameter is entered with N>0, the integer value entered will be used in Marc's SETNAME parameter section as the first value of SETNAME,N,M which is an undocumented Marc option. N is the number of sets and M is the largest number of items in any set. This entry is sometimes necessary if large lists of elements or nodes are used to describe materials, properties or contact. Both MRSETNA1 and MRSETNA2 must be included for either to take effect. This option is no longer required for MSC Nastran 2005 r2 and subsequent releases.

MRSETNA2, M

Default = Program calculated, SOL 600 only.

- M If this parameter is entered with M>0, the integer value entered will be used in Marc's SETNAME parameter section as the second value of SETNAME,N,M which is an undocumented Marc option. N is the number of sets and M is the largest number of items in any set. This entry is sometimes necessary if large lists of elements or nodes are used to describe materials, properties or contact. Both MRSETNA1 and MRSETNA2 must be included for either to take effect. This option is no longer required for MSC Nastran 2005 r2 and subsequent releases.

MRSETNAM, N

Integer, Default = Program calculated, SOL 600 only.

- N If this parameter is entered with N>0, the value entered will be used in Marc's SETNAME parameter section. This entry is sometimes necessary if large lists of elements or nodes are used to describe materials, properties or contact. To get around a Marc bug, if a computed setname value is large, Nastran will normally use the undocumented form (see MRSETNA1). To prevent this, set N to -1.

MRSPAWN2, CMD

Default = Nastran, SOL 600 only.

CMD Character. Name a command to run D Nastran (limited to 8 characters single field or 16 for large field) used in conjunction with the CONTINUE options on the SOL 600 command. The Nastran run to be spawned will have the form:

CMD jid.nast.dat rcf=RCF

where file RCF is provided by PARAM,MRRCFILE,RCF. As an example, if CMD is nastran, jid is myjob (original file myjob.dat) and RCF=nast.rc, the spawned run will execute using:

nastran myjob.nast.dat rcf=nast.rc

Notes:

1. See PARAM*,HEATCMD for SOL 600 thermal contact heat transfer analysis.
2. CMD will be converted to lower case regardless of the case entered.

MRSPLINE

Default = 2, SOL 600 only.

Determines whether RSPLINE will be converted to CBAR or to MPC's.

- 1 RSPLINE will ignored (this option should only be used for model checkout)
- 1 RSPLINE will be converted to stiff CBAR elements
- 2 RSPLINE will be converted to MPC's (Marc's servo link) using the same equations used by other Nastran solutions, except that the MPC's are capable of large rotation. This option requires somewhat longer translation times than the others but can save computer time during the Marc execution phase. (Default)

MRSPRING

Default = 0.0, SOL 600 only.

Specifies a stiffness value to be added to the main diagonal of each translational term of the stiffness matrix. This option is useful in nonlinear static analysis with 3D contact of two or more separate structures. Some of the pieces may not be grounded until contact occurs. By adding a small spring to ground, such as K=1.0, these pieces are stabilized until contact occurs. This option applies to SOL 600 only and generates SPRINGS in the Marc input file for all nodes in the model and all three translational degrees-of-freedom. IF K is entered as a negative number, the absolute value of K will be added to all 6 degrees of freedom of each grid in the model.

If the run is for heat transfer and K is positive, the spring will only be added to DOF 1.

MRSRVER

Integer, Default = -1, SOL 600 only

Controls how CELAS and all other items map to the Marc input data.

- 0 Non-table form is used which generally means the Marc input file version is less than 11.
- 11 The table form for SPRINGS is used which requires two lines, the first with spring rates and the second with table ID's. PARAM,MRSRVER,11 is usually required if PARAM,MRCONVER,11 is entered.

MRSTEADY

Default = 1, SOL 600 only.

Controls the solution method for SOL 600 steady state heat transfer.

- 1 Marc STEADY STATE is used with TIME STEP of 1.0. (Default)
- 2 AUTO STEP is used.

MRT16STP, N

Default = 0, SOL 600 only.

Enter in Case Control at the subcase level.

- N Number of load increments to put on the t16/t19 files for each subcase when the AUTO STEP method is used. The default of 0 puts all adaptive increments on the t16/t19 file. If a value of N is entered, load steps for times=tmax/N will always be introduced into the auto stepping process and the t16/t19 files will have outputs only at zero and those times. This corresponds to field 1 of Marc's AUTO STEP 2nd option and can also be set using the NLAUTO option. If this is the only non-default NLAUTO variable to set, it is more easily accomplished using this parameter. It is suggested that this parameter always be used for large models and that N be 10 or greater, otherwise the size of the t16/t19 files may become very large. This is especially important for Windows systems, which presently has a 4GB limit in converting t16 to op2 files due to compiler limitations.

MRTABLIM

Integer, Default = 0, SOL 600 only

Determines the maximum number of points in any TABLES1 used when tables are specified or MATHE input.

- 1 Unlimited, not recommended - If used could lead to numerical errors or program aborts
- 0 The limit is 500 points.
- N The limit is set to N points. It is recommended that N does not exceed 500.

Remark:

If more than about 500 points are used for the tables, the curve filling processes used to determine the material constants may encounter numerical errors and the program could abort. If a table with more than the number points specified by this parameter is entered, the number of points will be reduced to the specified limit so that numerical errors are unlikely to occur.

MRTABLS1

Default = 0, SOL 600 only.

TABLES1 stress-strain curves are converted to Marc WORK HARD stress-plastic strain curves according to the following formulas for the value entered for MRTABLS1. In the formulas s is the stress entered for TABLES1, e is the strain entered in TABLES1, S is the Marc WORK HARD stress and E is the Marc WORK HARD Plastic strain. ey is the yield strain (sy/E) where EE is Young's modulus, sy is the yield stress. The first point of the Nastran curve will be skipped unless MRTABLS2 is set to 1.

0 (Default)	$S=s$ $E=e - ey$
1	$S=s(1+e)$ $E=\ln(e+1)$
2	$S=s$ $E=e - s/E$
3	$S=s(1+e)$ $E=\ln(1+e) - s/EE$
4	$S=s$ $E=e$
5	$S=s$ $E=\ln(1+e)$
6	$S=s$ $E=\ln(1+e-s/E)$
7	$S=s(1+e)$ $E=\ln(1+e)-S/EE$
8	$S=s(1+E)$ $E=\ln(1+e)$
9	$S=s(1+E)$ $E=\ln(1+e)-\ln(1+ey)$

Note: This parameter (like any other parameter) can only be entered once in an input file.

MRTABLS2

Default = 0, SOL 600 only.

- 0 TABLES1 stress-strain curves are converted to Marc WORK HARD stress-plastic strain curves starting with the yield point. The first point will be skipped. (Default)
- 1 All points in TABLES1 will be connected to WORK HARD, however, the first plastic strain will be set to zero if entered as non-zero.

Note: This parameter (like any other parameter) can only be entered once in an input file.

MRTABPRC

Integer, Default = 0, SOL 600 only

Determines the precision of stress-strain curves when translated from Nastran to Marc using SOL 600.

- 0 Standard precision (8 significant figures) is used.
- 1 Maximum precision (14 significant figures) is used.

Remarks:

1. This parameter is available starting with the 2010 entry. For MD Nastran 2010 it only applies to stress-strain curves entered in conjunction with MATEP.
2. Prior to MD Nastran 2010, only option zero was available.
3. In most cases, the stress-strain values are not known to even 8 significant figures. However, if the stress-strain is calculated from other data the extra precision might be useful.

MRTFINAL

SOL 600 only.

Real. In some dynamic problems, due to numerical roundoff an extra time step with very small initial and final times is generated. If these times are too small, Marc sometimes will diverge even though the "final time" actually desired by the user has been reached within reasonable accuracy. MRTFINAL is a value below which this extra step will be eliminated to prevent excess computations and/or possible divergence. If this parameter is not entered, the default value is 1.0E-8. This parameter is only used with the AUTO STEP procedure. The 'extra' step will also be eliminated if the initial time step is less than MRTFINAL/100.0.

MRTHREAD

Integer > 0, Default = 1, SOL 600 only.

Determines the number of threads to be used by the run. For multi-core processors speed can frequently be gained by specifying a number of threads equal to the number of cores assuming the job has exclusive use of the machine.



- 1 Use 1 thread.
- 2 Use 2 threads.
- N Use N threads.

Remarks:

1. This parameter may only be used by solver types 8 (multifrontal direct sparse) and 11 (Pardiso).
2. This parameter may be combined with PARAM,MARPROCS to use several processors each using several threads.

MRTIMING

Default = 0, SOL 600 only.

If MRTIMING is 1, timing summaries for various portions of the internal Marc translator will be provided in the f06 and log files.

MRTSHEAR

Default = 0, SOL 600 only.

By default, parabolic shear deformation is not included in the formulation of beam and shell elements if there are no composite elements. If composite shell elements are present, parabolic shear deformation is included by default. This parameter can be used to override these defaults.

- 1 Parabolic shear deformation is included in the formulation of beam and shell elements.
- 1 or 0 Parabolic shear deformation is not included even if composite elements are present.
(Default)

Remark:

This parameter maps to Marc TSHEAR parameter.

MRUSDATA

Default = 0, SOL 600 only.

Determines whether user subroutine USDATA will be activated and if so the size of data.

- 0 User subroutine USDATA is not active in the current run. (Default)
- N User subroutine USDATA will be active the in the current run and N is the number of real*4 words needed for the data stored in common block /usdam/

Remark:

In addition to adding this parameter, the user must enter USDATA on Bulk Data entry USRSUB6.

MRVFIMPL

Default = 0.01, SOL 600 only.

Controls the fraction of the maximum view factor that is to be treated implicitly (contribute to operator matrix). View factor values smaller than this cutoff are treated explicitly.

MRV09V11

Default = 1, SOL 600 only.

Determines whether certain Marc “features” which are default in Marc “version 11” are added for SOL 600-generated models that use “version 9”.

- 1 Do not add the features.
- 1 Add the following features:
 - feature,4703 to speed up DDM jobs for one-processor jobs, it has no effect
 - feature,5701 to disable old rigid rotation checking which was too stringent
 - feature,601 to improve contact
 - feature,5301 to improve deformable-deformable contact
 - feature,3201 to improve contact friction types 6 and 7“
 - feature,5601 to improve thickness updating when the updated Lagrange method is used
 - feature5801 to improve in-plane bending of advance nonlinear element type 140
 - feature,6001 to improve concrete cracking analysis

Note: The above features are only used for certain problems, even though all are included with the default option, they have no effect on models that do not use advance nonlinear element type 140, feature 601 has no effect on models that do not have contact, etc.

MSIZOVRD

Default = -1, for small models and +1 for large models, SOL 600 only.

Determines whether Marc SIZING values for large models will be updated or not. Nastran includes Marc directories with a file named “include” in the tools directory which includes a line MAXNUM=N, where N is some value like 1000000. If the maximum number of nodes or elements in the model exceeds N, memory overwrites or job aborts are possible unless either N is set larger than the actual max node or element number in the model or the values are specified on the SIZING entry (field 3 for max element and field 4 for max node).

- 1 The sizing entry will not be updated (either the model is not large, MAXNUM in the include file has been updated or parameters such as MARCSIZ3 and/or MARCSIZ4 have been entered into the run to provide values that are large enough. (Default)
- 1 If the number of nodes or elements in the model exceeds 1,000,000 the sizing entry will be updated to the max number of nodes and elements actually in the model. Extra nodes and/or elements to account for welds, pinned members, Herrmann elements, etc. will be included.

Note: For PARAM,MSIZOVRD large models are considered to be those with more than 1,000,000 nodes and/or elements.

MSOLMEM, MBYTE

Default = Program determined value, SOL 600 only.

MBYTE If entered, the integer value entered here is the 8th field of Marc's SOLVER option, and is the maximum memory in Mega Bytes that can be used by Marc's solver types 6 (hardware provided direct) and 8 (multi-frontal direct, which is the default solver for SOL 600) before going out of core. This parameter is the same as the MBYTE field on the NLSTRAT entry any may be easier to enter as a parameter if no other NLSTRAT values are needed.

Note: This parameter should only be entered if NLSTRAT entries are not required. If any NLSTRAT entries are made, use the MBYTE field instead of this parameter.

MSPCCCHK

Integer, Default = 0, SOL 600 only

Effects multiple subcases where SPCD's are applied and SPC's potentially change between the subcases.

- 0 A comprehensive check is made to ensure that all SPC changes are accounted for all subcases.
- 1 The check is not made. The check can take significant computer time and in most cases it is not actually needed. If the user knows it is not needed, computer time can be saved by turning it off.

Remarks:

1. This parameter is available starting with MD Nastran 2010.
2. This parameter may be set in rc files.

MSPCKEEP

Default = 0, SOL 600 only.

Determines which SPC's entered in the bdf file are retained in the Marc input file for 2D models.

- 0 Only retain those dof's that are normally associated with the type of 2D analysis (normally 2 translations) (Default)
- 1 Retain all dof entered in the bdf file in the Marc input file.

Remark:

Some types of Marc 2D analyses support one rotation in addition to two translations and it is important to restrain this rotation.

MSPEEDCB

Default = -1, SOL 600 only.

Determines whether CBEAM increased speed options are to be applied. This option may be necessary for models with a large number of beams whose element IDs are large.

- 1 No increased speed options are used for CBEAM. (Default)
- 1 cbeam.prp file will be standard unformatted rather than direct access. A small table to provide the location of each entry is saved in memory.
- 2 The entire cbeam.prp file is saved in memory

Note: See param,marccbar,1 to change CBAR to CBEAM.

MSPEEDCW

Default = 0, SOL 600 only.

Determines whether CWELD elements will be translated to Marc in core (for increased speed) or out of core. This parameter is needed if many CWELD elements are present in the model to avoid large translation times.

- 0 Processed out of core. (Default)
- 1 Process in core.

MSPEEDOU

Default = 1, SOL 600 only.

Determines whether speed enhancements are activated for the t16op2 conversion.

- 0 Speed enhancements are not activated.
- 1 Speed enhancements are activated which will place certain scratch data in memory. For large models make sure enough memory is available (if PARAM,MSPEEDSE and/or PARAM,MSPEEDP4 are used, there is probably sufficient memory unless there is a large number of output "time" points. (Default)

MSPEEDP4

Default = -1, SOL 600 only.

Determines whether PLOAD4 entries will be translated to Marc in core (for increased speed) or out of core.

- 1 PLOAD4 will be processed in core.
- 1 PLOAD4 will be processed out of core. (Default)

Note: The MSPEEDP4=1 option may require more memory than is available on certain computers for large models. Do not use if CWELD elements are present.

MSPEEDPS

Default = -1, SOL 600 only.

Determines whether additional memory will be used if the Marc PSHELL option is invoked (PARAM,MRPSHELL,1) or smear option on SOL 600, ID in order to speed up processing. This will be beneficial if there are many materials and properties in the model and not very beneficial if all elements use just a few properties and materials.

- 1 No additional memory will be used. (Default)
- 1 Additional memory will be used if available.

MSPEEDSE

Default = -1, SOL 600 only.

Determines whether speed enhancements are activated using extra memory and/or special low level I/O routines.

- 1 No speed enhancements will be activated. (Default)
- 1 Solid elements 2D elements data will be processed in core.
- 2 All elements will be processed in core.
- 3 2D and 3D elements will be processed in core, 1D elements will be processed using bioxxx (low level direct access routines used by gino).

Note: The MSPEEDSE=1 option may require more memory than is available on certain computers for large models. MSPEEDSE > 0 should not be used for 64-bit integer versions of Nastran or DEC Alpha computer systems.

MSPEEDS2

Default = -1, SOL 600 only.

Determines whether additional memory will be used when PARAM,MSPEEDSE with options 1, 2, 3 is entered to obtain additional speed.

- 1 Additional memory will not be used. (Default)
- 1 Additional memory will be used and additional speed will usually be obtained.

MSPEEDSH

Default = -1, SOL 600 only.

Determines whether additional speed enhancements will be used to process shell/plate elements (CQUAD4, CTRIA3, etc.) using incore memory.

- 1 No additional speed enhancements are used. (Default)
- 1 Additional speed in-core enhancements will be used. Sufficient memory must be available. If not, the job will abort with an appropriate message.

Note: This parameter should only be used if shell speed enhancements are selected with one of the PARAM,MSPEEDSE options.

MSPLINCO

Integer, Default = 0, SOL 600. For SOL 400, MSPLINCO =1 is always used.

This parameter controls whether to enforce C0 continuity for all spline options if any are requested by setting IDSPL=1 on any BCBODY entry.

- 0 C0 continuity is not enforced.
- 1 C0 continuity is enforced.

MSTFBEM

Default = 0, SOL 600 only.

Determines if all rigid elements (RBE2, RBE3, RBAR, RTRPLT) will be converted to stiff beams. This parameter should only be used if PARAM,MARMPCHK and/or PARAM,AUTOMSET options fail during a Marc execution.

- 1 All rigid elements will be converted to stiff beams or plates after they have initially been formed in the Marc file as RBE2 or RBE3 elements. All 6 dof's at each end of the rigid beam will be used.
- 2 All rigid elements will be converted to stiff beams or plates after they have initially been formed in the Marc file as RBE2 or RBE3 elements. Pin codes specified by the original Nastran rigid elements will be used when possible.

Remark:

See PARAM,MARCSCLR to specify a scale factor for the "Default" properties of these stiff beams.

MSTRFREE

Integer, Default = 0

Determines whether the nodal coordinates in the GEOM1 datablock of SOL 600 OOP2 files will have the original coordinates or revised stress-free coordinates.

- 0 GEOM1 will have the original coordinates (same as defined on GRId entries)
- 1 GEOM1 will have modified coordinates to enforce stress-free conditions if applicable

Remarks:

1. ICOORD on the BCTABLE with ID=0 entry must be set to 1 or 3 for at least one slave-master combination.
2. The parameter only addresses op2 files (t16 and t19 files automatically use MSTRFREE=1)
3. This capability is available starting with MD Nastran 2010. Prior version always used MSTRFREE=0.

MSTTDYNE

Integer, Default = 0, SOL 600 only

Controls whether SOL 600 may have static and dynamic load cases in the same analysis.

- 0 All subcases must either be static or dynamic
- 1 Mixed static and dynamic load cases are allowed.

Remarks:

1. Use of this entry requires SOL 600,106 or SOL 600,NLSTATIC (not SOL 600,NLTRAN).
2. Do not enter , DLOAD, LOADSET, LSEQ, TLOADi in the Case Control or Bulk Data.
3. Each subcase must have NLAUTO entries. Whether the subcase is static or dynamic is controlled by IFLAG on each NLAUTO entry. For static subcases IFLAG=3 and for dynamic subcases IFLAG=0.
4. All loads in each subcase build up from an initial value at the end of the previous subcase to the full value entered on the various load-type entries (such as FORCE, PLOAD4, SPCD, etc.) at the end of the subcase. Time histories for individual load cases are not allowed with this parameter. If all loads for a particular subcase use the same time history, this subcase may be divided into N-1 smaller subcases where N is the number of points on the time history curve. For this situation, the loads for each smaller subcase should be scaled by the amplitude of each point on the time history curve. This technique may also be used to simulate force vs. time for an analysis with all static subcases.
5. When MSTTDYNE=1, the following will automatically be added to the Bulk Data:

```
marcin,-1,0,dynamic,6
param,marcauv,0
```

These two entries may be added by the user in place of PARAM,MSTTDYNE,1 however both should not be done.

MTABIGNR

Default = 0, SOL 600 only.

Determines whether tables for VCCT analyses will be ignored or used.

- 1 VCCT tables are not used
- 0 VCCT tables are used. (Default)

Remark:

Use of tables for VCCT SOL 600 2D analyses with contact is not supported. If the table variation is not important, this parameter may be used to obtain an approximate solution.

MTABLD1M

Default = 1, SOL 600 only.

Option to modify or not modify all TABLED1 entries which do not start with the first point of (0.0, 0.0).

- 1 Do not modify any TABLED1 entries
- 1 Modify all TABLD1 entries that do not start with (0.0, 0.0). (Default)

Remark:

See PARAM,MTABLD1T.

MTABLD1T

Default = 0.01, SOL 600 only.

Specifies the second time value of all TABLED1 entries that do not start with the first point being (0.0, 0.0) if PARAM,MTABLD1M=1. Modified tables will start with the first point of (0.0, 0.0), the second point will be at the time specified by this parameter with an amplitude of the original first amplitude. The 3rd point will be at time 2*MTABLD1T and amplitude of the original first amplitude.

Remark:

The proper value of MTABLD1T to enter to analyze a step input depends on the fundamental natural frequency of the model. If MTABLD1T is too small, the response will not fully build up and if it is too large, it will act like a ramp input rather than a step. The best value to use is approximately 0.1/f_n, where f_n is the first linear natural frequency in Hz.

MTEMPCHK

Integer, Default = 0, SOL 600 only

Controls how temperature-dependent properties are checked in the Marc portion of SOL 600.

- 0 Check at zero degrees only. (Marc feature 9601)
- 1 Check at all temperatures, stop checking and fatal the job when the first problem is found. (Marc feature 9600)
- 2 Check at all temperatures and report all problems, then fatal the job if any problems were found. (Marc feature 9602)

Remarks:

1. This parameter is available starting with MD Nastran 2010.
2. The default in Marc previously was feature 9601.
3. The default in SOL 600 is different than in Marc. The default in Marc is 9600.
4. Checking in Marc will occur from the lowest temperature of any curve to the highest of any curve. Thus, to ensure that all curves pass, define all curves from the lowest temperature of any curve to the highest temperature of any curve (even though one particular curve can never reach the lowest or highest temperature specified on another curve).
5. features,960i will be entered if any TABLEMi or TABL3Di are found.

MTEMPDWN

Default = -1, SOL 600 only.

Option to automatically choose the FeFp multiplicative decomposition plasticity model (PARAM,MARCPLAS,5) for plasticity problems with thermal loading when the temperature decreases (see PARAM,MARCPLAS).

- 1 Use the standard mean normal with large strain method unless PARAM,MARCPLAS is entered.
(Default)
- 1 Attempt to determine whether the temperature at the start of the run is lower than the initial temperature, and if so choose the FeFp method.

Remarks:

1. For some problems with very soft materials, the FeFp method is required to achieve convergence for plasticity cases where the temperature decreases during the early stages of the run. Since most users would not know this, PARAM,MTEMPDWN is offered to automatically let Nastran decide to invoke this method or not.
2. The FeFp method takes substantially more computer time than the default mean normal method.
3. This parameter can be entered in RC files if it is desired to be used for a large number of runs.

MTET4HYP

Integer, Default = 0, SOL 600 only

Controls settings for TET4 elements with hyperelasticity.

- 0 Use Marc element 157 which uses the Herrmann formulation.
- 1 Use Marc element 134 and automatically set PARAM,MARUPDAT,1

Remarks:

1. MTET4HYP=1 should normally only be set if any tet4 elements use MATHE or MATHP and if one or more of the grids in such tet4 have field 7 set to a cylindrical or spherical coordinate system.
2. If Remark 1 does not apply, MTET4HYP=0 is more accurate, although it is highly recommended that tet10 elements be used for all SOL 600 models which will be even more accurate.

MTLD1SID

Integer, Default = -1 for SOL 600

Determines whether the SID of TLOAD1s entered in the input will be changed or not. (SOL 600 only)

- | | |
|----|--|
| -1 | Do not change the SID of any TLOAD1 entry |
| 0 | Change the SID of all TLOAD1s to match the lowest SID of any TLOAD1 found in the original input. |
| N | Change the SID of all TLOAD1s to the value N. |

Remark:

This parameter may only be set in the input file.

MUALLUDS

Integer, Default = 0, SOL 600 only

Controls how material, contact and element-related user subroutines are specified in SOL 600.

- | | |
|---|---|
| 0 | User subroutines are entered using the MATUSR entry (material), contact-related (BCBODY, BCTABLE, etc.) or element entries as well as the USRSUB6 entry. |
| 1 | User subroutines are entered using the MATUDS, BCONDUDS and/or ELEMUDS. They should not be specified on MATUSR, BCxxx, element entries or by using the USRSUB6 entry. |

Remark:

For version prior to MD Nastran 2010 only option 0 was available.

MULRFORC

Default = -1, SOL 600 only.

Option to activate multiple RFORCE entries for different portions of The model in the same subcase.

- | | |
|----|--|
| -1 | Do not allow this capability (only one RFORCE entry for subcase can be entered). (Default) |
| 1 | Allow multiple RFORCE entries for each subcase |

MUMPSOLV

Integer > 0, Default = 1, SOL 600 only.

This can also be set by the Bulk Data entry PARAMARC.

- | | |
|---|------------------|
| 1 | Use 1 processor. |
| 2 | Use 2 processor. |
| N | Use N processor. |

Remarks:

1. This parameter may only be used by solver the MUMPS solver (solver type 12).
2. The PARAMARC Bulk Data entry cannot specify this type of parallel processing.

MUSBKEEP

Default = 0, SOL 600 only.

- | | |
|---|--|
| 0 | If user subroutines are included, they are compiled and linked to form a new version of Marc if MUSBKEEP=0, the new version of Marc will be deleted at the end of the run. (Default) |
| 1 | The new Marc executable will be saved on disk in the same directory as the Nastran input file. Its name will be the name used in PARAM,MARCUSUB with the extension marc. |

MVERMOON

Integer, Default = 0, SOL 600 only

Controls whether 5-term Mooney series or 5-constant Mooney will be used in the Marc portion of SOL 600.

- | | |
|---|--|
| 0 | 5-constant Mooney formulation will be used. |
| 1 | 5-term series Mooney formulation will be used. |

Remark:

Prior to MD Nastran 2010, the Marc only allowed a 5-constant Mooney formulation with no volumetric terms. Now Marc allows a full 5-term series for both distortional and volumetric behavior for both MATHP and MATHE entries. If a version of Marc prior to this is to be used, set param,mvermoon,0 or use the default.

MWINQUOT

Integer, Default = 0, SOL 600 only

Determines whether SOL 600 run on Windows will attempt to place quote strings in the command to execute Marc when PATH is not specified on the SOL 600 Executive Control statement.

- | | |
|---|---|
| 0 | Attempt to place quote strings around various fields. |
| 1 | Do not place any quote strings. |

Remarks:

1. Quotes are normally only required if spaces are needed in the file names or directory names.
2. This parameter may be placed in rc files.
3. Depending on the complexity of the command to execute Marc, quote strings may sometimes be incorrect. If the command is complex, it is suggested that PATH=1 be placed on the SOL 600 Executive statement and the statement to execute Marc be placed in the marcrun.pth file.

MWLDSGSA

Integer, Default = 1, SOL 600.

Controls CWELD type ELEMID GS, GA, GB usage in SOL 600.

- 0 When GS, GA and GB are all greater than zero for CWELD type ELEMID, Marc ignores GS and uses GA and GB to determine if a connection is possible. MWLDGSA=0 retains this default behavior, however experience shows that some models will fail with a message that the weld connection could not be achieved.
- 1 This option will re-set GA and GB to zero so that GS will be used to achieve the connection. Experience shows that this option may be more reliable for many models with CWELD type ELEMID.

MXICODE0

Default = 5

In nonlinear harmonic response analysis, if the solution fails to converge more than MXICODE0 times in succession, a new trial displacement vector is calculated.

MXLAGM1

Default = 0.0

MXLAGM1 is to override the maximum Lagrange Multiplier that is used by the merit function in Trust Region.

NASPRT

Default = 0

NASPRT specifies how often data recovery is performed and printed in SOL 200. By default, SOL 200, in addition to performing an analysis and optimization, will perform full data recovery operations at the first design cycle and upon completion of the last design cycle.

If NASPRT > 0, then data recovery operations are performed at the first design cycle; at every design cycle that is a multiple of NASPRT; and the last design cycle. For example, if PARAM,NASPRT,2 and the maximum number of design iterations requested is 5, then data recovery is obtained at design iterations 1, 2, 4, and 5.

If NASPRT < 0, then no data recovery operations are performed.

NDAMP, NDAMPM

NDAMP:

Default = 0.01 for SOLs 129 and 159, -0.05 for SOL 400 heat transfer and mechanical non-contact analysis, 0.0 for SOL 400 contact.

NDAMPM:

Default = 0.0 for SOL 400 non-contact, 1.0 for SOL 400 contact. Not used for SOL 400 heat transfer.

NDAMP/NDAMPM:

In SOLs 129 and 159, numerical damping may be specified for the METHODS = "ADAPT" on the entry through the NDAMP entry in order to achieve numerical stability. A value of zero for NDAMP requests no numerical damping. The recommended range of NDAMP values is from 0.0 to 0.1.

SOL 400 Mechanical: In SOL 400, numerical damping may be specified for all METHODS (ADAPT, FNT, etc. on the entry. NDAMP and NDAMPM are two parameters that control the damping scheme and the associated dynamic operator. The equilibrium equation for the most general operator (the Generalized- α operator) is given by

$$M\ddot{u}_{n+1+\alpha_m} + C\dot{u}_{n+1+\alpha_f} + F_{n+1+\alpha_f}^{int} = F_{n+1+\alpha_f}^{ext}$$

where α_m is NDAMPM and α_f is NDAMP. In the previous equation, a typical quantity $F_{n+1+\alpha}$ is given by the expanded form

$$F_{n+1+\alpha} = (1 + \alpha)F_{n+1} - \alpha F_n$$

Depending on the values of NDAMP and NDAMPM, the equilibrium equations can reduce to the HHT- α scheme (NDAMPM = 0) or the WBZ- α scheme (NDAMP = 0) or the Generalized- α scheme (NDAMPM \neq 0, NDAMP \neq 0). For the HHT- α scheme, NDAMP can vary in the range of $-0.33 \leq NDAMP \leq 0.0$. For the WBZ- α scheme, NDAMPM can vary in the range of $0.0 \leq NDAMPM \leq 1.0$. For the Generalized- α scheme, NDAMP can vary in the range of $-0.5 \leq NDAMP \leq 0.0$ and NDAMPM in the range of $-0.5 \leq NDAMPM \leq 1.0$.

For problems involving no contact, the HHT- α is used in SOL 400 with default NDAMP = -0.05 and NDAMPM = 0.0 except that the model is linear, or no mass and no damping matrix. In this case, the defaults are NDAMP = 0.0 and NDAMPM = 0.0. For problems involving contact, the WBZ- α scheme is used in SOL 400 with default NDAMPM = 1.0 and NDAMP = 0.0

SOL 400 Heat Transfer: Numerical damping may be specified through the value of NDAMP only. NDAMPM is not used. NDAMP can be varied in the range of (-2.414,0.414). At these outer limits, the transient scheme reduces to the Backward-Euler method. Any value that is outside this range is automatically reset to the closest outer limit. For NDAMP = 0.0, the transient scheme reduces to the Crank-Nicholson scheme. The default value of NDAMP is -0.05.

SOL 400 Thermo-Mechanical Coupled Analysis: If NDAMP and NDAMPM are unspecified, then the default that is appropriate for the particular sub-step is used. For e.g., for a coupled transient contact problem, NDAMP = -0.05 during the heat transfer sub-step and NDAMP = 0.0, NDAMPM = 1.0 during the mechanical sub-step. If the values of NDAMP and NDAMPM are specified by the user in the Bulk Data

Section, then these values would be applicable for both the thermal and mechanical sub-steps. To make the values of the NDAMP/NDAMPM physics-dependent, the user can define the values under each sub-step of the Case Control Section. E.g.,

SUBSTEP 1
ANALYSIS=HTRAN
PARAM,NDAMP,-2.414

SUBSTEP 2
ANALYSIS=NLTTRAN
PARAM,NDAMP,-0.05
PARAM,NDAMPM,0.0

NEWMARK

Default = NO

See CQC under [SCRSPEC, 987](#).

NFDOPTS

Default = 2

PARAM, NFDOPTS, 2 computes frequency dependent matrices at master frequencies. In general, master frequencies via MFREQ in case control section of input deck is less than the number of forcing frequencies. Hence, PARAM, NFDOPTS, 2 can improve performance as well as reduce disk space requirement for large frequency response jobs. With PARAM, NFDOPTS, 1, frequency dependent matrices are computed at forcing frequencies.

NHPLUS

Default = 20

In nonlinear harmonic response analysis, in order to avoid aliasing in the calculation of the Fourier coefficients, a certain number of extra evaluation points are used. NHPLUS allows the number of extra points to be defined.

NINTPTS

Default = 10

NINTPTS requests interpolation over the NINTPTS elements closest to each grid point. NINTPTS=0 requests interpolation over all elements, which can be computationally intensive. Related parameters include BIGER, CURV, NUMOUT, OG, OUTOPT, S1G, S1M, S1AG and S1AM.

NLAYERS

Default = 5 for CQUAD4 and CTRIAR, 7 for CQUADR and CTRIAR, 3 for CAXISYM (minimum=1, maximum=11) (SOLs 106, 129, 153, 159, and 400)

NLAYERS is used in material nonlinear analysis only and is the number of layers used to integrate through the thickness of CQUAD4, CTRIA3 CQUADR, CTRIAR and CAXISYM elements with isotropic material properties. Set NLAYERS=1 for efficiency if no bending is selected (MID2=0 or -1 on all PSHELL entries). Do not specify NLAYERS=1 if MID2 is greater than zero. A larger value of NLAYERS will give greater

accuracy at the cost of computing time and storage requirements. For CQUADR and CTRIAR, the maximum is 11.

NLHTLS

Default = 0

See Remarks under Case Control command, [TSTRU \(Case\), 632](#).

NLHTOL

Default = 1.0E-5

During the iteration procedure of nonlinear harmonic response, the norm of the residual load vector for the current step is divided by the norm of the residual load vector for the previously converged step; this value is then compared with NLHTOL. If the value is smaller than NLHTOL, the system is assumed to have converged.

NLHTWK

Default = 1.0E-5

In nonlinear harmonic response analysis, if convergence is not obtained, a line search procedure is initiated to calculate a scaling factor for the displacement vector from which updated nonlinear loads are subsequently calculated. If the solution fails to converge more than 5 times in succession a new trial displacement vector is calculated using a push off factor the size of which is defined by NLHTWK. See also parameter [MXICODEO, 955](#).

NLMAX

Default = 60

The number of suspected massless mechanisms is determined from the number of high ratio messages. If this number exceeds NLMAX, the number of trial massless mechanisms is reduced to NLMAX. This avoids an expensive debug run when there may be thousands of massless mechanisms due to systematic modeling error, such as having CONM2 elements on many grid points for which structural elements have been left out through oversight. The value of this parameter may be increased on initial debug runs where many high ratio DOFs may be present, and the user prefers to see them all at once, rather than on successive runs where only a part are removed at one time. See also [MECHFIX, 898](#).

NLMIN

Default = 10

In the case of only one or a few high ratio DOFs more MMs may be present. More trial MM vectors are used, and those that do not indicate true problems are discarded. A smaller value could be considered on a stable model undergoing small modeling changes. See also [MECHFIX, 898](#).

NLPACK

Default=100

NLPACK is used in the nonlinear solution sequence SOL 400 only. The SOL 400 will pack output data for NLPACK output time steps and restart data for the last time step as a single data package. For example, if NLPACK=100 (the default), then one data package have output data for 100 output time steps and restart

data for the last time step. Later usage such as restart or initial condition for later step can be performed only at data package boundaries. If NLPACK= -1, all output data for a STEP and restart data for end of the STEP are grouped into a single package. In this case, the restart can be performed only at STEP boundaries. NLPACK=0 is illegal. If NLPACK=1, each package of data on the database includes output data for one output time step and restart data. In this case, restart can be performed at every output time step. Please note that the output time step is controlled by the NO field on the Bulk Data entry, or NO field of FIXED time step control as well INTOUT field of ADAPT time step control in NLSTEP Bulk Data entry. NLPACK also influences the intermediate output selected by NLOPRM.

Please note that NLPACK is based on the number of the output load steps (concerning INTOUT in NLSTEP, and so on), but not the number of the calculated load steps. For example, one NLTRAN analysis has NLSTEP as:

```
NLSTEP, 900, 0.2
, fixed, 2000, 20
, mech, u
```

It has total 2000 increments, and asks output every 20 increments. Therefore, total output is 100 time step. In NASTRAN, default value of NLPACK for NLTRAN is 100, in this model, therefore, NASTRAN will write the results to OP2 after collecting 100 output, i.e., this model will write results to OP2 only one time. With intermediate output request, only one OP2 file will be created.

If NLPACK=1, NASTRAN will write results to OP2 for every output request. In this model, NASTRAN will write results to OP2 for every output request, i.e., writing 100 times in this model. With "intermediate output request, 100 OP2 files will be created.

If NLPACK=2, NASTRAN will write results to OP2 for every two output requests. In this model, NASTRAN will write results to OP2 for every two output request, i.e., writing 50 times in this model. With "intermediate output request, 50 OP2 files will be created.

NLTOL

Default = $\begin{cases} 2 & (\text{SOL 106 and MD Nastran SOL 400}) \\ 0 & (\text{SOL 153, nonlinear heat transfer}) \end{cases}$

NLTOL establishes defaults for the CONV, EPSU, EPSP and EPSW fields of NLPARM Bulk Data entry for nonlinear static and heat transfer analysis according to the following table:

NLTOL	Level of Accuracy
0	Very high
1	High
2	Engineering Design
3	Preliminary Design

See Remark 16. of the NLPARM entry for further details and corresponding default NLPARM field values.

NMLOOP

Default = 0

In SOL 106 nonlinear static analysis, normal modes will be computed with the updated nonlinear stiffness if PARAM,NMLOOP is requested in the subcase section, it must be for all subcases. The nonlinear normal modes will be computed at the last iteration loop of each subcase in which a METHOD command appears.

NMNLFRQ

Default = 0.0

PARAM, NMNLFRQ, Real_Value allows users to select material or property values for frequency dependent materials or properties at a desired frequency other than the “nominal” values specified on the MATi or PBUSH, PCOMP, PCOMPG, PDAMP, or PELAS entries.

If there is only CBUSH frequency dependency and NO other type of material dependency, then Nastran will not use/update the CBUSH properties at the specified NMNLFRQ.

NOCOMPS

Default = +1

NOCOMPS controls the computation and printout of composite element ply stresses, strains and failure indices. If NOCOMPS = 1, composite element ply stresses, strains and failure indices are printed. If NOCOMPS = 0, the same quantities plus element stresses and strains for the equivalent homogeneous element are printed. If NOCOMPS=-1, only element stresses and strains are printed. MAT1, MAT2, or MAT8 entries with MIDs>99999999 run the risk of conflicting with internally generated MAT2 entries when composites are present and should be avoided. Even if no composites are present, MIDs>99999999 signal to shell element stress computations that no STRESS is to be computed. In the case of MIDs>99999999 if the user desires stress calculations for shell elements, even if no composites are present, then PARAM, NOCOMPS, -1 is required.

Homogenous stresses are based upon a smeared representation of the laminate's properties and in general will be incorrect. Element strains are correct however.

NOCOMPS only supports conventional elements, not advanced elements. For advanced nonlinear element, only ply stress/strain will be output.

NODCMP

Default = 0

For some type of nonlinear transient problems, including heat transfer, the decomposition of the solution matrix is not required. In order to increase efficiency, NODCMP is created to determine whether the solution matrix will be decomposed. If NODCMP=0, the solution matrix will be decomposed. If NODCMP=1, the solution matrix will NOT be decomposed. In this case, MAXBIS and DJUST on the Bulk Data entry, NLPARM, must be set zero on the Bulk Data entry. NODCMP is available for SOL 129 and 159 only.

NOELOF

Default = -1

If NOELOF > 0, then the grid point forces (GPFORCE Case Control command) are computed along the edges of the two-dimensional elements. The default value will suppress this output.

NOELOP

Default = -1

If NOELOP > 0, then the sum of the grid point forces (GPFORCE Case Control command) are computed parallel to the edges of adjacent elements. The effect of CBAR element offsets is not considered in the calculation of the forces. The default value will suppress this output.

NOGPF

Default = 1

NOGPF controls the printout of grid point forces (GPFORCE Case Control command). NOGPF > 0 specifies that the grid point forces are to be printed. NOGPF < 0 suppresses the printing of the grid point forces.

NOMSGSTR

Default = 0

If NOMSGSTR = -1, the MSGSTRES module will be skipped even though Bulk Data entries provided for it. See Part III of the *MSGMESH Analyst Guide* for a discussion of MSGSTRESS.

NONCUP

Default = -1

In SOLs 111 and 112, NONCUP selects either a coupled or uncoupled solution algorithm in the modal dynamic response analysis. This parameter has the following meanings in both solution sequences:

- | | |
|--------------------------|---|
| NONCUP = -1
(Default) | Use the uncoupled solution algorithm if there are no off-diagonal terms in any of the modal matrices (MHH, BHH and KHH); otherwise use the coupled solution algorithm. However, in order to ensure correct results, the coupled solution algorithm is used when large mass enforced displacement, large mass enforced velocity or SPC/SPCD enforced motion is employed in SOL 112, regardless of the existence of off-diagonal terms in the modal matrices. |
| NONCUP > -1 | Use the coupled solution algorithm unconditionally |
| NONCUP = -2 | Use the uncoupled solution algorithm unconditionally |

User Information Message 5222 indicates which solution algorithm is used in the analysis.

NQSET

Default = 0

If NQSET > 0, and the PARAM entry is in Case Control, all part superelements that do not contain QSET entries, or are not referenced by SENQSET entries in the main Bulk Data Section, have NQSET generalized coordinates assigned to them. These QSET variables are functionally equivalent to those generated by SENQSET entries.

NUMOUT, NUMOUT1, NUMOUT2, NUMOUT3

See S1, S1G, S1M.

OELMOPT

Integer; Default=3.

PARAM,OELMOPT selects the class of elements to include in the search for grid points:

- 0 All regular elements but not PLOTEL and rigid elements.
- 1 All regular elements and PLOTEL.
- 2 All regular elements and rigid elements.
- 3 Both 1 and 2 (default).

OELMSET

Integer; Default = 0

Identification number of a Case Control command SET definition. The members of the specified SET represent the identification numbers of the finite elements that are to be retained in the “reduced” op2 file element connection data block.

OG

Default=0

See CURV.

OGEOOM

Default = YES

See POST < 0.

OGRDOPT

Integer; Default = 1

Selects the method used to create the set of grid points retained in the reduced grid point geometry data block. The default simply uses the set of grid point IDs listed in the OGRDSET Case Control SET. Set consistency is checked. OGRDOPT=2 uses the list of grid point IDs that are connected to elements in the OELMSET Case Control SET. OGRDOPT=3 merges the contents of the OGRDSET Case Control SET with the contents of the grid point list connected to the elements in the OELMSET Case Control SET. There is no consistency check for OGRDOPT=2 or OGRDOPT=3. OGRDOPT=0 turns the SET consistency check off altogether. For this case, the grid points retained are those specified in the OGRDSET SET and the elements retained are those specified in the OELMSET SET. See also PARAM,OELMOPT description.

OGRDSET

Integer; Default = 0

Identification number of a case control command SET definition. The members of the specified SET represent the identification numbers of the grid points that are to be retained in the “reduced” op2 file grid geometry data block.

OLDSEQ

Default =
$$\begin{cases} -1 & \text{for non-iterative-distributed-parallel solutions} \\ 5 & \text{for iterative solutions using distributed parallel methods} \\ & (\text{NASTRAN ITER}=1 \text{ and system}(231)>0) \\ 6 & \text{if SUPER}=2 \end{cases}$$

OLDSEQ selects from the following options for resequencing:

- 1 No resequencing is performed.
- 1 Use the active/passive option.
- 2 Use the band option.
- 3 For the active/passive and the band option select the option giving the lowest RMS value of the active columns for each group of grid points.
- 4 Use the wavefront (Levy) option.
- 5 Use the Gibbs-King option even if the CPU estimate is higher than for nonsequencing.
- 6 Use the automatic nested dissection option even if the CPU estimate is higher than for no resequencing. See the following SUPER=2 description.
- 8 Semiautomatic selection. The program will compute estimates for two options that are suitable for the decomposition method selected by the PARALLEL and SPARSE keywords on the NASTRAN statement and select the option with the lowest estimate. The following table shows the suitable options for each decomposition method.

Decomposition Method	Suitable Options
regular	1 and 4
parallel	2 and 5
sparse	6 and 7

- 9 The extreme partitioning method is used to partition the model into domains.
- 10 The METIS partitioning method is used to partition the model into domains.
- 11 The MSCMLV partitioning method is used to partition the model into domains.

Notes:

1. The model partitioning options make sense only when running with the DOMAINSOLVER command in the Executive Control Section. For DOMAINSOLVER (PARTOPT=GRID), param,oldseq,9 is the default. For all other DOMAINSOLVER options, the default is param,oldseq,11.
2. The wavefront option does not support superelement resequencing or starting nodes. Also note that the automatic nested dissection option uses starting nodes only to establish the root of the initial connectivity tree.

If the value of OLDSEQ is changed in superelement analysis, an SEALL=ALL restart is required.

PARAM,FACTOR is used to generate the sequenced identification number (SEQID) on the SEQGP entry as follows:

$$\text{SEQID} = \text{FACTOR} * \text{GRP} + \text{SEQ}$$

where:

SEQ = generated sequence number

GRP = group sequence number

If GRP=0, use GRP(MAX)+1 where GRP(MAX) is the largest group sequence number in the database.

PARAM,MPCX controls whether the grid point connectivity created by the MPC, MPCADD, and MPCAX entries and/or the rigid element entries (e.g., RBAR) is considered during resequencing:

- 1 Do not consider the connectivity of the MPC, MPCADD, MPCAX, or rigid element entries.
- 0 Consider the connectivity of the rigid element entries only. (Default).
- >0 Consider the connectivity of the rigid element entries and the MPC, MPCADD, and MPCAX entries with the set identification number set to the value of this parameter.

PARAM,SEQOUT controls the output options as follows:

- 0 Do not generate any printed or punched output for the new sequence (Default).
- 1 Print a table of the internal/external sequence in internal order.
- 2 Write the SEQGP entries to the PUNCH file.
- 3 Perform SEQOUT=1 and 2.

PARAM,START specifies the number of the grid points at the beginning of the input sequence. The input sequence will be the sorted order of the grid point numbers including the effect of any SEQGP entries input by the user. A single SEQGP entry can be input to select the starting point for the new sequence. Otherwise, the first point of lowest connectivity will be used as the starting point.

If PARAM,SUPER<0, all grid points from the connection table that are not part of the group currently being processed are deleted. This option provides for sequencing only the interior points of a superelement. If any

superelements are present, the residual structure is not resequenced. If all of the grid points are in the residual structure, they are resequenced.

If PARAM,SUPER=0 or 1, all grid points in the connection table are considered. This option provides for the recognition of passive columns.

If PARAM,SUPER=2, then all points that are connected to multipoint constraints (via MPC entries) or rigid elements (e.g., the RBAR entry) are placed in a special group at the end of the sequence. This option also forces OLDSEQ=6 and may not be selected with other values of OLDSEQ. This option is intended primarily for models that have many active columns due to MPCs or rigid elements; e.g., a model with disjoint structures connected only by MPCs or rigid elements.

OLDWELD

Default = NO

By default, the program uses new formulations for connector elements (CFAST, CSEAM and CWELD). To acquire identical connector results as those from previous versions before v2017.1, specify OLDWELD=YES.

OMACHPR

Default = NO

See POST < 0.

OMAXR

Default = $2 \cdot \text{BUFFSIZE}$

OMAXR specifies the maximum record length of data written by the OUTPUT2 module under PARAM,POST,<0 and PARAM,OPTEXIT,-4. BUFFSIZE is a machine-dependent value defined in the *MSC Nastran Configuration and Operations Guide*. For further information, see the OMAXR parameter description under the OUTPUT2 module description in [MSC Nastran DMAP Programmer's Guide](#).

OMID

Default = NO

To print or punch the results in the material coordinate system, set the parameter OMID to yes. Applicable to forces, strains, and stresses for TRIA3, QUAD4, TRIAR6, and QUAD8. This is available for linear analysis only. Other elements and outputs are not supported. This capability is not supported by post-processors (xdb and op2 output are not changed), by grid point stress output that assumes output is in element coordinate system, random response analysis, heat transfer analysis, stress sorting, MAXMIN Case Control, composite elements, MONPNT3 Bulk Data and external superelement data recovery in the assembly run.

OMSGLVL

Integer; Default = 0

Set consistency check error message severity flag. The default causes FATAL messages to be generated if the grid set is not consistent with the element-related grid point set and the job is terminated. If OMSGLVL=1, the FATAL messages are reduced to WARNINGS and the job is allowed to continue.

OPCHSET

Integer; Default = 0

SET punch request flag. If OPCHSET=1, then the list of grid points used to reduce the grid point geometry data block will be punched in case control SET definition format.

OPGEOM

Default = -1

OPGEOM > -1 prints the aerodynamic set definitions for all degrees-of-freedom. To print structural degree-of-freedom sets, please see [USETPRT](#).

OP2GM34

Default = TRUE

By default, the GEOM3 and GEOM4 are written in OP2 file. To avoid GEOM3 and GEOM4 output, set PARAM, OP2GM34, FALSE.

OPGTKG

Default = -1

OPGTKG > -1 prints the matrix for the interpolation between the structural and aerodynamic degrees-of-freedom.

OPPHIB

Default = -1

In the flutter (SOLs 145 and 200) and aeroelastic (SOLs 146 and 200) solution sequences, OPPHIB > -1 and a DISPLACEMENT request in the Case Control Section will output the real vibration modes with the structural displacement components transformed to the basic coordinate system.

OPPHIPA

Default = -1

In the flutter (SOLs 145 and 200) and the dynamic aeroelastic (SOL 146) solution sequences, OPPHIPA > -1 and a DISPLACEMENT command in the Case Control Section will output the real vibration modes at all degrees-of-freedom, including the aerodynamic degrees-of-freedom in the global coordinate system. Use PARAM,OPPHIPB to output in the basic system.

OPTEXIT

Default = 0

In SOL 200, especially during the checkout of the analysis model and the design optimization input data (design model), it may be desirable to exit the solution sequence at certain points before proceeding with full optimization. OPTEXIT may be set to values of 1 through 7 and -4. The DSAPRT Case Control command overrides the specification of PARAM,OPTEXIT,4, -4, or 7. The description of OPTEXIT values follow.

OPTEXIT Value	Description
0	Do not exit. Proceed with optimization.
1	Exit after the initialization of the analysis and design model but before finite element analysis begins.
2	Exit after finite element analysis and initial design response and shape basis vector processing.
3	Exit after design constraint evaluation and screening.
4	Exit after design sensitivity analysis and print the matrix of design sensitivity coefficients (DSCM2). This is equivalent to the DSAPRT (UNFORM,END=SENS) Case Control command.
-4	Exit after design sensitivity analysis and write the data blocks related to sensitivity coefficients (DSCM2 and DSCMCOL) to an external file using the OUTPUT2 and OUTPUT4 modules. This is equivalent to the DSAPRT (NOPRINT,EXPORT END=SENS) Case Control command. See related parameters ITAPE, IUNIT, and OMAXR.
5	Exit after the first approximate optimization of the design model.
6	Exit after the first update of the analysis model based on the first approximate optimization of the design model.
7	Compute and output design sensitivity coefficients at the end of normal program termination: hard convergence, soft convergence, or maximum design cycles. This is equivalent to the DSAPRT (UNFORM,START=LAST) Case Control command. If the final design is a discrete design optimization, no sensitivity is performed and the OPTEXIT=7 request is not honored.

OPTION

Default = ABS

See SCRSPEC.

OSETELE

Default = 2

See [AUTOGOUT, 793](#).

OSETGRD

Default = 1

See [AUTOGOUT, 793](#).

OSWELM

Default = -1

Offset for identification numbers of internally generated RBE3 rigid body elements (generated by all formats of CFAST and CSEAM as well as CWELD with formats ELEMID, GRIDID, ELPAT and PARTPAT). By default, OSWELM=-1, the numbering starts with SYSTEM(182) + 1. The default of system cell 182 is SYSTEM(182)=100,001,001. If the user defines OSWELM > 0, then the numbering starts with OSWELM + 1.

Active ONLY for "PARAM,OLDWELD,YES", for each CWELD element, a pair of RWELD constraint elements is generated if MSET=ON is specified, see the entry PWELD, 3329 for an explanation.

Active ONLY for "PARAM,OLDWELD,YES", in an SOL 400 nonlinear analysis this defines the offset for identification numbers of internally generated RBE3 rigid body elements (generated by CWELD and CFAST, all formats) and CONM2 mass elements (generated by CFAST when it has nonzero mass). The default behavior in SOL 400 is the same as in the linear solution sequences.

OSWPPT

Integer, Default = -1

Offset for internally generated grid identification numbers for connector elements (GA and GB for CWELD and CFAST if these are not specified and all auxiliary grids for CWELD, CFAST and CSEAM). By default, OSWPPT=-1, the numbering starts with SYSTEM(178) + 1. The default of system cell 178 is SYSTEM(178)=101,000,000. If the user provides OSWPPT > 0, then the numbering starts with OSWPPT + 1.

Active ONLY for "PARAM,OLDWELD,YES", for each CWELD or CFAST element, a pair of grid points GA and GB is generated internally if the formats ELEMID, GRIDID, ELPAT or PARTPAT are used and if no identification numbers for GA and GB are specified, see the entry CWELD, 1696 and CFAST, 1505 for a definition of the formats.

Active ONLY for "PARAM,OLDWELD,YES", in an SOL 400 nonlinear analysis this defines the offset for identification numbers of internally generated grids of connector elements CWELD and CFAST (GA and GB if these are not specified in the bulk data input and all auxiliary grids). The default behavior in SOL 400 is the same as in the linear solution sequences.

OUGCORD

See POST.

OUGSPLIT

Default=YES

By default, displacements, velocities, and accelerations are written to separate tables.

PARAM,OUGSPLIT,NO will combine these tables into a single table and is intended for third-party post-processors which are not able to read the separate tables. The value of OUGSPLIT does not affect the results in the f06 file.

OUNIT1

Default = Value of OUNIT2

For PARAM,POST,-1 and -2 defines the unit that geometry data blocks are output to. See PARAM,POST. This parameter should not be specified after BEGIN SUPER.

OUNIT2

Default = 12

For PARAM,POST,-1 and -2 defines the unit that results data blocks are output to. See PARAM,POST. This parameter should not be specified after BEGIN SUPER.

OUTOPT

Default = 0

See CURV.

PACINFID

Integer, no default.

This parameter is used only in data recovery operations for elements in acoustic field point mesh bulk data. PACINFID is the ID of the PACINF entry that contains the location of the pole used to determine element orientation. Data recovery operations require that the element surface normal vector point away from the pole location. This parameter is required only if the acoustic model references multiple PACINF bulk data entries and the pole locations on the PACINF entries are not coincidental.

PANELMP

Replaced by a keyword on the FLSPOUT Case Control command.

PATPLUS

Default = NO

PARAM,PATPLUS,YES may be used with PARAM,POST,0 to allow the user to also write data blocks GPFORCE, ESE, EKE, EDE, WETSENS, and ESE-(with PARAM,XFLAG,2) to a Fortran unit as is done under PARAM,POST,-1.

PATVER

Default = 3.0

See POST = -1.

PCOMPRM

Default = 0

PCOMPRM controls the computation and printout of composite element extreme ply stresses, strains, failure Indices and strength ratios. Note that extreme ply output encompasses ply stresses, ply strains, ply failure indices and ply strength ratios together. Punch output is not available for extreme ply results.

PCOMPRM = 0 (default) , STANDARD composite ply responses are output.

PCOMPRM = 1 STANDARD & EXTREME composite ply responses are output
 PCOMPRM = 2 ONLY EXTREME composite ply responses are output

PDRMSG

Default = 1

PDRMSG controls the printout of messages associated with deformed plots, including error messages.
 PDRMSG = 0 suppresses the printout. Contour values will not be displayed unless the default value is used.

PEDGE

Default = 0

Cubic edges of p-elements can be created with the FEEDGE Bulk Data entry by defining two vertex grids and two points in between. By default, the two points on an edge are moved to the parametric 1/3 and 2/3 locations of the edge. For PEDGE = 1 the points are not moved. MSC Patran V7 generates points so that adjacent edges are C1 continuous. These points should not be moved. Therefore, MSC Patran generates a Bulk Data Section with PARAM,PEDGE,1 if p-elements are in the model.

PEMFRIM

PARAM,PEMFRIM,1 is used to request full Reduced Impedance Matrix (RIM) from Actran, instead of lower-triangular RIM. By default (PARAM,PEMFRIM,0), Actran returns lower-triangular RIM. Note that PEMFRIM will increase disk space demand of PEM job. In addition, OOC field of ACPEMCP will cause Actran to generate full RIM when set to an integer greater than 1.

PEMMAPC

PEMMAPC is used to generate lists of GRD IDs of the coupling surfaces. 'PARAM,PEMMAPC,1' (default is 0) will produce following files in 'mapping_control' subdirectory of job submit directory.

For	SET ID	Generic File name
Structure GID coupled with all TRMCs	100000	<deckname>.<job seq>.x_nas.structure_component1.set
Fluid GID coupled with all TRMCs	200000	<deckname>.<job seq>.x_nas.fluid_component2.set
Structure GID coupled with TRIMID=xxxx only	y00018xxxx for GLUED or z00018xxxx for SLIDING	<deckname>.<job seq>.x_nas.structure_component1.trim_coupling.<xxxx>.<y or z>.set
Fluid GID coupled with TRIMID=xxxx only	Y00028xxxx for OPEN or Z00028xxxx for IMPER	<deckname>.<job seq>.x_nas.fluid_component2.trim_coupling.<xxxx>.<Y or Z>.set
TRIMID=xxxx GID coupled with structure	xxxx90001	<deckname>.<job seq>.x_nas.interface<xxxx>.surface.1.set
TRIMID=xxxx GID coupled with fluid	xxxx90002	<deckname>.<job seq>.x_nas.interface<xxxx>.surface.2.set

Guidelines for PARAM,PEMMAPC,1

1. PARAM,PEMMAPC,1' must be placed in the main bulk data section or under 'BEGIN BULK' to be effective.
2. Without 'PARAM,PEMNPART,n', PEMMAPC logic has a TRIM ID limit of less than 10000.
3. With 'PARAM,PEMNPART,n', PEMMAPC login has a TRIM ID limit of less than 100.

PEMNCOP4

PARAM,PEMNCOP4,1 (default) is used to request each Reduced Impedance Matrix (RIM) from Actran to be written into individual file. With PARAM,PEMNCOP4,0, Actran writes all RIMs into a single file. Note that PARAM,PEMNCOP4,1 can reduce footprint on disk space of large PEM jobs.

PEMNPART

PEMNPART can be utilized to partition a large trim component. The default value for PEMNPART is 1. 'PARAM,PEMNPART,n' will partition a large trim component into 'n' parts. The guidelines are:

1. PARAM,PEMNPART,n' must be placed under 'BEGIN TRMC=xx' to be effective for trim component 'xx'.
2. Each trim component can have its own 'PARAM,PEMNPART,n'
3. When value of 'n' for PEMNPART is greater than 1, new trim components are generated with ID equal to 'xx*100+<seq no> where <seq no> goes from 0 to n-1.
4. For no loss of accuracy, setting 'n' to a value equal to the number of disconnected parts of a large trim component.
5. For accepting loss of accuracy to run on a machine with limited memory, setting 'n' to a value larger than the number of disconnected parts of a large trim component.
6. 'PARAM,PEMNPART,n' with n>1 should be utilized only as an alternative for running PEM jobs with large trim component and insufficient memory on the computer. Other alternatives for running PEM job with large trim component and insufficient memory are:
 - a. Split large trim component into smaller trim components and/or
 - b. Acquire more memory for the computer
7. For a trim component with 'PARAM,PEMNPART,n' where n>1 and 'PARAM,TRMBIM,physical', data recovery will be skipped for all partitioned trim components.

PEMSKIN

'PARAM,PEMSKIN,1' is used for Actran to couple the trim bodies only to the surface of the cavity and will not search any node inside the cavities. The default value for PEMSKIN is 0 which allows Actran to target both surface and interior nodes of the cavities for trim bodies coupling.

PENFN

Default = 1.0e+5 (See LMFAC)

PERCENT

Default = 40

See CQC under [SCRSPEC, 987](#).

PH2OUT

Default = 0, SOL 400 only

For nonlinear solution sequence, SOL 400, in addition to the regular phase III output, the user can also request the phase II output. This is useful when the run is terminated abnormally before the phase III outputs are formatted and printed. The phase II output consists of all outputs requested by the Case Control commands in the input file and prints in sort1 format. If there's no PH2OUT, MSC Nastran outputs phase III outputs only. This is the regular output. If PH2OUT=1, MSC Nastran outputs phase II outputs only. In this case, there will be no output for the upstream superelements. If PH2OUT=3, MSC Nastran outputs both phase II and phase III outputs. In this case, some of the outputs for the residual structure may be redundant.

Case Control command, NLOPRM OUTCTRL, takes precedence over PH2OUT. In other words, when NLOPRM OUTCTRL is present, PH2OUT is no longer functional. NLOPRM OUTCTRL=SOLUTION is equivalent to PH2OUT=1 and NLOPRM OUTCTRL=(STD,SOLUTION) to PH2OUT=3.

PKRSP

Default = -1

If PKRSP=0, the magnitude of the output quantities at the time of peak acceleration of the modal variables is output. This option is available only for modal transient analysis.

PLTMSG

Default = 1

PARAM,PLTMSG,0 suppresses messages associated with undeformed plot requests, including error messages.

POST

Default = 9999999

If PARAM,POST,0, then the following parameters and discussion apply (not support in SOL 400):

The data blocks often used for pre- and postprocessing will be stored in the database and also converted, by the DBC module (see [MSC Nastran DMAP Programmer's Guide](#)), to a format suitable for processing by MSC Patran. These data blocks include input data related to geometry, connectivity, element and material properties, and static loads; they also include output data requested through the Case Control commands OLOAD, SPCF, DISP, VELO, ACCE, THERMAL, ELSTRESS, ELFORCE, FLUX, GPSTRESS, GPFORCE, ESE, GPSDCON, and ELSDCON.

The converted data is written to logical FORTRAN units, which may be assigned to physical files in the File Management Section. The FORTRAN unit numbers are specified by the parameters GEOMU, POSTU, and LOADU. By default, all data is written to the logical FORTRAN unit indicated by GEOMU. If LOADU > 0, static load data may be diverted to another unit indicated by LOADU. If POSTU > 0, then output data requested with the Case Control commands listed above will be diverted to the logical unit indicated by POSTU. See [Database Concepts in the MSC Nastran Reference Guide](#) for the procedure for assigning physical files.

By default, if converted data already exists on the files indicated by GEOMU, POSTU, and LOADU, then the DBC module will overwrite the old data. If this is not desirable, then PARAM,DBCOVWRT,NO must be entered. The parameters MODEL and SOLID may be used to store more than one model and solution in the graphics database. These parameters are not supported by MSC Patran.

PARAM,DBCDIAG > 0 requests the printing of various diagnostic messages from the DBC module (see [MSC Nastran DMAP Programmer's Guide](#)) during the data conversion. By default, no messages are printed.

If PARAM,PATPLUS,YES is specified along with PARAM,POST,0 then the PARAM,POST,-1, operation will also be performed.

If PARAM,POST,< 0 or 1, then the following parameters and discussion apply:

- PARAM,POST,1 outputs the appropriate files for the SimXpert and Patran programs and should be used to obtain results from the latest result data blocks from SOL 400 and other recent MSC Nastran developments such as NEF.
- PARAM,POST,-1 outputs the appropriate files for the Patran program (PARAM,POST,1 should be used for more recent versions of MSC Nastran > 2012).
- PARAM,POST,-2 outputs the appropriate files for the Siemens I-deas® program.
- PARAM,POST,-4 outputs the files indicated below along with OPHIG for the MSC_NF interface by LMS International.
- PARAM,POST,-5 outputs the files indicated in the table below along with LAMA and OPHG1 for the FEMtools interface by Dynamic Design Solutions.
- POST=-4 and -5 are intended for SOL 103 only.
- PARAM,POST,-6 outputs the files indicated below for Siemens Unigraphics®.
- PARAM,POST,-7 outputs the files for Free Field Technologies.

An OUTPUT2 file for FORTRAN unit 12 in binary format is automatically created in the same directory and with the same name as the input file and with the extension ".op2". For example, if the input file is fender.dat then the OUTPUT2 file will be called fender.op2.

An ASSIGN statement is required in the FMS Section only if neutral file format is desired as follows:

ASSIGN OP2='filename of FORTRAN file' FORM

Geometry data blocks are output with PARAM,OGEOM,YES (Default) and are written to a FORTRAN unit specified by PARAM,OUNIT1 (Default = OUNIT2) for POST = -1, -2, -4, and -6.

PARAM,OUNIT2K (Default = 91) specifies the unit number for KELM and KDICT with PARAM,POST,-5. PARAM,OUNIT2M (Default = 92) specifies the unit number for MELM and MDICT with PARAM,POST,-5. Note that PARAM,POST,-5 is not supported with DMP. See the following table for the specific geometry data blocks written for different values for POST.

See also the PARAM,POSTEXT description for additional data blocks written to the .op2 file.

POST					Geometry Data Block	Description
-1,1 or -7	-2	-4	-5	-6		
YES	NO	NO	NO	NO	GEOM1S, GEOM1VU	Grid Point Definitions (Superelement)
NO	YES	YES	NO	YES	CSTM	Coordinate System Transformations
NO	YES	YES	NO	YES	GPL	Grid Point List
NO	YES	YES	NO	YES	GPDT	Grid Point Definitions
NO	YES	YES	NO	NO	EPT	Element Properties
NO	YES	YES	NO	NO	MPT	Material Properties
YES	YES	YES	NO	NO	GEOM2	Element Definitions
YES*	YES	NO	NO	NO	GEOM3	Load Definitions
YES*	YES	NO	NO	NO	GEOM4	Constraint Definitions
YES	NO	NO	NO	NO	DIT	Dynamic Table Input
YES	NO	NO	NO	NO	DYNAMICS	Dynamic Loads Definition
NO	NO	YES	YES	NO	KDICT	Element Stiffness Dictionary
NO	NO	YES	YES	NO	KELM	Element Stiffness Matrices
NO	NO	YES	YES	NO	MDICT	Element Mass Dictionary
NO	NO	YES	YES	NO	MELM	Element Mass Matrices
NO	NO	NO	NO	YES	ECTS	Element Connections
YES	NO	NO	NO	NO	VIEWTB	View Element Table
YES	NO	NO	NO	NO	EDOM	Design Model Input
YES	NO	NO	NO	NO	GEOM2S, GEOM2VU	Same as GEOM2 for superelements
YES	NO	NO	NO	NO	CSTMS	Same as CSTM for superelements
YES	NO	NO	NO	NO	EPTS	Same as EPT for superelements
YES	NO	NO	NO	NO	MPTS	Same as MPT for superelements

* To avoid having GEOM3 and GEOM4 written if POST=-1, set PARAM,OP2GM34,FALSE.

PARAM,OMACHPR,NO selects the single precision format for GPDT, CSTM, and GEOM1.

PARAM,OMACHPR,YES selects the machine precision format. With PARAM,POST,< 0,
PARAM,OMACHPR,NO is the default. With PARAM,POST,1, PARAM,OMACHPR,YES is the default.

For PARAM,POST = -1 and -2, results data blocks are output to a FORTRAN unit specified by
PARAM,OUNIT2 (Default = 12). This parameter is allowed to vary between superelements. In buckling
solution sequence (SOL 105), a unique value of OUNIT2 should be specified for the buckling subcase. For
all other solution sequences or within a given analysis type, changes to PARAM,OUNIT2 between subcases
is not supported. See the POST Case Control command to change the unit number between subcases. See
also the related parameter OMAXR.

By default the displacements are output in the global coordinate system. To output in the basic coordinate system, specify PARAM,OUGCORD,BASIC.

PARAM,POST,-1 or 1: Results Data Blocks for Patran and SimXpert

By default, the following data blocks are output under PARAM,POST,-1. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, if PARAM,OQG,NO is specified, then the SPCFORCE output is not written to the OUTPUT2 file. It is recommended when processing large post files in Patran and SimXpert that the following environment variable setting is used when starting the session: DRANAS_NAST_MEM=2048MB.

PARAM,PATVER		Parameter Name	Case Control	Data Block Name	Description
< 3.0	≥ 3.0				
YES	YES	OQG	SPCFORCE	OQG1	Forces of single-point constraint
YES	NO	OUG	DISP	OUGV1PAT	Displacements in the basic coordinate system
YES	YES	OUG	DISP	OUGV1	Displacements in the global coordinate system
YES	NO	OES	STRESS	OES1	Element stresses (linear elements only)
YES	NO	OEF	FORCE	OEF1	Element forces or heat flux (linear elements only)
YES	YES	OEE	STRAIN	OSTR1	Element strains
YES	YES	OGPS	GPSTRESS	OGS1	Grid point stresses
YES	YES	OESE	ESE	ONRGY1	Element strain energy
YES	YES	OGPF	GPFORCE	OGPFB1	Grid point force balance table
NO	YES	OEFX	FORCE	OEF1X	Element forces with intermediate (CBAR and CBEAM) station forces and forces on nonlinear elements
NO	YES	OESX	STRESS	OES1X	Element stresses with intermediate (CBAR and CBEAM) station stresses and stresses on nonlinear elements
NO	YES	OPG	OLOAD	OPG1	Applied static loads
NO	YES	OCMP	STRESS	OES1C	Ply stresses
NO	YES	OCMP	STRAIN	OSTR1C	Ply strains
NO	YES	none	DISP SPCFORCE FORCE STRESS STRAIN	OUPV1 OQP1 DOEF1 DOES1 DOSTR1	Scaled Response Spectra
			none	LAMA	Nonlinear Buckling
NO	YES	none	DISP OLOAD	OCRUG OCRPG	
NO	YES	none	NLSTRESS	OESNLXR	Nonlinear static stresses

PARAM,PATVER		Parameter Name	Case Control	Data Block Name	Description
< 3.0	≥ 3.0				
NO	YES	none	BOUTPUT	OESNLBR	Slideline stresses
NO	YES	none	NLLOAD	OPNL1	Nonlinear loads
NO	YES	none	STRESS	OESNLXD	Nonlinear transient stresses
NO	YES	none	none	ERRORN	p-element error summary table

PARAM,POST,-2: Results Data Blocks for Siemens I-deas®

By default, the following data blocks are output under PARAM,POST,-2. By default, the displacements are output in the basic coordinate system. To output in the global coordinate system, specify PARAM,OUGCORD,GLOBAL. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, if PARAM,OQG,NO is specified, then the SPCFORCE output is not written to the OUTPUT2 file.

PARAMeter Name	Case Control	Results Data Block Name	Description
OQG	SPCFORCE	OQG1	Forces of single-point constraint
OUG	DISPLACE	BOUGV1	Displacements in the basic coordinate system
		BOPHIG	Eigenvectors in the basic coordinate system
		OUGV1	Displacements in the global coordinate system
		TOUGV1	Grid point temperatures
OES	STRESS	OES1	Element stresses (linear elements only)
OEF	FORCE	OEF1	Element forces (linear elements only)
	FLUX	HOEF1	Element heat flux
OEE	STRAIN	OSTR1	Element strains
OESE	ESE	ONRGY1	Element strain energy
OCMP	STRESS	OEFIT	Failure indices
	STRESS	OES1C	Ply stresses
	STRAIN	OSTR1C	Ply strains
OUMU	ESE	LAMA	Eigenvalue summary
		ONRGY2	Element strain energy
OEFX	FORCE	OEF1X	Element forces (nonlinear elements only)
OESX	STRESS	OES1X	Element stresses (nonlinear elements only)
none	none	ODELBGPD	Shape optimization geometry changes

[PARAM, POST, -4: Results Data blocks for LMS International/MSC_NF](#)

By default, the following data blocks are output under PARAM,POST,-4. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, PARAM,OUG,NO requests that eigenvectors not be written to the OUTPUT2 file.

PARAMeter Name	Case Control	Data Block Name	Description
OUG	DISPLAC	OPHIG	Eigenvectors in the global coordinate system.

[PARAM, POST, -5: Results Data blocks for Dynamic Design Solutions/FEMtools](#)

By default, the following data blocks are output under PARAM,POST,-5. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, PARAM,OUG,NO requests that eigenvectors not be written to the OUTPUT2 file. PARAM,OUNIT2O (Default51) specifies the unit number of the OUTPUT2 file.

PARAMeter name	Case Control	Data Block Name	Descriptions
OUG	DISPLAC	OUGV1	Eigenvectors in the global coordinate system.
		LAMA	Eigenvalue summary.

[PARAM, POST, -6: Results Data Blocks for Siemens Unigraphics®](#)

By default, the following data blocks are output under PARAM,POST,-6. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, PARAM,OUG,NO requests that displacements not be written to the OUTPUT2 file.

PARAMeter Name	Case Control	Data Block Name	Description
OQG	SPCFORCE	OQG1	Forces of single-point constraints
OUG	DISPLAC	OUGV1	Displacements
OES	STRESS	OES1	Element stresses
OEF	STRESS	OEF1	Element forces
OEE	STRAIN	OSTR1	Element strains
OESX	STRESS	OES1X	Element stresses with intermediate station stresses and stresses on nonlinear elements
OEFX	STRESS	OEF1X	Element forces with intermediate station forces and forces on nonlinear elements
OPG	OLOAD	OPG1	Applied static loads
none	none	DVPTAB	Designed property table

PARAMeter Name	Case Control	Data Block Name	Description
none	none	OPTPRMG	Optimization parameters
none	none	PROPO	Final element properties
none	none	DBCOPT	Optimization summary data

PARAM,POST,-7

PARAM,POST,-7 Results Data Blocks for Free Field Technologies.

Except eigenvectors, all other output data blocks are the same as PARAM,POST,-1. For eigenvectors, the matrix form is written to the OUTPUT2 file.

POSTEXT

Default = NO

Under PARAM,POST,-1 and -2, and if PARAM,POSTTEXT,YES is specified, then the following data blocks are also written to the .op2 file specified by PARAM,OUNIT2.

Data Block Name	PARAM POST	Description
FRL	-1, -2	Frequency response list (modal frequency response only).
BHH	-1, -2	Modal damping matrix (modal frequency response only).
KHH	-1, -2	Modal stiffness matrix (modal frequency response only).
BGPDT	-1	Basic grid point definition table.
PVT0	-1, -2	User parameter value table
CASECC	-1	Case Control table
EQEXIN(S)	-1	Equivalence external to internal grid ID table
CLAMA	-1, -2	Complex eigenvalue table
OEDE1	-1, -2	Element energy loss output table
OEKE1	-1, -2	Element kinetic energy output table
OUGV2	-1, -2	Displacement output table in SORT2
PSDF	-1, -2	Power spectral density table
OGPWG	-1, -2	Grid point weight generator output table
TOL	-1, -2	Time output list
OPHSA	-1, -2	Solution set eigenvectors (modal frequency response only)
LAMA	-1	Eigenvalue summary table
ONRGY2	-1	Element kinetic energy (obsolete)
PSDFH	-1, -2	Power spectral density table for modal coordinates
DSCM2	-1, -2	Design sensitivity coefficient matrix
DSCMCOL	-1, -2	Design sensitivity parameters table

POSTADF

Default = NO

This parameter is used to export the results of frequency analysis responses and participation factors by PFMODE, PFPANEL, and PFGRID in ADF format file. This exportation is requested by setting this parameter to YES. The unit information of the exported functions should be defined by DTI, UNITS statement for this function (see ADAMSMNF statement). If not defined, SI unit will be assumed with the warning message. The ADF file is defined as input data name with the extension “afu” automatically. Alternatively the name can be defined with ASSIGN ADFFILE= statement in FMS section. For the participation factor output, FILTER=0. option should be applied in PFMODE and PFPANEL.

POSTU

Default = -1

See POST=0.

PREFDB

Default = 1.0

See ACOUT.

PRGPST

Default = NO

PRGPST is replaced by the PRINT keyword on the AUTOSPC Case Control command. The specification of the AUTOSPC command overrides the specification of PARAM,PRGPST.

PRINT

Default = YES

PARAM,PRINT,NO suppresses the automatic printing of the flutter summary in flutter analysis.

PROUT

Default = -1

PARAM,PROUT,-1 suppresses execution and printout from the ELTPRT module. PARAM,PROUT,-1 prints a list of all elements sorted on EID and summary tables giving the range of element identification numbers for each element type.

ELTPRT will provide an information message when it detects any duplicate element ID. For the case of CONV and CHYBDi elements, duplicate element IDs are by design. The information message provided by ELTPRT can be safely ignored for these elements.

PRPA

Default = 1.0E37

PRPJ

PRPA and PRPJ control the printout of intermediate load matrices for diagnostic purposes during superelement assembly. If the value of PRPA (or PRPJ) is positive, all terms larger in magnitude than the

value are printed. If the value of PRPA (or PRPJ) is negative, all terms smaller in magnitude than the value are printed. The default value requests no printout. PARAM,IRES,1 must be present for these parameters to be effective. The PA matrix contains the internal loads transmitted to the downstream superelement. The PJ matrix contains external loads applied on the superelement; that is, it has the same content as the data produced by the Case Control command OLOAD. All of this data may be obtained on restart using the SELR Case Control command option. A related parameter is IRES.

PRPHIVZ

Default = 1.0E37

PRPHIVZ controls the printout of the PHIVZ matrix that contains the component mode eigenvectors of the model. It includes all degrees-of-freedom with motion except the m-set, eliminated for multipoint constraints. The FAPPROX matrix contains the square root of the diagonal terms of the generalized stiffness matrix divided by 2π . For fixed-boundary solutions, it is a good approximation for the natural frequencies of the component. For free- or mixed-boundary solutions, it is of a lower value than the natural frequencies. All terms larger than PRPHIVZ in both matrices will be printed (i.e., PARAM,PRPHIVZ,0.0 causes all terms to be printed).

PRTMAXIM

Default = NO

PRTMAXIM controls the printout of the maximums of applied loads, single-point forces of constraint, multipoint forces of constraint, and displacements. The printouts are titled “MAXIMUM APPLIED LOADS”, “MAXIMUM SPCFORCES”, “MAXIMUM MPCFORCES”, and “MAXIMUM DISPLACEMENTS”.

PRTRESLT

Default = YES

PRTRESLT controls the printout of the resultants of applied loads, single-point forces of constraint, and multipoint forces of constraint. The printouts are titled “OLOAD RESULTANTS”, “SPCFORCE RESULTANTS”, and “MPCFORCE RESULTANTS”.

PSENPCH

Default = NO, for SOL 200 job with PART SEs only.

Setting PSENPCH to YES causes updated Bulk Data entries of a PART SE for a design cycle punched to a separate file named as follows

JOBNAME_psex_{xx}_yy.pch – where xx is the PART SE ID and yy is the design cycle.

PVALINIT

Default = 1

Starting p-value in a p-adaptive analysis restart.

Q

Default = 0.0

Q specifies the dynamic pressure. Q must be specified in aeroelastic response analysis (SOLs 146), and the default value will cause a User Fatal Message.

RADMOD

Default = YES

This parameter only applies to heat transfer solution sequences for SOLs 153, 159 and 400. The parameter, RADMOD, only affects radiation heat transfer problems. The radiation matrix is modified to avoid the temperature overshoot with a coarse mesh. If the user wants the program to skip this operation (modify the radiation matrix) then insert PARAM,RADMOD,NO.

RBSETPRT

Default=0

This parameter controls the printout of rigid element IDs that are included in the analysis. Setting PARAM,RBSETPRT to a value not equal to zero activates the printing of rigid element IDs. Note that this function works in conjunction with the rigid element selection via MPC=n in case control and SET3,n,RBxx in bulk data.

RESLOPT

Default = 8

RESLOPT's default value provides component-level force summary output for model checkout (PARAM, CHECKOUT, YES), loads generation (OLOAD output), and SPC and MPC forces. Setting RESLOPT to a value of 1 produces abbreviated output formats only.

RESVEC

This parameter and the related parameters RESVINER, RESVSO, RESVSE, and RESVSLI are obsolete or replaced by options on the RESVEC Case Control command.

RFORSET

Default = 0

A PARAM, RFORSET, n where n is a Bulk Data SET1 entry, allows the user to apply rotational force loading (RFORCE) to just a portion of the structure. RFORSET like RFORCE loading is global and is computed once at the beginning of the analysis. The Bulk Data SET1 n defines the GRIDS to which the rotational force loading is to be applied. RFORSET can appear in either the **Bulk Data** or **ABOVE** or **IN the 1st Subcase**. A RFORSET appearing in any other Subcase will be ignored and may cause incorrect results. A RFORSET appearing in or above the first subcase takes precedence over a RFORSET appearing in Bulk Data. See also the companion GRAVSET entry. If there is no RFORCE loading called out, then PARAM, RFORSET, n needs to be removed from the run.

RHOCP

Default = 1.0

This parameter represents a scale factor used in the computation of ERP in units of dB. See the ERPPREFDB parameter. This parameter may alternatively be set using the ERP Case Control command.



RIMINTP

RIMINTP can be used to select the interpolation method for Reduced Impedance Matrix(RIM) for PEM and ACTRIM. The input options for RIMINTP are LINEAR and LOG10. The default is LINEAR.

RKSHEME

Default = Remark 1., SOL 700 only

Defines the type of time-integration scheme used in the Riemann solution-based Euler solvers.

Format:

PARAM,RKSHEME,NUMBER

Example:

PARAM,RKSHEME,3

NUMBER Number of Runge-Kutta stages. (Integer > 0)

Remarks:

1. This parameter can be used in combination with the original Roe solver and the improved full 2nd order fluid- and gas Euler solver. The default number of stages depends on the spatial accuracy of the solution scheme. One stage is used for first order spatial accuracy, and three stages for second order spatial accuracy.
2. It is recommended that the user leave the setting to the default values as defined depending on the selected spatial accuracy of the solution.
3. It has been found that in some cases with fluid flows, it may be necessary to manually limit the time step to a fixed and lower value than the scheme determines. These occasions are noticeable when you view a contour plot of the pressure. The user may find that the pressure field looks like a checkerboard. This is then caused by a numerical instability due to a time step that is too large for the fluid flow. The face fluxes are always correct but the element values get decoupled due to the time integration instability. Lowering the time step to about half the maximum value the scheme determines solves this problem. Please note that this significantly slows down the computation! Since this problem rarely occurs we have chosen not to automatically limit the time step to a lower value for performance reasons. However, the user needs to be aware that the analysis may exhibit the aforementioned behavior.

RMSINT

Default = LINEAR for the trapezoidal approximation.

RMSINT specifies the interpolation method for numerical integration when computing both RMS (Root Mean Square) and N0 (Number of Zero Crossings or Mean Frequency) from PSDF (Power Spectral Density Function). RMSINT = LINEAR requests the trapezoidal approximation, which is the existing Nastran approach. RMSINT = LOGLOG requests the Log-Log interpolation.

ROHYDRO

Default = Remark 3., SOL 700 only

Defines the minimum density for hydrodynamic, single-material Eulerian elements.

Format:

PARAM,ROHYDRO,VALUE

Example:

PARAM,ROHYDRO,1.E-6

VALUE Density cutoff. (Real > 0.0)

Remarks:

1. Hydrodynamic, single-material Eulerian elements with a density less than ROHYDRO are considered to be empty. All of the variables are set to zero, and the equation of state is bypassed.
2. In the Eulerian transport calculation, if the material is flowing from element A to element B, and
 - a. If the density of element B after transport is less than ROHYDRO, then no transport is done.
 - b. If the density of element A after transport is less than ROHYDRO, then all of the mass is transported to element B.
3. By default, the cutoff density for hydrodynamic Eulerian elements is set to 1.E-5 times the material reference density. For the Riemann solution-based solvers, the default is set to 1.E-6 times the reference density.
4. Please note that this parameter has a different effect in the Riemann-solution based Euler solvers. The elements are never viewed as empty, but a small amount of mass (equal to the element's volume times the cut-off density value) remains in the element. All other state variables (velocity, energy and pressure are reset to zero for these types of elements). For fluid flows where you wish to model cavitation, use Tait's equation of state with a critical density at which the pressure remains constant and the fluid cavitates.

ROMULTI

Default = Remark 3., SOL 700 only

Defines the minimum density for multimaterial Eulerian elements.

Format:

PARAM,ROMULTI,VALUE

Example:

PARAM,ROMULTI,1.E-6

VALUE Density cutoff. (Real > 0.0)

Remarks:

1. Multimaterial Eulerian elements with a density less than ROMULTI are considered to be empty. All of the variables are set to zero, and the equation of state is bypassed.
2. In the Eulerian transport calculation, if the material is flowing from element A to element B, and
 - a. If the density of a specific material in element B after transport is less than ROMULTI, no transport is done.
 - b. If the density of a specific material in element A after transport is less than ROMULTI, all of the mass of that material is transported to element B.
3. By default, the cut-off density is set for each material separately as 1.E-5 times the material reference density.

ROSTR

Default = Remark 3., SOL 700 only

Defines the minimum density for single-material Eulerian elements with shear strength.

Format:

PARAM,ROSTR,VALUE

Example:

PARAM,ROSTR,1.E-6

VALUE Density cutoff. (Real > 0.0)

Remarks:

1. Single-material Eulerian elements with shear strength with a density less than ROSTR are considered to be empty. All of the variables are set to zero, and the equation of state is bypassed.
2. In the Eulerian transport calculation, if the material is flowing from element A to element B, and
 - a. If the density of element B after transport is less than ROSTR, then no transport is done.
 - b. If the density of element A after transport is less than ROSTR, then all of the mass is transported to element B.
3. By default the cut-off density for Eulerian elements with shear strength is set to 1.E-5 times the material reference density.

ROTSEKE

Default = -1

Controls the printout of rotor energies in the .f06 file. For a model with only one rotor, the strain and kinetic energy percentages will always be 100%.

- 1 Rotor energies are printed in the .f06 file. (Default)
- 0 Rotor energies are not printed in the .f06 file.

RSPECTRA

Default = -1

RSPECTRA = 0 requests that response spectra be calculated for transient analysis. See [Response Spectrum Analysis](#) (Ch. 9) in *Dynamic Analysis User's Guide* for a discussion of this capability. Response spectra will be calculated for any superelements or the residual structure for which other output requests are present in the same run. The requirements for the other output requests are also in [Response Spectrum Analysis](#) (Ch. 9) in *Dynamic Analysis User's Guide*. Any punch data produced is sent to the standard Nastran PUNCH file. Related parameters are TABID and RSPRINT.

RSPRINT

Default = 0

RSPRINT controls the printout of tabulated values of response spectra. RSPRINT = -1 suppresses the printout. The related parameter is RSPECTRA.

RSTTEMP

Default = NO

In SOL 106, PARAM,RSTTEMP,YES will cause the automatic restart to ignore changes to or additions of TEMPij Bulk Data entries. This is applicable to SOL 106 restart runs in which the temperature changes are only intended to affect the loading and not the material properties. The restart run also requires the use of the DBLOCATE FMS statement instead of the RESTART statement to reference the data base; e.g.,

```
assign run1=plate-run1.MASTER  
dbloc logical=run1
```

S1, S1G, S1M

Default = -1

The MAXMIN Case Control command offers more features with much greater efficiency. PARAM,S1i,+1 requests the sorting and/or filtering of the element stresses selected on the DTI,INDTA entry. Stresses in the element coordinate systems (S1), at grid points (S1G), and/or in material coordinate systems (S1M) based on the parameters BIGER, NUMOUT, SRTOPT, and SRTELTYP may be requested. The S1G and S1M options also require the presence of PARAM,CURV,1.

Parameter	Quantity	Coordinate System	Location	Elements
$S1 \geq 0$	Stresses	Element	Element centers	CQUAD4, CQUAD8, CTRIA3, CTRIA6
$S1M \geq 0$	Stresses	Material	Element centers	CQUAD4, CTRIA3
$S1G \geq 0$	Stresses	Material	Grid points to which elements connect	CQUAD4, CTRIA3

NUMOUT, in conjunction with BIGER, controls the amount of stress output.

NUMOUT2 and BIGER2 serve the same function as NUMOUT and BIGER except that they apply only to composite element failure indices and do not require PARAM,S1i,+1. Similarly, NUMOUT3 and BIGER3 are used for composite strength ratio.

1. NUMOUT = +N requests that N element stresses be printed (or punched) for each element type.
2. NUMOUT=0 outputs all elements in a group when one or more exceeds BIGER. Some of the elements will have stresses small than BIGER. This is conceptually the same as describing an element set in case control, and limiting output in this manner. Stress files obtained with element group filtering may be used for xy plotting and other postprocessor options with DMAP alters. By contrast, the stress file when NUMOUT = -2 is more discontinuous, and may not be used for xy plotting.
3. NUMOUT = 0 does not sort but filters according to BIGER by element group. In static analysis an element group is defined as all case control selected elements for a given load case for SORT1 output. For SORT2 output an element group is defined as the data for a given element type for all load cases. In transient analysis an element group is defined as all case control selected elements at a given time for SORT1 output. For SORT2 output an element group is defined as the data for a given element at all time steps. The element group option applies only to output types described above for PARAM,S1. This option is not available with output types selected by PARAMs S1G and S1M.
4. NUMOUT = -1 requests that stresses be sorted and only those stresses with an absolute value that is greater than BIGER will be output.
5. NUMOUT = -2 (the Default) does not sort but filters according to BIGER. Related parameters include BIGER, NOELOF, NOELOP, and NOGPF.
6. NUMOUT2 = -3 outputs the maximum value of Failure Index for each element.

BIGER controls the elements for which stresses will be printed. Elements with stresses that are smaller in absolute value than BIGER will not be output. The quantity tested is element type dependent. Related parameters include CURV, NUMOUT, S1, S1G, and S1M. SRTOPT controls the scanning option to be performed.

SRTOPT Value	Description
0	Filter/sort on maximum magnitude.
1	Filter/sort on minimum magnitude.
2	Filter/sort on maximum algebraic value.
3	Filter/sort on minimum algebraic value.

SRTELTYP controls the element type to be processed, as described in the following table.

SRTELTYP Value	Description
0	All element types will be processed.
> 0	Only element type SRTELTYP will be processed.

NUMOUT1 and BIGER1 serve the same function as NUMOUT and BIGER except that they apply only to composite element stresses and do not require PARAM,S1i,+1.

NUMOUT2 and BIGER2 serve the same function as NUMOUT and BIGER except that they apply only to composite element failure indices and do not require PARAM,S1i,+1.

S1AG,S1AM

Default = -1

See CURV.

SBSPFORM

No Default

This parameter can be used to specify the Subspace Iteration method formulation for the structure-fluid coupled modes computation.

Subspace iteration method selects the formulation automatically based on the ratio of structure and fluid size and the existence of rigid body or constant pressure modes. But using this parameter, user can force one or other of the methods.

MF: Mobility-Flexibility formulation

SM: Stiffness-Mass formulation

SCRSPEC

Default = -1 (SOLs 103 and 115 only)

SCRSPEC=0 requests that structural response be calculated for response spectra input in normal modes analysis. See [Response Spectrum Analysis](#) (Ch. 9) in the *MSC Nastran Dynamic Analysis User's Guide* for a discussion of this capability. The scaled response calculations are made for elements and grid points of the

residual structure only. There exist two basic methods which are controlled by PARAM,CQC. The default (CQC=0) selects the traditional method. CQC > 0 selects a more recent method called the complete quadratic method of peak response combination—also called the CQC method. In both methods, the responses are summed with the ABS, SRSS, NRL, or NRLO convention, depending on the value of PARAM,OPTION. If the SRSS, NRL, or NRLO options are used, close natural frequencies will be summed by the ABS convention, where close natural frequencies meet the inequality $f_{i+1} < \text{CLOSE} \cdot f_i$. Both PARAM,OPTION and PARAM,CLOSE may be set in any subcase, allowing summation by several conventions in a single run.

In Version 70, the NRL option has been modified slightly to correspond to the NAVSEA-0908-LP-000-3010 specification. NRLO provides the V69 NRL.

PARAM,CQC,1 or 2 selects the complete quadratic method of peak response combination; also called the CQC method. PARAM,CQC,1 selects CQC method of response combination for sum across modes. PARAM,CQC,2 is same PARAM,CQC,1, but outputs the CQC coefficients for each mode ij pair and frequency and damping for each mode. The default is to consider all modes in the calculation but if only a subset of modes are of interest then a DTI,CQC,1,... Bulk Data entry may be used to specify a list of modes (by mode number) to retain for the CQC solution. When multiple excitation directions are specified, PARAM,CQC,1 or 2 specifies the summation across the modes and PARAM,OPTION specifies the summation across excitation directions. As with PARAM,OPTION and PARAM,CLOSE, when PARAM,CQC is used at the subcase level, each subcase can have a different value specified for OPTION, CLOSE and CQC. This allows comparisons to be made in the same run among summation across directions using ABS and SRSS methods while using the CQC method to sum across modes.

For the CQC method, PARAM,DIROUT,YES (Default = NO) outputs the responses combined across the modes for each separate excitation direction as well as the responses combined across modes and directions. This allows results to be assessed per excitation direction and for the total combined response.

PARAM,POST,-1 will write the additional directional responses to the op2 file for post processing with Patran. In this case, the directional responses will be labeled in the Patran Results menu with a TIME value corresponding to the excitation sequence number. For example, if the SUPORT entry specifies the following degrees of freedom to be excited: SUPORT,1,123 then direction 1 will be labeled with TIME value 1, direction 2 with TIME value 2, etc. PARAM,POST,-1 will also write the additional maximum response to the op2 file for post processing. In this case, the maximum response will be labeled in the Patran Results menu with a TIME value 4.

For the CQC method, PARAM,NEWMARK,YES (Default = NO) may be used in the case where 3 simultaneous excitation directions are defined and calculate the Newmark combinations across the excitations directions using the 40% rule. PARAM,PERCENT (Default = 40) may be used to specify a different percentage for Newmark combinations.

SDCSV

Default = 0

Dictates what stability derivative information is to be stored on a CSV (comma separated values) file in a SOL 144 (static aeroelasticity) task. The unit the CSV file is stored to is specified by PARAM, SDUNIT, n. SDCSV has the following options which can be summed to select multiple options:

- 0 No output
- 1 Rigid aero
- 2 Rigid Splined
- 4 Elastic restrained
- 8 Elastic unrestrained
- 16 Intertial restrained
- 32 Intertial unrestrained

SDRPOPT

Default = SDRP

SDRPOPT controls the storage of the principal stresses and strains in the stress and strain tables (OES1 and OSTR1 data blocks) in p-version analysis. By default, the principal stresses and strains are stored in the stress and strain tables to support postprocessing. PARAM,SDRPOPT,OFP requests that the principal stresses and strains are not stored in the tables. This can result in a significant reduction in disk space usage.

SDUNIT,n

n SDUNIT is used in conjunction with an FMS ASSIGN statement to specify the unit number for the storage of stability derivative results in SOL 144.

SEKD

Default = RSONLY

Controls the calculation of the upstream superelement differential stiffness matrix in SOL 106.

- RSONLY Calculate the differential stiffness for the residual structure only.
- ALL Calculate and reduce the differential stiffness for all superelements and the residual structure.

SEMAP, SEMAPOPT, SEMAPPRT

SEMAP Default = SEMAP

SEMAPOPT Default = 42

SEMAPPRT Default = 3

The superelement map (SEMAP table) contains several lists useful for determining how the program has partitioned superelement models. It is printed automatically each time this table is generated. It consists of three major parts:



GPM	The Grid Point Map contains a list of each grid point, its interior superelement placement, and the SEID of all grid points connected directly to it by elements. Three tables follow that summarize the connectivity between superelements, sorted on grid point sequence, SEID, and the number of connections.
ISM	The Individual Superelement Map lists the interior grid points, exterior grid and scalar points, elements, and time and storage space estimates for each superelement.
SDT	The Superelement Definition Table contains the SEID of every superelement in the model, the processing order, and a pictorial representation of the superelement tree.

SEMAP, SEMAPOPT, and SEMAPPRT are used to control the amount of output that is printed and other special features. The possible values for SEMAP are shown in the following table.

SEMAP Value	Output and Application
SEMAP (Default)	ISM, SDT. The lengthy GPM is suppressed. This is the appropriate value for use after the model is stable and only minor changes are to be made.
SEMAPALL	GPM, ISM, SDT. All tables are printed. This value is useful on the initial debug run of a model and when making extensive modeling changes.
SEMAPCON	Only the summary tables of the GPM and the estimation data is output. This is a useful value when iterating to an economic partitioning scheme for large, complex models.
SEMAPEST	Only the estimation data is printed. This is useful when evaluating several alternative partitioning schemes.
SEMAPPUN	No output is printed. The exterior grid points of the superelement with a SEID that is input on SEMAPOPT are placed on a CSUPER entry image on the PUNCH file, allowing the superelement to be used as an external superelement. If SEMAPOPT > 0, the superelement entry is given an SSID of SEMAPOPT. If SEMAPOPT < 0, the exterior points listed are those of the residual structure, but the CSUPER entry is given an SSID of SEMAPOPT .

Other special features are available with parameters SEMAPOPT and SEMAPPRT. They are fully described under parameters OPT1 and OPT2 in the description of the TABPRT module in the [MSC Nastran DMAP Programmer's Guide](#).

If the default value of SEMAP is used, the other two parameters may be used to further refine this output, as described in [MSC Nastran DMAP Programmer's Guide](#) under the TABPRT module description. The printing of the SEMAP table can be avoided by the use of PARAM,SEMAPPRT,-1.

SENSUOO

Default = NO

By default, in dynamic sensitivity analysis in SOL 200, displacements at the o-set due to pseudo-loads do not include any effect due to inertia and damping. If PARAM,SENSUOO,YES is specified then these effects will be computed in a quasi-static manner and included in the sensitivity analysis.

SEP1XOVR

Default = 0

The old and new location of moved shell grid points are printed if SEP1XOVR = 16. When the RSSCON shell-to-solid element connector is used. By default, the moved shell grid points are not printed, SEP1XOVR = 0. See the description of PARAM,TOLRSC for more details.

SEQOUT

Default = 0

See OLDSEQ.

SERST

Default = AUTO

By default, all restarts are considered automatic (see [Restart Procedures in the MSC Nastran Reference Guide](#)). If none of the following Case Control commands are entered, then SEALL=ALL is the default action: SEMG, SELG, SEKR, SELR, SELA, SEMA, SEMR, and SEALL.

These commands may be used to partition the analysis into several runs. By default, the restart will proceed in automatic fashion for each command, regenerating only that data that is affected by modifications in the Bulk Data and Case Control or changes in upstream superelements. If the user wishes to overwrite the data, even if it is not affected by modifications to the data, then PARAM,SERST,MANUAL must be entered.

With PARAM,SERST,AUTO or MANUAL, all superelements will be processed through Phase 0 (see [Superelement Analysis in the MSC Nastran Reference Guide](#)). This phase includes execution of the sequencer module (SEQP), initial superelement processing (SEP1), and initial geometry processing (GP1 and GP2) modules, which can result in significant CPU overhead. If this overhead is not desired, then PARAM,SERST,SEMI will limit Phase 0 and Phase 1 to only those superelements specified on the SEMG, SELG, SEKR, SELR, SELA, SEMA, SEMR, and SEALL Case Control commands. If none of these commands is entered, then execution will skip Phase 0 and 1.

In the modal solution sequences (SOLs 110, 111, 112, 145, 146, and 200), the modes of the residual structure are automatically computed in Phase 2 if any SE-type command (e.g., SEMG=n) is requested for the residual structure. If PARAM,SERST,SEMI and no SE-type command is specified for the residual structure, then, by default, its modes will not be recomputed. This logic is intended for restarts from SOL 103 into one of the modal solutions. If, however, the modes have not already been computed or need to be recomputed, then PARAM,SERST,RSMDS must be specified to force the calculation of the residual structure modes.

If PARAM,SERST,SEDR is specified, then Phases 0, 1, and 2 will be skipped. This option is intended for data recovery (Phase 3) runs only.

The options of SEMI, RSMDS, and SEDR are intended for models that are defined on more than one database; i.e., superelements are defined on separate databases (multiple MASTER DBsets) and processed in separate runs. Also, with this technique, databases are attached with the DBLOCATE File Management statement rather than the RESTART File Management statement. In general, these options are not recommended because they disable the automatic restart capability, which compromises the database integrity.

SESDAMP

Default = NO

PARAM,SESDAMP,YES

Modal damping is calculated for superelements if PARAM,SESDAMP,YES is specified. An SDAMPING Case Control command that selects a TABDMP1 Bulk Data entry must also be specified in the desired superelement's subcase. By default, modal damping is added to viscous damping (B). If you insert PARAM,KDAMP,-1 (or PARAM,KDAMPFL,-1 for fluid superelements) then modal damping will be added to structural damping (K4). The SESDAMP parameter needs to be specified as NO for an assembly run.

PARAM,SESDAMP,AUG

By its nature, classical modal damping in superelements is computed only for the component modes. If fixed boundary or mixed boundary (some fixed, some free) methods are used to compute the component modes, no damping is computed for the constraint modes, which can lead to answer differences when comparing the same model with and without superelements. An enhanced modal damping method has been developed to account for damping on the constraint modes. To use the enhanced damping method instead of the classical method, define "PARAM,SESDAMP,AUG". When this enhanced damping method is defined, the user may choose how the damping factor for the constraint modes is determined via the parameter TDAMP.

SESEF

Default = -1 (SOLs 103 and 115 only)

If SESEF = 0 in superelement normal modes analysis, the fraction of total strain energy for a superelement in each of the system's modes is output in the vector SESEFA for tip superelements and in SESEFG for nontip superelements. If SESEF = 1, strain energy fractions are output, and expansion of the eigenvectors from a-set size to g-set is branched over for tip superelements. If SESEF = -1 (the default value), no strain energy fractions are computed.

Output requests must be present in order for strain energy fractions to be calculated. If SESEF = 1, no other output results for tip superelements.

SHIFT1

Default = -1.234

The negative shift used when computing massless mechanism constraint modes with PARAM,MMMETH,OLD. For very stiff model (1000. hz for the first flexible mode), consider using a larger value. See also [MECHFIX, 898](#) and [MMMETH, 908](#).

SHLDAMP

Default = SAME

If SAME, then structural damping is obtained from MID1 material of PSHELL. If DIFF or any value not equal to SAME, each MIDi field of the PSHELL will have its own structural damping. See Remark 5. of the PSHELL.

SIGMA

Default = 0.0

The radiant heat flux is proportional to

$$\text{SIGMA} \cdot (T_{grid} + T_{ABS})^4$$

where SIGMA is the Stefan-Boltzmann constant, T_{grid} is the temperature at a grid point, and T_{ABS} is the scale factor for absolute temperature and may be specified by PARAM,TABS. These parameters must be given in units consistent with the rest of the data in the model. The value for SIGMA in SI units is

$$5.67 \times 10^{-8} \text{ watts/m}^2 \text{K}^4$$

The default value causes radiant heat effects to be discarded.

SKINOUT

Default = NONE

Request that sets of grid and element lists be output for both the fluid and structure at the fluid-structure interface.

- | | |
|-------|--|
| NONE | Requests no output of sets. |
| PUNCH | Requests set output to .pch only. |
| PRINT | Requests set output to .f06 only. |
| ALL | Requests set output to both .pch and .f06. |

See the Case Control command FLSPOUT as an alternative selection.

SKPAMP

Default = 0

For SOLs 145, 146, and 200, SKPAMP = -1 suppresses all unsteady aerodynamic calculations. The automatic restart performs a similar function without this parameter. Specifying it ensures suppression of the calculations, regardless of the determination of the automatic restart.

SLOOPID

Default = 0 (SOL 129 and 159 only)

In a nonlinear transient analysis (SOLs 129 and 159) restart, SLOOPID identifies the initial conditioning previous nonlinear analysis run (SOLs 106 and 153 respectively). Setting SLOOPID greater than 0 will cause SOLs 129 and 159 to start from the static deformed position.

SMALLQ

Default = 0.0

By default Nastran removes unused superelement q-set degrees-of-freedom from the residual structure solution set. Set this parameter to a small value (e.g., 1.0E-10) if you do not want unused superelement q-set degrees-of-freedom removed.

SNORM

Default = 20.0

SNORM > 0.0 requests the generation of unique grid point normals for adjacent shell elements (see [Figure 6-2](#)). Unique grid point normals are generated for the CQUAD4, CTRIA3, CQUADR, and CTRIAR elements. The grid point normal is the average of the local normals from all adjacent shell elements including CQUAD8 and CTRIA6 elements. If grid point normals are present, they are used in all element calculations of the CQUAD4, CTRIA3, CQUADR, and CTRIAR elements.

SNORM	Tolerance in Degrees
> 0.0	Unique grid point normals are generated if each angle between the grid point normal and each local normal of the adjacent shell elements is smaller than SNORM. SNORM Bulk Data entries overwrite a generated normal.
= 0.0	The generation of grid point normals is turned off. The user can define normals with the SNORM Bulk Data entry.
< 0.0	Grid point normals are not generated. SNORM Bulk Data entries are ignored.

Caution: If the grid shown in [Figure 6-2](#) is located on a symmetric half model boundary and, hence, Shell 2 is not present, you may attain the same result as a full model by specifying the normal direction with the SNORM Bulk Data entry.

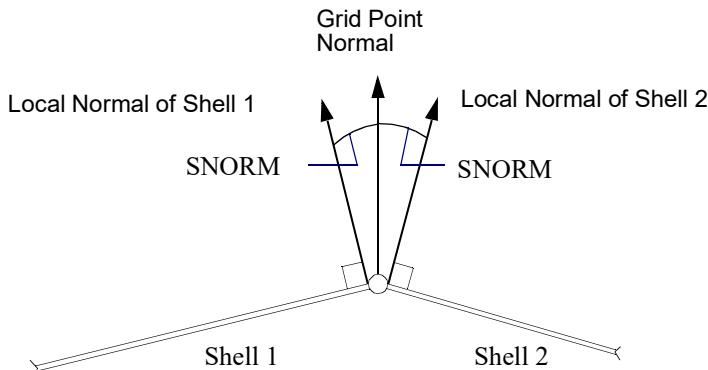


Figure 6-2 Unique Grid Point Normal for Adjacent Shell Elements

SNORMPRT

Default = -1

PARAM,SNORMPRT,>0 writes the grid point normals of the model in the basic coordinate system to the .f06 and/or .pch files.

SNORMPRT	Switch to Print Out Normals
≤ 0	No output
1	Print out to the punch file (.pch)
2	Print out to the print file (.f06)
3	Print out to the punch (.pch) and print file (.f06)

SOFTEXIT

Default = NO

In SOL 200, if soft convergence is achieved during optimization, before completing the maximum number of design iterations, the user may request an exit with PARAM,SOFTEXIT,YES.

SOLADJC

Default = 0

PARAM SOLADJC indicates if adjoint solution vectors are to be calculated during the analysis:

- 1 Do not calculate adjoint solution vectors during the analysis. Any required adjoint solution vectors will be computed during sensitivity analysis.
- ≥ 0 For ANALYSIS=DFREQ, the adjoint vectors will be computed during the solution if:
 1. All frequency response DRESP1 entries are grid responses.
 2. Each subcase has the same set of excitation frequencies.
 3. The number of degrees-of-freedom referenced on DRESP1 entries < (number of independent design variables + number of type-2 properties + number of spawned nonlinear beam library properties) x (number of subcases.)
 4. PARAM AUTOADJ=YES (Default)
- 999999 For ANALYSIS=MFREQ, the adjoint vectors will be computed during the solution if the four items above are satisfied and SOLADJC=999999. SOLADJC=999999 cannot be used in conjunction with FREQ3, FREQ4 or FREQ5 entries to specify excitation frequencies.

SOLID

Default = 0

SOLID also allows several models to be stored in the same graphics database created by PARAM,POST,0.

SPARSEDM

Default = YES

See SPARSEDR.

SPARSEDR

Default = YES

SPARSEDR=YES limits the data recovery matrix calculations to recover only those grid points specified on SET commands referenced by grid point output requests (DISP, SPCF, etc.) or those points connected to elements specified on element output requests (STRESS, FORCE, etc.). In SOL 200, the design model and design responses are also taken into consideration for determining which grid points are needed for data recovery.

SPARSEDM=YES is for SOL 200 and takes further advantage of small design models/responses during the adjoint or pseudo-load sensitivity calculations.

These methods take advantage of very small output requests, and/or small design models in SOL 200, for large models resulting in significant CPU and disk space savings.

If, however, the output requests and/or the size of the design model in SOL 200 require the calculation of the solution over a large enough percentage of degrees-of-freedom, then it is more efficient to compute the solution at all grid points. The user PARAMeter SPDRRAT (Default75) and SPDMRAT (Default60) specifies this percentage.

The sparse data recovery method is not supported in Aeroelastic (SOL 146), Flutter (SOLs 145 and 200), Complex Eigenvalue (SOLs 107, 110, 145, and 200), Nonlinear (SOLs 106, 129, 153, 159, and 400) and Cyclic Symmetry Analysis (SOLs 114, 115, and 118).

The sparse data recovery method is deactivated when the following Case Control commands are specified: EKE, ESE, EDE, and CMSENRGY.

PARAM,DDRMM is ignored under PARAM,SPARSED,YES. To restore the data recovery solution process to pre-V2004 methods insert PARAM,SPARSED,NO.

SPARSEPH

Default = YES

The default selects the very efficient sparse data recovery method during eigenvector data recovery in DOMAIN SOLVER ACMS (PARTOPT=DOF). This method uses the same process for eigenvector data recovery as is requested by PARAM,SPARSED,YES in other dynamic solution sequences like SOLs 107 through 112.

SPARSEPH will be automatically changed to NO if ACPOWER or INTENSITY calculations are requested. These calculations require all eigenvector DOF to be computed.

PARAM,SPARSEPH,NO forces full data recovery on all degrees-of-freedom and can be significantly less efficient if data recovery is required at only a few degrees-of-freedom. See related user parameters [MDOTM](#), [897](#) and [SPARSED, 996](#).

SPDRRAT

Default = 75

See SPARSED.

SPCGEN

Default = NO

SPCGEN is analogous to and described by the PUNCH keyword on the AUTOSPC Case Control command. The specification of the AUTOSPC command overrides the specification of PARAM,SPCGEN.

SPDMRAT

Default = 60

See SPARSED.

SQSETID

Default = 99000001

See [AUTOQSET, 794](#).

SRCOMPS

Default = NO

SRCOMPS controls the computation and printout of ply strength ratios. If SRCOMPS=YES, ply strength ratios are output for composite elements that have failure indices requested.

SRTELTYP

Default = 0

See S1, S1G, S1M.

SROPT

Default = 0

See S1, S1G, S1M.

START

Default = 0

See OLDSEQ.

STEPFCT

Real, Default = 0.666, SOL 700 only. The scale factor has an effect on the time step computation of Lagrangian and Euler elements.

STEPFCTL

Real > 0, Default = 0.9, SOL 700 only.

Defines a scale factor to be used on the internally calculated time step. When both of STEPFC and STEPFCTL are defined, STEPFCTL will be used for Lagrangian elements and STEPFC will be used for Euler elements.

STIME

Default = 0.0 (SOLs 109, 112, 129 and 159 only)

In restarts from previous transient analysis runs, the user provides STIME = t_N where t_N is the last time step of the subcase to be continued with a new or changed subcase in the new run. Thus, the loading and printout will start from t_N as though the original run contained the new subcase data.

In SOLs 109 and 112 restarts from previous SOLs 109 and 112 runs, STIME is used to specify the proper starting time of the restart run. If STIME exceeds the last output time of the previous run, the starting time is assumed to be the last output time. Otherwise, the starting time is assumed to be the output time of the previous run (not necessarily the last output time) that is closest to STIME. In other words, the starting time of the restart run need not be the last output time of the previous run, but may be any time earlier than that. The program informs the user that it is a restart run and indicates the starting time (determined as above) that is used for the restart run.

In SOLs 109 and 112 restarts, the user must ensure that the model and the constraints as well as the subcase setup in the restart run are the same as those in the previous run. The user may, however, specify different TSTEP and DLOAD requests in Case Control and also different TSTEP and dynamic loading entries in the Bulk Data compared to the previous run. The loading and the results output from the restart run will start from the new starting time.

Notes:

1. The responsibility for ensuring that the model and the constraints as well as the subcase setup in the restart run are the same as those in the previous run is left to the user; the program does not check for this condition. If this condition is not met, the program may terminate the execution with a fatal error or give erroneous results.
2. Restarts in SOLs 109 and 112 using STIME > 0.0 are supported **only** for applied loads, **not** for SPC/SPCD enforced motion. If this usage is violated, the program terminates the execution with an appropriate fatal message.

STRUCTMP

Replaced by a keyword on the FLSPOUT Case Control command.

SUBCASID

Default = 0

PARAM,SUBCASID,n where n is greater than zero, specifies that the restart proceeds from SUBCASE n in nonlinear static analysis, SOL 106. SUBCASID is an alternative to SUBID and is recommended over SUBID which indicates the subcase sequence number.

SUBID

Default = 1

In SOL 106 by default, the restart proceeds from the last LOOPID in the last subcase. SUBID may be used to specify an earlier subcase by specifying the sequential number (for SEID = 0) of the subcase. In SOLs 106 and 153, PARAM,LOOPID may also be specified for an earlier LOOPID. SUBCASID is an alternative to SUBID and is recommended over SUBID. See [Restarts for Nonlinear Static Analysis, 684](#) in Appendix C: Nonlinear Analysis in *MSC Nastran Reference Guide* for further discussion.

SUPAERO

Default = ZONA

If SUPAERO=ZONA, then the ZONA51 code is used for supersonic aerodynamic calculations. If SUPAERO=CPM, then the CPM method is used. If ZONA51 is not available at a particular installation, PARAM,SUPAERO,CPM must be specified to avoid a fatal error when performing supersonic aerodynamic analyses. Only one supersonic aerodynamics method can be selected in a given run.

Note:

CPM is only supported for planar configurations.

SUPDOF

Default = 0

Digits 1 through 6 indicate which of the six rigid body degrees of freedom are supported for a Solution 144 run. For example, SUPDOF=35 indicates that the plunge (3) and pitch (5) degrees of freedom are supported. This parameter is optional. If omitted, Nastran will compute these data using a numerical method.

SUPER

Default = $\begin{cases} 0 & \text{(nonsuperelement sequences)} \\ -1 & \text{(superelement sequences)} \end{cases}$

See OLDSEQ.

TABID

Default = 2

TABID controls the punch output for response spectra. See [Appendix C page 597](#) in *MSC Nastran Reference Guide*. A related parameter is RSPECTRA.

TABS

Default = 0.0

TABS is used to convert units of the temperature input ($^{\circ}\text{F}$ or $^{\circ}\text{C}$) to the absolute temperature ($^{\circ}\text{R}$ or $^{\circ}\text{K}$). Specify:

PARAM,TABS,273.16 When Celsius is used.

PARAM,TABS,459.69 When Fahrenheit is used.

Refer to the Bulk Data entry, [CREEP, 1628](#) for a creep analysis with SOLs 106 or 153. Refer to PARAM,SIGMA for heat transfer analysis.

TDAMP

PARAM,TDAMP,Real

When using enhanced modal damping for superelements (see PARAM,SESDAMP,AUG), the user may choose how the damping factor for the constraint modes is determined.

If TDAMP=0.0 (the default), the damping factor is averaged over all the damping coefficients defined on TABDMP1.

If TDAMP is less than 0.0, the damping factor from TABDMP1 associated with the lowest natural frequency is used.

If TDAMP is positive, the damping factor defined by TDAMP is used directly.

TESTNEG

Default = $\begin{cases} -2 & \text{for Newtons method} \\ -1 & \text{for Arc-length method and SOLs 101-112,115,118,187, and} \\ & 200 \end{cases}$

In nonlinear static analysis (SOLs 106 and 153), and differential stiffness generation with preload (SOLs 101-112, 115, 118, 187, and 200 with STATSUB) this parameter specifies the action to take when negative terms are encountered on the factor diagonal of matrix decomposition. Negative terms indicate that the differential

stiffness has introduced a structural instability. The instability may be real (structural buckling) or mathematical (the current iteration appears unstable, but a stable solution exists).

TESTNEG	Results
-1	Stop if negative terms occur.
1 or 0	Continue if negative terms occur.
-2	If negative terms exist, do not use differential stiffness.
2	Do not use differential stiffness.
3	Use differential stiffness with preload even if negative terms exist (SOLs 101-112, 115, 118, 187, and 200 with STATSUB only)

TFSYMFAC

Default = 1.D-08

TFSYMFAC specifies the tolerance at which matrices generated via the TFL Case Control command are treated as anti-symmetric in the solution process. The default will not symmetrize the matrix even if the tolerance is not exceeded. Whereas, if TFSYMFAC is negative then the absolute values is used as the tolerance and the matrix will be symmetrized if the tolerance is exceeded. If TFSYMFAC is 0.D0 then the symmetry is not checked.

TINY

Default = 1.E-3

Any elements with strain energy that is less than a TINY percentage of the total strain energy for any superelement will not be printed or made available for postprocessing by MSC Patran or other programs. TINY may also be used to suppress the printing of small numbers in the constraint check matrix [E_{mh}] described in [Geometry Processing in SubDMAP PHASE0 in the MSC Nastran Reference Guide](#).

TOLRSC

Default = 0.05

When the RSSCON shell-to-solid element connector is used, the connecting grid points of the shell element are moved on to the solid face if the grid points are close enough. The tolerable distance of the shell grid point to the solid edge or face is $\epsilon \cdot h$ where h is the height of the solid edge; see the sample figure below. The relative tolerance is user modifiable using the parameter.

PARAM,TOLRSC, ϵ

The default for the relative tolerance is $\epsilon = 0.05$. Rigid body invariance is satisfied with double-precision accuracy if the shell grid points are adjusted.

TOPOCONV

Default = 80

Parameter TOPOCONV is applicable only to ESLNRO topology optimization tasks. It sets a lower bound for the percentage of the design variables whose maximum relative changes are within the tolerance specified by CONVDV on the DOPTPRM entry.

By default, when more than 80% of the design variables show their maximum relative changes are within CONVDV, the job will be terminated.

TRMBIM

Default = PHYSICAL

TRMBIM is used to define the formulation used for reduced impedance matrix, RIM, for all trim components requested under TRIMGRP case control commands. It is default to 'PHYSICAL' which will generate RIM in physical coordinates. The other valid option for TRMBIM are:

1. 'MODAL' which will produce RIM in modal coordinates.
2. 'DATACHK' which perform dry run without producing Reduce Impedance Matrix or results. Job will be terminated afterwards.

It is important to note that data recovery for trim components are supported only under 'PARAM,TRMBIM,PHYSICAL' and 'PARAM,PEMNPART,1 (default)' for all trim components. In addition, SOL 200 supports for trim components is limited to following conditions,

1. Design model on structure and/or fluid model only (not the trim/PEM material).
2. Design responses/constraints on responses of structure and/or fluid only (not the trim/PEM material).

TSTATIC

Default = -1 (SOLs 129, 159, 400 and 600 only)

If TSTATIC = 1, a static solution may be obtained while ignoring inertial and damping forces. For SOLs 129 and 159, this option is available only with the adaptive time-stepping method (see METHOD = "ADAPT" on the Bulk Data entry, [TSTEPNL, 3286](#)). For SOL 600, TSTATIC=1 may only be used for the default auto step iteration procedure.

UGASC

SOL 700 only,

Real. Defines a value for the universal gas constant.

1. Must be used if the molar weight is used on a GAS option of the AIRBAG entry, or if molar gas fractions are given on an INFLFRAC entry.
2. Specify only one universal gas constant per problem.
3. In IS units, R equals $8.3145 \text{ J mol}^{-1} \text{ K}^{-1}$.

Using the tonne, mm, system of units R has a value of $8314.5 \text{ tonne mm}^2 \text{ s}^{-2} \text{ mol}^{-1} \text{ K}^{-1}$.

In imperial units, R equals $1.9859 \text{ Btu lbmol}^{-1} \text{ }^{\circ}\text{R}^{-1}$ or $1545.3 \text{ ft lbf lbmol}^{-1} \text{ }^{\circ}\text{R}^{-1}$

UNSYMF

Default = NO

In SOL 106, nonlinear statics, PARAM,UNSYMF,YES is required to include damping effects in the calculation of complex eigenvalues. See PARAM,NMLOOP.

UNSYMKB

Character, default = 'NO'

In modal solutions or real eigenvalue solutions, if the stiffness or damping matrix is not symmetric, the run will normally terminate with a FATAL error message. Setting UNSYMKB to 'YES' will allow the run to continue.

UPDTBSH

Default = NO

UPDTBSH controls the update of boundary shapes generated by auxiliary boundary model analysis in SOL 200. By default, the auxiliary boundary models and shapes are generated only once at the initial design cycle and will not be updated in subsequent cycles even if the shape of the primary model is changing. PARAM,UPDTBSH,YES requests that the auxiliary models and shapes are updated and reanalyzed at every cycle.

USETPRT

Default = -1

USETPRT controls the tabular printout of the structural degree-of-freedom sets. See [Degree-of-Freedom Sets, 1111](#). To print aerodynamic degree-of-freedom sets, please see [OPGEOM](#).

Sequence	Print	USETPRT
None	None (Default)	-1
	Row sort only	0
Internal	Column sort only	1
	Row and Column sort	2
External	Row sort only	10
	Column sort only	11
	Row and Column sort	12

The degrees-of-freedom can be listed in ascending order according to their internal or external sequence number, but not both. The external sequence number is the grid, scalar, or extra point identification number. The internal sequence number is the number assigned after resequencing (see PARAM,OLDSEQ).

The row sort is not recommended in p-version analysis because large integers are generated for hierarchical grid point identification numbers and they will be truncated.

For a given sequence there are two types of tables that may be printed: row sort and column sort. For row sort, a table is printed for each set selected by USETSEL. Here is an example of row sort (USETPRT = 0 or 10):

U S E T D E F I N I T I O N T A B L E (I N T E R N A L S E Q U E N C E , R O W S O R T)									
	A								
-1-	-2-	-3-	-4-	-5-	-6-	-7-	-8-	-9-	-10-
1=	2-1	2-2							

For column sort, a single table is printed for the following sets: SB, SG, L, A, F, N, G, R, O, S, M, E. Here is an example of column sort (USETPRT=1 or 11):

U S E T D E F I N I T I O N T A B L E (I N T E R N A L S E Q U E N C E , C O L U M N S O R T)																		
EXT	GP.	DOF	INT	DOF	INT	GP.	SB	SG	L	A	F	N	G	R	O	S	M	E

1	- 1	1-		1 G		1						1	1					1
	- 2	2-				2						2	2					2
	- 3	3-					1					3	3					3
	- 4	4-						2				4	4					4
	- 5	5-						3				5	5					5
	- 6	6-						4				6	6					6

USETSEL

Default = 0

USETSEL specifies the sets which will be printed in the row sort (USETPRT = 0 or 10). In order to select specific sets to be printed, you must sum their corresponding decimal equivalent numbers. For example, sets A, L, and R are selected with USETSEL=128+256+8=392.

USETSEL	Sets Printed
-1	All sets as defined in Degree-of-Freedom Sets, 1111 .
0	Mutually exclusive sets only; i.e., sets M, SB, SG, O, Q, R, C, B, E, and A.

USETSTR1

Input-character-Default' ‘

USETSTR1 through USETSTR4 specifies the sets that will be printed by the specification of parameters USETPRT and USETSEL. Any set in [Degree-of-Freedom Sets, 1111](#) may be specified. A “:” is used as a separator. In the following example, the m-set (degrees-of-freedom eliminated by multipoint constraints) and s-set (degrees-of-freedom eliminated by single point constraints) are specified.

Example:

PARAM,USETSTR1,M:S

VARPHI

Default = $\pi/4$ (0.78539816)

Feature angle for detection of singular geometric features (sharp edges or corners) in the mesh during adaptive mesh refinement.

When adaptive meshing is requested (see Case Control command, [HADAPT \(Case\), 417](#) and Bulk Data entry, [HADPLT, 2126](#)), singular geometric features such as sharp edges or sharp corners must be detected. To this end face outward normals N_1, N_2 of each pair of adjacent mesh faces and the edge oriented tangents T_1, T_2 of each pair of adjacent mesh edges are computed (see [Figure 6-3](#)). If the angle between N_1 and N_2 for mesh faces, or between T_1 and T_2 for mesh edges is bigger than the feature angle φ , then the common edge or vertex will be considered a splitting edge or vertex where surfaces or lines are broken and a singular geometric feature is defined.

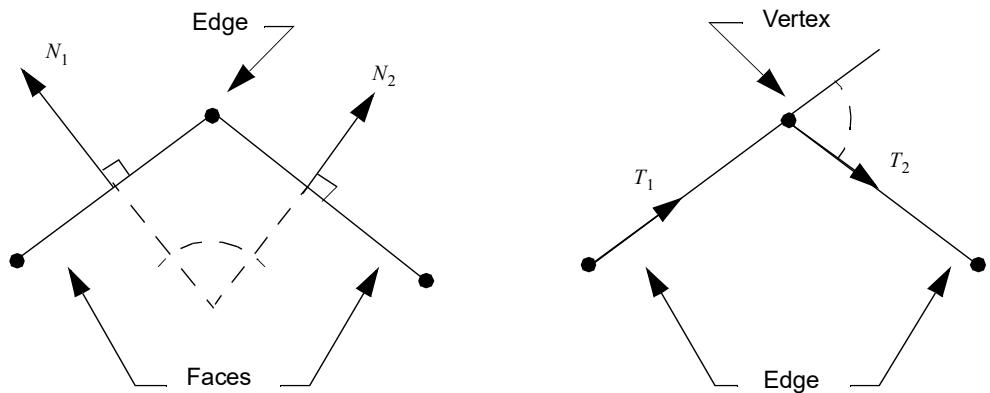


Figure 6-3

Mesh faces and elements are preprocessed to ensure consistent orientation and that the appropriate sign of face normals and edge tangents will be accounted for during the computation of their mutual angle.

VELCUT

Default = 1.E-6, SOL 700 only

Defines the minimum velocity in Eulerian meshes.

Format:

PARAM,VELCUT,VALUE

Example:

PARAM,VELCUT,1.0E-6

VALUE Minimum velocity. (Real > 0.0)

Remark:

Any velocity less than VELCUT is set to zero. It is mainly used to eliminate harmless but annoying small values of velocity caused by round-off error and numerical dispersion.

VMOPT

Default = 0

By default (VMOPT=0), virtual mass is included after the component modes are computed. Virtual mass is not taken into consideration when using GRAV, RFORCE, or ACCEL loading.

If VMOPT=1, then the virtual mass will be included in the mass matrix at the same time as all other mass elements. In other words, the component modes will reflect the virtual mass. Virtual mass is taken into consideration when using GRAV, RFORCE, or ACCEL loading.

If VMOPT=2, the modes of the structure or component without the fluid are computed first ("dry" modes). The fluid effects are added to the modal basis during the residual flexibility computation to produce the "wet" modes for the component. Both eigenvalue tables are printed, allowing comparison of the dry and wet modes. The wet modes are used in modal dynamic analysis. Cost savings result from the dense Virtual Mass (VM) matrix being kept out when computing dry modes in the physical basis. Its presence can increase memory and computation times by an order of magnitude. The VM is added only to the smaller generalized basis used in Residual Flexibility Computations. The approximations introduced by this approach are generally small due to the homogeneous nature of the fluid. It is the preferred method when the number of wetted elements exceeds several hundred, for reasons of efficiency. It is to be noted that MAX normalization for eigenvectors with PARAM,VMOPT,2 may produce incorrect results. Therefore, MAX normalization is not recommended when using PARAM,VMOPT,2.

If VMOPT is not equal to 0, 1 or 2, then no virtual mass is computed.

VREF

Default = 1.0

In modal flutter analysis, the velocities are divided by VREF to convert units or to compute flutter indices.

VUELJUMP, VUGJUMP

Default = 1000

Specifies the separation in identification numbers for display elements and grid points generated in p-version analysis. The defaults are sufficient for a 9 9 display element mesh.

Identification numbers for display elements and grid points start with 10001001 and 201001001, respectively. For example, by default the identification numbers for the display elements of the first p-element will be numbered 10001001 through 100002000 and the second p-element 100002001 through 100003000, etc.

VUBEAM, VUHEXA, VUPENTA, VUQUAD4, VUTETRA, VUTRIA3

Default = VUBEAM

Default = VUHEXA

Default = VUPENTA

Default = VUQUAD4

Default = VUTETRA

These parameters are used in p-version analysis to specify the names of the display elements in the data recovery output tables; such as those created by the VUGRID Case Control command and PARAM,POST. They should be used if your postprocessor does not recognize display elements. For example, PARAM,VUHEXA,CHEXA renames the display element VUHEXA to “CHEXA” in the output files.

WHIRLOPT

Default = FWD

Control forward whirl or backward whirl analysis for SYNC option in SOL 107 and SOL 108 for analysis in rotating reference frame using the ROTOR entry.

Usage:

PARAM, WHIRLOPT, FWD: For forward whirl analysis

PARAM, WHIRLOPT, BWD: For backward analysis

WMODAL

Default = NO

In Modal Transient analysis (SOL 112), PARAM, WMODAL, YES will automatically produce accurate conversion at all frequencies using the modal frequencies based on the eigenvalues as shown below. This relieves the user of trying to determine the “best” value for (PARAM, W3 and W4). Note, use of WMODAL does not suppress user specified values of W3 and W4.

$$\begin{bmatrix} 1/\sqrt{\omega_1} & 0 & \dots & 0 \\ 0 & 1/\sqrt{\omega_2} & \dots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \dots & 1/\sqrt{\omega_q} \end{bmatrix} [\Phi_{dq}^\tau] (G[K_{dd}^1] + [K_{dd}^4]) [\Phi_{dq}] \begin{bmatrix} 1/\sqrt{\omega_1} & 0 & \dots & 0 \\ 0 & 1/\sqrt{\omega_2} & \dots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \dots & 1/\sqrt{\omega_q} \end{bmatrix}$$

Note the CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements when computing the element forces for data recovery do not currently take into account the WMODAL conversion in SOL112.

WRBEAMB

Integer, Default = 0

- 0 Write equivalent radius for all beams (see PARAM, BEAMBEA) whether beam-beam contact is anticipated or not. The equivalent radius is the 7th field of Marc’s GEOMETRY values for beam type elements.
- 1 Do not write equivalent radius (7th field is blank). This might be necessary for versions of Marc earlier than 2003.

WTMAS600

Integer, Default = 1

Determines whether GRAV entries will be scaled by WTMAS600 or not for SOL 600 only.

- 0 The value of A (4th field) on the GRAV entry will not be scaled by WTMAS600.
- 1 The value of A (4th field) on the GRAV entry will be scaled by WTMAS600.

Remarks:

1. This parameter is available starting with the MD Nastran 2010.
2. Prior to MD Nastran 2010, WTMAS600=1 for all analyses.

WTMASS

Default = 1.0

The terms of the structural mass matrix are multiplied by the value of WTMAS600 when they are generated. In coupled fluid-structure analysis WTMAS600 is applied to the structural portion of the model only. WTMAS600 applies to MFLUID entries but it is not recommended for use in hydroelastic problems.

W3, W4, W3FL, W4FL

Default = 0.0

The damping matrix for transient analysis is assembled from the equation:

$$[B_{dd}] = [B_{dd}^1] + [B_{dd}^2] + \frac{G}{W3}[K_{dd}^1] + \frac{1}{W4}[K_{dd}^4]$$

In coupled fluid-structure analysis, W3 and W4 are applied to structural portion of the model and W3FL and W4FL to the fluid portion of the model. The default values of 0.0 for W3, W4, W3FL, and W4FL cause the $[K_{dd}^1]$ and $[K_{dd}^4]$ items to be ignored in the damping matrix, regardless of the presence of the PARAM, G or GFL or $[K_{dd}^4]$. $[K_{dd}^1]$ is the stiffness.

$[K_{dd}^4]$ is the structural damping and is created when GE is specified on the MATi entries.

$[K_{dd}^1]$ is the stiffness. The units of W3, W4, W3FL, and W4FL are radians per unit time. (See [Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS in the MSC Nastran Reference Guide](#) for further discussion.)

In SOLs 129 and 159, W4 may vary between subcases. However, the linear portion of the model uses only the W4 value from the first subcase and the values in the subsequent subcases are applied to the nonlinear portion of the model.

Parameters G, W3, W4 are case control selectable by either the PARAM entries in Case Control, or RSDAMP or SEDAMP appearing in Case Control pointing to a bulk data DAMPING entry. If both DAMPING type entries and PARAM entries occur in a subcase, the DAMPING entries take precedence.

When there is no Parameters G, W3, W4 or DAMPING reference in the first Case Control, the Bulk Data specification (if any) will be used. Subsequent Case Control specification will override the Bulk Data specification and Case Control rules apply to subsequent specifications.

The CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements when computing the element forces for data recovery take into account W3 and W4 in SOL109 and SOL112.

Linear CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements when computing the element forces for data recovery take into account W3 and W4 in SOL400 with ANALYSIS=NLTTRAN. These elements behave in SOL400 with ANALYSIS=NLTTRAN in a linear fashion when PARAM, LGDISP, -1 (Default) and there are no PBUSHT referring to the CBUSH or PELAST entries referring to the CELAS1, CELAS2, CELAS3.

For linear CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements in SOL400 with ANALYSIS=NLTTRAN, SOL400 considers Case Control specification of G, W3, or W4 as a data recovery boundary condition change and each STEP*i* for the linear elements will pick up the appropriate G, W3, and W4 values.

Any of these elements in SOL400 with ANALYSIS=NLTTRAN that are nonlinear only use the standard force output data recovery F=Ku(t) since nonlinear elements compute only the unbalanced stiffness force.

In SOL129 or SOL159 linear CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements (determined as in SOL400 above) should never have G, W3, or W4 selected in the Case Control as the linear force recovery will be incorrect as only the G, W3, and W4 of the first subcase is used.

In SOL129, the FUSE option of the CBUSH should never be used if GE or B is specified for the fusing CBUSH. This is because, unlike SOL400, SOL129 does not update the B or K4 matrix after the CBUSH fuses. Thus remnant CBUSH forces due to damping will be present.

WR3, WR4, WRH

Default = 0.0, no rotor damping or circulation terms.

Specifies “average” excitation frequency for calculation of rotor damping and circulation terms. See “Equations Used in Analyses” on page 192 of the *MSC Nastran 2004 Release Guide* for equations.

XFACT

Default=1.0

Additional factor to determine upper bound for Q-set reorthogonalization subspace, for ACMS VERSION=NEW (module ACMS1).

XFACTX

Default=2.0

Multiplier applied to user's maximum frequency to determine maximum frequency for reorthogonalization for ACMS VERSION=NEW (module ACMS1).

XFLAG

Default = 0

By default (XFLAG = 0), when temperature loads and element deformations are present, the element strain energy for the linear elements is calculated using the following equation:

$$E = \frac{1}{2} u^T K_e u - u^T P_{et}$$

where u is the deformation, K_e is the element stiffness and P_{et} is the element load vector for temperature differences and element deformations. If XFLAG is set to 2, the element strain energy for linear elements is calculated using the following equation:

$$E = \frac{1}{2} u^T K_e u - \frac{1}{2} u^T P_{et}$$

The latter formula is the same strain energy calculation used for nonlinear elements.

When requesting Case Control commands ESE or EDE and executing SOL 108 or SOL 111, if Nastran detects frequency dependent elements (CBUSH if referred to by PBUSH/PBUSHT; CELAS1, CELAS3 if referred to by PELAS/PELAST; CDAMP1, CDAMP3 if referred to by PDAMP/PDAMPT), Nastran will automatically remove the entire class of CBUSH, or CELAS1, or CELAS3, or CDAMP1, or CDAMP3 elements, whether or not all of a class are frequency dependent, from the residual (SE0) energy calculations because frequency dependent elements are not fully implemented in the ESE/EDE calculations and may result in wrong answers. USER INFORMATION MESSAGE 5245 will be issued for each class removed. If the user wishes to override this removal, set XFLAG to 32. If the user wants XFLAG=2 and XFLAG=33, then set XFLAG=34.

XYMPCH

Default = NO

The XYPILOT/XYPUNCH commands do not support SOL103 type mode shapes. XYMPCH when set to a value PARAM, XYMPCH, YES allows the XYPUNCH command XYPUNCH, DISP, RESPONSE / ... to punch without selective selection SOL103 mode shapes as SOL101 type displacements. For example:

```
$  
METHOD = 10  
SET 100 = 6, 7, 13  
PARAM, XYMPCH, YES  
DISP=100  
MODES=20 $  
...  
OUTPUT (XYPILOT)  
XYPUNCH DISP RESPONSE / 6 (T3) $  
XYPUNCH DISP RESPONSE / 7 (T3) $  
XYPUNCH DISP RESPONSE / 13 (T2) $
```

A more modern solution is to include in a standard SOL103 run the Nastran entry

NASTRAN HDF5=0

With for example DISP(PLOT)=ALL

And extract the resulting mode shapes from the resulting HDF5 Data Base.

Then the PARAM, XYMPCH, and MODES=n is not needed nor is any of the OUTPU(XYPILOT) commands.

XYUNIT, n

- n XYUNIT is used in conjunction with an FMS ASSIGN statement to specify the unit number for the storage of design optimization results and design sensitivity data in comma separated value format for use in a spreadsheet in SOL 200 or the^c storage of trim results in SOL 144.

ZROCMAS

Default = NO

When performing component modal synthesis with free or mixed boundary conditions, the c-set mass is normally included during the calculation of the component modes. If the component has large masses on the c-set degrees-of-freedom, or if the user requests too many modes for the component, the c-set residual flexibility will become singular. This causes a failure of the component reduction. The singularity may be avoided by setting ZROCMAS to YES, which will exclude the c-set mass when calculating the component modes.

ZROVEC

Default=1.E-06

ZROVEC specifies the tolerance at which a residual vector is not linearly independent and subsequently removed from the residual vector computation process.

Parameter Applicability Tables

[Table 6-1](#), [Table 6-2](#), [Table 6-3](#), [Table 6-4](#), and [Table 6-5](#) list parameter applicability to the solution sequences (SOLs 101 through 114).

[Table 6-6](#), [Table 6-7](#), [Table 6-8](#), [Table 6-9](#), [Table 6-10](#), [Table 6-11](#), [Table 6-12](#), [Table 6-13](#), [Table 6-14](#), [Table 6-15](#), and [Table 6-16](#) list parameter applicability to the solution sequences (SOLs 115 through 700).

- B Must be specified in the Bulk Data Section only.
- E May be specified in either the Bulk Data and/or Case Control Section.
- C Must be specified in the Case Control Section only.

Table 6-1 PARAMeter Names in SOLs 101 Through 114 Part 1

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
ACEXTMD									B		
ACEXTSET									B		
ACOUT	B				E	E	E	E	E	E	
ACOWEAK									B		
ACSYM					B				B		
ADJMETH											
ADMPOST									E	E	
ADPCON				E							
ADSTAT						B				B	
AERODOF											
AESDISC											
AESMAXIT											
AESMETH											
AESRNDM											
AESTOL											
ALPHA1					B	B	B	B	B	B	
ALPHA2					B	B	B	B	B	B	
ALPHA1FL											
ALPHA2FL											
ALTRED	B		B								B
ALTSHAPE	B	B			B	B	B	B	B	B	
ARBMASP											
ARBMFEM											
ARBMPMS											
ARBMS											
ARBMS											
ARBMSTYP											
ARF											
ARS											
ASCOUP	B	B	B	B	B	B	B	B	B	B	
ASING	E	E	E	E	E	E	E	E	E	E	
AUNITS											
AUTOADJ											

Table 6-1 PARAMeter Names in SOLs 101 Through 114 Part 1 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
AUTOGOUT	E	E	E	E	E	E	E	E	E	E	E
AUTOMSET	E	E	E		E	E	E	E	E	E	E
AUTOQSET	E	E	E	E	E	E	E	E	E	E	E
AUTOSPC	E	E	E	E	E	E	E	E	E	E	E
AUTOSPCR				E							
AUTOSPRT		E		E	E	E	E	E	E	E	
BAILOUT	E	E	E	E	E	E	E	E	E	E	E
BEAMBEA											
BEIGRED				E	E	E	E	E	E	E	
BETA											
BIGER	E	E	E								E
BIGER1	E	E	E								E
BIGER2	E	E	E								E
BUCKLE				E							
					E						
CASPIV	E										
CB1, CB2	E	E	E	E	E	E	E	E	E	E	E
CDIF											
CFDIAGP	E	E	E	E	E	E	E	E	E	E	E
CFRANDEL	E	E	E	E	E	E	E	E	E	E	E
CHECKOUT	E	E	E	E	E	E	E	E	E	E	E
CK1, CK2, CK3	E	E	E	E	E	E	E	E	E	E	E
CLOSE		B									
CM1, CM2	E	E	E	E	E	E	E	E	E	E	E
CONFAC	B	B	B	B	B	B	B	B	B	B	B
COUPMASS	E	E	E	E	E	E	E	E	E	E	E
CP1, CP2	E		E	E		E	E		E	E	E
CQC		B									
CURV	E	E	E								E
CURVPLOT	E	E	E	E							E
CWDIAGP	E	E	E	E	E	E	E	E	E	E	E
CWRANDEL	E	E	E	E	E	E	E	E	E	E	E
DBALL	E	E	E	E	E	E	E	E	E	E	E

Table 6-1 PARAMeter Names in SOLs 101 Through 114 Part 1 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
DBCCONV	E	E	E	E	E	E	E	E	E	E	E
DBC DIAG	E	E	E	E	E	E	E	E	E	E	E
DBCOVWRT	E	E	E	E	E	E	E	E	E	E	E
DBDICT	B	B	B	B	B	B	B	B	B	B	B
DBDN	E	E	E	E	E	E	E	E	E	E	E

Table 6-2 PARAMeter Names in SOLs 101 Through 114 Part 2

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
DBDRPRJ	B	B	B	B	B	B	B	B	B	B	B
DBDRVER	B	B	B	B	B	B	B	B	B	B	B
DBEXT	E	E	E	E	E	E	E	E	E	E	E
DBRCV	E	E	E	E	E	E	E	E	E	E	E
DBUP	E	E	E	E	E	E	E	E	E	E	E
DDRMM									B	B	
DESPCH											
DESPCH1											
DFREQ						B			B		
DOPT	E	E	E								E
DPEPS											
DSNOKD			B								
DSZERO	B	B	B								
DYNSPCF		E			E	E	E	E	E	E	E
EIGFILT		B							B		
ENFMETH					E	E			E	E	
ENFMOTN					E	E			E	E	
ERPC											
ERPREFDB											
ERPRHO											
ERRLRF											
ERROR	E	E	E	E	E	E	E	E	E	E	E
ESLFSAV											

Table 6-2 PARAMeter Names in SOLs 101 Through 114 Part 2 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
ESLMOVE											
ESLRCF											
EST	E	E	E	E	E	E	E	E	E	E	E
EXTDR	B	B			B	B	B	B	B	B	
EXTDROUT	B	B			B	B	B	B	B	B	
EXTDRUNT	B	B			B	B	B	B	B	B	
EXTOUT	B	B	B	B	B	B	B	B	B	B	B
EXTRCV	E	E	E	E	E	E	E	E	E	E	E
EXTUNIT	B	B	B	B	B	B	B	B	B	B	B
FACTOR	B	B	B	B	B	B	B	B	B	B	B
FBATOLR						E			E		
FBLEND											
FIXEDB	E	E									
FKSYMFAC					E						
FLUIDSE		B			B	B	B	B	B	B	
FOLLOWK		E	E	E	E	E	E	E	E	E	
FRQDEPO						B				B	
FULLSEDR	E	E	E	E	E	E	E	E	E	E	E
FZERO		E		E	E	E	E	E	E	E	E
G					B	B	B	B	B	B	
GEOMU	E	E	E	E	E	E	E	E	E	E	E
GFL					B	B	B	B	B	B	
GPECT	E	E	E	E	E	E	E	E	E	E	E
GRDPNT	E	E	E	E	E	E	E	E	E	E	E
GUSTAERO											
GYROAVG						E			E		
HEATSTAT	B										
HFREQ								B	B	B	
HFREQFL								B	B	B	
HTOCITS					E						
HTOCPRT					E						
HTOCTOL					E						
IFP	E	E	E	E	E	E	E	E	E	E	E

Table 6-2 PARAMeter Names in SOLs 101 Through 114 Part 2 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
INREL	E		E								
IRES	E		E	E		E	E		E	E	E
ITAPE	B	B	B								
IUNIT	B	B	B								
KDAMP								B	B	B	
KDAMPFL								B	B	B	
KDIAG				E							
K6ROT	E	E	E	E	E	E	E	E	E	E	E
LANGLE				B							
LFREQ								B	B	B	
LFREQFL								B	B	B	

Table 6-3 PARAMeter Names in SOLs 101 Through 114 Part 3

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
LGDISP				E							
LMFACT	B	B	B								
LMODES		E						B	B	B	
LMODESFL								B	B	B	
LOADU	E	E	E	E	E	E	E	E	E	E	E
LOOPID				E							
MACH											
MAXLP											
MAXRATIO	E	E	E	E	E	E	E	E	E	E	E
MDK4OPT		B						B	B	B	
MDOPT14		B						B	B	B	
MDOTM		B						B	B	B	
MDOTMFAC		B						B	B	B	
MDREDOPT		B						B	B	B	
MESH	B										
METHCMRS		E			E	E	E	E	E	E	
MHRED		E			E	E	E	E	E	E	

Table 6-3 PARAMeter Names in SOLs 101 Through 114 Part 3 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
MINIGOA	B	B	B	B	B	B	B	B	B	B	B
MODACC								B	B	B	
MODEL	B	B	B	B	B	B	B	B	B	B	B
MPCX	B	B	B	B	B	B	B	B	B	B	B
MPTUNIT											
MXICODE0											
NASPRT											
NDAMP											
NEWSET	B										
NHPLUS											
NINTPTS	E	E	E								E
NLAYERS					E						
NLHTLS					E						
NLHTOL											
NLHTWK											
NLMAX		E		E	E	E	E	E	E	E	
NLMIN		E		E	E	E	E	E	E	E	
NMLOOP				E							
NOCOMPS	E	E	E	E							E
NOELOF	E	E	E								E
NOELOP	E	E	E								E
NOGPF	E	E	E								E
NOMSGSTR	E	E	E								E
NONCUP											B
NQSET		E			E	E	E	E	E	E	
NLTOL				B							
NUMOUT	E	E	E								E
NUMOUT1	E	E	E								E
NUMOUT2	E	E	E								E
OCMP	E	E	E	E							E
OEE	E	E	E	E							E
OEF	E	E	E	E	E	E	E	E	E	E	
OEFX	E	E	E	E	E	E	E	E	E	E	

Table 6-3 PARAMeter Names in SOLs 101 Through 114 Part 3 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
OELMOPT	E	E	E	E	E	E	E	E	E	E	E
OELMSET	E	E	E	E	E	E	E	E	E	E	E
OES	E	E	E	E	E	E	E	E	E	E	E
OESE	E	E	E	E	E	E	E	E	E	E	E
OESX	E	E	E	E	E	E	E	E	E	E	E
OG	E	E	E	E							E
OGEM	E	E	E	E	E	E	E	E	E	E	E
OGPF	E	E	E	E							E
OGPS	E	E	E				E			E	E
OGRDOPT	E	E	E	E	E	E	E	E	E	E	E
OGRDSET	E	E	E	E	E	E	E	E	E	E	E
OLDSEQ	B	B	B	B	B	B	B	B	B	B	B
OMAXR	E	E	E	E	E	E	E	E	E	E	E
OMID	E	E	E			E	E	E	E	E	E
OMGLVL	E	E	E	E	E	E	E	E	E	E	E
OPCHSET	E	E	E	E	E	E	E	E	E	E	E
OPG	E		E	E		E	E		E	E	E
OPGEOM											

Table 6-4 PARAMeter Names in SOLs 101 Through 114 Part 4

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
OPGPKG											
OPPHIB											
OPPHIPA											
OPTEXIT											
OPTION		B									
OQG	E	E	E	E	E	E	E	E	E	E	E
OSETELE	E	E	E	E	E	E	E	E	E	E	E
OSETGRD	E	E	E	E	E	E	E	E	E	E	E
OSWELM	B	B	B	B	B	B	B	B	B	B	B
OSWPPT	B	B	B	B	B	B	B	B	B	B	B

Table 6-4 PARAMeter Names in SOLs 101 Through 114 Part 4 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
OUG	E	E	E	E	E	E	E	E	E	E	E
OUGCORD	E	E	E	E	E	E	E	E	E	E	E
OUGSPLIT				E		E	E		E	E	
OUMU		E									
OUNIT1	E	E	E	E	E	E	E	E	E	E	E
OUNIT2	E	E	E	E	E	E	E	E	E	E	E
OUTOPT	E	E	E	E							E
PACINFIID						B			B		
PATPLUS	E	E	E	E	E	E	E	E	E	E	E
PATVER	E	E	E	E	E	E	E	E	E	E	E
PCOMPRM	B					B	B		B	B	
PDRMSG	E	E	E	E	E	E	E	E	E	E	E
PEDGEP	E	E			E	E	E	E	E	E	
PENFN	B	B	B								
PLTMSG	E	E	E	E	E	E	E	E	E	E	E
POST	E	E	E	E	E	E	E	E	E	E	E
POSTEXT	E	E	E	E	E	E	E	E	E	E	E
POSTU	E	E	E	E	E	E	E	E	E	E	E
PREFDB							E	E	E	E	
PRINT											
PROUT	E	E	E	E	E	E	E	E	E	E	E
PRPA	E			E							
PRPHIVZ		E			E	E	E	E	E	E	E
PRPJ	E		E			E	E		E	E	E
PRTBGPDT	E	E	E	E	E	E	E	E	E	E	E
PRTCSTM	E	E	E	E	E	E	E	E	E	E	E
PRTEQXIN	E	E	E	E	E	E	E	E	E	E	E
PRTGPDT	E	E	E	E	E	E	E	E	E	E	E
PRTGPL	E	E	E	E	E	E	E	E	E	E	E
PRTGPTT	E	E	E	E	E	E	E	E	E	E	E
PRTMAXIM	E	E	E								
PRTMGG	E	E	E	E	E	E	E	E	E	E	E
PRTPG	E	E	E	E	E	E	E	E	E	E	E

Table 6-4 PARAMeter Names in SOLs 101 Through 114 Part 4 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
PRTRESLT	E	E	E	E	E	E	E	E	E	E	E
PVALINIT	B	B			B	B	B	B	B	B	
Q											
RBSETPRT	E	E	E	E	E	E	E	E	E	E	E
RESLOPT	E	E	E	E	E	E	E	E	E	E	E
RHOCP											
ROTSEKE					E			E			
RSPECTRA							E				E
RSPRINT							E				E
RSTTEMP				E							
S1	E	E	E	E			E			E	E
S1A	E	E	E	E							E
S1AG	E	E	E	E							E
S1AM	E	E	E	E							E
S1G	E	E	E	E							E
S1M	E	E	E	E							E
SCRSPEC		B									
SEMAP	B	B	B	B	B	B	B	B	B	B	B
SEMAPOPT	B	B	B	B	B	B	B	B	B	B	B
SEMAPPRT	B	B	B	B	B	B	B	B	B	B	B
SENSUOO											
SEP1XOVR	B	B	B	B	B	B	B	B	B	B	B
SEQOUT	B	B	B	B	B	B	B	B	B	B	B
SERST	B	B	B	B	B	B	B	B	B	B	B
SESDAMP		E			E	E	E	E	E	E	
SESEF		E									

Table 6-5 PARAMeter Names in SOLs 101 Through 114 Part 5

PARAMeter Name	Solution Sequence Numbers (101 through 114)									
	101	103	105	106	107	108	109	110	111	114
SHLDAMP									E	
SIGMA										
SKINOUT		E			E	E	E	E	E	E
SKPAMP										
SLOOPID										
SMALLQ	B	B	B	B	B	B	B	B	B	B
SNORM	B	B	B	B	B	B	B	B	B	B
SNORMPRT	B	B	B	B	B	B	B	B	B	B
SOFTEXIT										
SOLADJC										
SOLID	B	B	B	B	B	B	B	B	B	B
SPARSEDM	E	E	E			E	E		E	E
SPARSEDRL	E	E	E			E	E		E	E
SPARSEPH		E						E	E	E
SPDDMMAT										
SPDRRAT	E	E	E			E	E		E	E
SQSETID	E	E	E	E	E	E	E	E	E	E
SRTELTYPE	E	E	E							E
SROPT	E	E	E							E
START	B	B	B	B	B	B	B	B	B	B
STIME							E			E
SUBCASID				E						
SUBID				E						
SUPAERO										
SUPER	B	B	B	B	B	B	B	B	B	B
TABID							E			E
TABS										
TESTNEG				E						
TFSYMFAC					E	E	E	E	E	E
TINY	E	E	E							E
TOLRSC	B	B	B	B	B	B	B	B	B	B
TSTATIC										

Table 6-5 PARAMeter Names in SOLs 101 Through 114 Part 5 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
UNSYMF				B							
UNSYMKB		B			B	B	B	B	B	B	
UPDTBSH											
USETPRT	E	E	E	E	E	E	E	E	E	E	
USETSEL	E	E	E	E	E	E	E	E	E	E	
VARPHI	C										
VMOPT		E	E	E	E	E	E	E	E	E	E
VREF						B					B
VUBEAM	E	E			E	E	E	E	E	E	
VUELJUMP	E	E			E	E	E	E	E	E	
VUGJUMP	E	E			E	E	E	E	E	E	
VUHEXA	E	E			E	E	E	E	E	E	
VUPENTA	E	E			E	E	E	E	E	E	
VUQUAD4	E	E			E	E	E	E	E	E	
VUTETRA	E	E			E	E	E	E	E	E	
WTMASS	E	E	E	E	E	E	E	E	E	E	E
W3							E				E
W3FL							E				E
W4							E				E
W4FL							E				E
XFACT		B								B	
XFACTX		B								B	
XFLAG	E		E								E
XYUNIT											
ZROCMAS		E		E	E	E	E	E	E	E	
ZROVEC		E		E	E	E	E	E	E	E	

Table 6-6 PARAMeter Names in SOLs 115 Through 700 Part 1

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
ACEXTMTD										B			
ACEXTSET										B			
ACOUT										E			
ACOWEAK										B			
ACSYM										B			
ADJMETH										B			
ADPCON				E					E	E		E	
ADSTAT													
AERODOF				E		E		E			E		
AESDISC				B							B		
AESMAXIT				B							B		
AESMETH				B							B		
AESRNDM				B							B		
AESTOL				B							B		
ALPHA1		B	E		B	B	B	B	B	B	E		
ALPHA2		B	E		B	B	B	B	B	B	E		
ALPHA1FL													
ALPHA2FL													
ALTRED													
ARBMASP													
ARBMFEM													
ARBMPMS													
ARB MSS													
ARBMSTYP													
ARF													
ARS													
ASCOUP													
ASING													
AUNITS													
AUTOADJ													
AUTOGOUT	E	E	E	E	E	E	E	E	E	E	E	E	
AUTOMSET	E	E	E		E	E	E		E		E		E

Table 6-6 PARAMeter Names in SOLs 115 Through 700 Part 1 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
AUTOQSET	E	E	E	E	E	E	E	E	E	E			
AUTOSPC													
AUTOSPCR												E	
AUTOSPRT	E		E	E	E	E	E		E		E		
BAILOUT	E	E	E	E	E	E	E	E	E	E	E		
BEAMBEA													
BEIGRED			E	E		E	E			E	E		
BETA				E						E			
BIGER	E	E			E				E		E		
BIGER1	E	E			E				E		E		
BIGER2	E	E			E				E		E		
BUCKLE													
CASIEMA												E	
CASPIV											E	E	
CB1, CB2	E	E	E	E	E	E	E	E	E	E	E		
CDIF												E	
CFDIAGP	E	E	E	E	E	E	E	E	E	E	E	E	
CFRANDEL	E	E	E	E	E	E	E	E	E	E	E	E	
CHECKOUT	E	E	E	E	E	E	E	E	E	E	E	E	E
CK1, CK2, CK3	E	E	E	E	E	E	E	E	E	E	E	E	E
CLOSE	B												
CM1, CM2	E	E	E	E	E	E	E	E	E	E	E	E	
CONFAC	B	B	B	B	B	B	B	B	B	B	B	B	
COUPMASS	E	E	E	E	E	E	E	E	E	E	E	E	
CP1, CP2		E	E	E	E		E	E	E	E	E		
CQC	E												
CURV	E	E			E			E			E		
CURVPLOT	E	E			E						E	E	
CWDIAGP	E	E	E	E	E	E	E	E	E	E	E	E	
CWRANDEL	E	E	E	E	E	E	E	E	E	E	E	E	
DBALL	E		E	E	E	E	E	E	E	E	E	E	
DBCCONV	E		E	E	E	E	E	E	E	E	E	E	

Table 6-7 PARAMeter Names in SOLs 115 Through 700 Part 2

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
DBCDIAG	E		E	E	E	E	E	E	E	E	E		
DBCOVWRT	E		E	E	E	E	E	E	E	E	E		
DBDICT	B		B	B	B	B	B	B	B	B	B		
DBDRPRJ	B		B	B	B	B	B	B	B	B	B		
DBDRVER	B		B	B	B	B	B	B	B	B	B		
DBDN	E		E	E	E	E	E	E	E	E	E		
DBEXT	E	E	E	E	E	E	E	E	E	E	E		
DBRCV	E		E	E	E	E	E	E	E	E	E		
DBUP	E		E	E	E	E	E	E	E	E	E		
DDRMM							B			E			
DEACTEL											E		
DECLUMP												B	
DESPCH											E		
DESPCH1											E		
DFREQ			B				B				B		
DOPPT	E				E			E			E		
DPEPS											B		
DPHFLG													
DSNOKD											B		
DSZERO													
DV3PASS											B		

Table 6-8 PARAMeter Names in SOLs 115 Through 700 Part 3

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
DYNSPCF	E		E			E	E			E			
ENFMETH							E			E			
ENFMOTN							E			E			
EPPRT													
EPSILONT													
EPZERO													
ERPC													
ERPREFDB													
ERPRHO													
ERPRLF													
ERROR	E		E	E	E	E	E	E	E	E	E	E	
ESLFSAV											E		
ESLLCOMP											E		
ESLMOVE											E		
ESLMPC1											E		
ESLOPTEX											E		
ESLPRT											E		
ESLPRT1											E		
ESLRCF											E		
EULBND												B	
EULSTRES												B	
EXTOUT	B		B	B	B	B	B	B	B	B	B	B	
EXTRCV	E		E	E	E	E	E	E	E	E	E	E	
EXTUNIT	B		B	B	B	B	B	B	B	B	B	B	
FACTOR	B		B	B	B	B	B	B	B	B	B	B	
FBATOLR													
FBLEND												B	
FIXEDB											E		
FKSYMFAC											E		
FLEXINCR													
FLUIDMP													
FLUIDSE											B		

Table 6-8 PARAMeter Names in SOLs 115 Through 700 Part 3 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
FMULTI													B
FOLLOWK	E	E								E	E	B	
FRQDEPO							B			B			
FULLSEDR	E	E	E	E	E	E	E	E	E	E	E		
FZERO												E	
G			B	E		B	B		B	B	B		
GFL							B				B		
GEOMU	E		E	E	E	E	E	E	E	E	E		
GFL							B				B		
GPECT	E		E	E	E	E	E	E	E	E	E		
GRADMESH													B
GRDPNT	E		E	E	E	E	E	E	E	E	E	E	

Table 6-9 PARAMeter Names in SOLs 115 Through 700 Part 4

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
GUSTAERO					B	B			B				
GYROAVG													
HEATCMD												B	
HEATSTAT													
HFREQ					B	B			B				
HFREQFL						B			B				
HTOCITS													
HTOCPRT													
HTOCTOL													
IFP	E		E	E	E	E	E	E	E	E			
INREL										E			
INRLM													
IRES		E	E	E	E	E	E	E	E	E			
ITAPE										B			
IUNIT										B			
KDAMP					B					B			
KDAMPFL						B				B			
KDIAG								E			E		
K6ROT	E		E	E	E	E	E	E	E	E	E		
LANGLE			B					B	B			B	
LDSUM				X						X			
LFREQ					B	B				B			
LFREQFL						B				B			
LGDISP			E					E	E		E	B	
LMFACT												B	
LMODES					B	B				B			
LMODESFL						B				B			
LOADU	E		E	E				E	E	E	E		
LOOPID				E				E	E		E		
MACH						B							
MALIAS02												B	
MALIAS03												B	

Table 6-9 PARAMeter Names in SOLs 115 Through 700 Part 4 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MARALPHA												B	
MARAUTO												B	
MARBK105												B	
MARBK106												B	
MARC3901												B	
MARC3D												C	
MARC4401												B	
MARCASUM												B	
MARCAUTO												B	
MARCASEL												B	
MARCBEAM												B	
MARCBUG												C	
MARCBUSK												B	
MARCCBAR												B	
MARCCENT												B	
MARCCON2												B	
MARCCON3												B	
MARCCPY												B	
MARCDEF												B	
MARCDILT												B	
MARCDIS2												B	
MARCDIS3												B	
MARCDIS4												B	
MARCDMIG												B	
MARCEKND												B	
MARCEXIT												B	
MARCFEAT,N												B	
MARCFIL												B	
MARCFRIC												B	

Table 6-10 PARAMeter Names in SOLs 115 Through 700 Part 5

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MARCGAPP											B		
MARCGLUE											B		
MARCHOST											B		
MARCIAMN											B		
MARCITER											B		
MARCL001											B		
MARCLOWE											B		
MARCLUMP											B		
MARCMATT											B		
MARCMEM											B		
MARCMNF											B		
MARCMPII											B		
MARCND99											B		
MARCNOER											B		
MARCOFFT											B		
MARCONLY											B		
MARCONTF											B		
MARCOOCC											B		
MARCOPT											B		
MARCOTIM											B		
MARCPARR											B		
MARCPENT											B		
MARCPINN											B		
MARCPPOS											B		
MARCPPOST											B		
MARCPR99											B		
MARCPRES											B		
MARCPRN											B		
MARCPRNG											B		
MARCPRNR											B		
MARCPROG											B		
MARCPTH											B		

Table 6-10 PARAMeter Names in SOLs 115 Through 700 Part 5 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MARCRBAL												B	
MARCRBAR												B	
MARCRBE2												B	
MARCRBE3												B	
MARCREVR												B	
MARCRIGD												B	
MARCRUN												B	
MARCSAME												C	
MARCSCLR												B	
MARCSETS												B	
MARCSETT												B	
MARCSINC												B	
MARCSIZ3												B	
MARCSIZ4												B	
MARCSIZ5												B	
MARCSIZ6												B	
MARCSLHT												B	
MARCSOLV												B	
MARCPGCC												B	
MARCSTIFF												B	
MARCSUMY												B	
MARCT16												B	
MARCT19												B	
MARCTABL												B	
MARCTEDF												B	
MARCTEDN												B	
MARCTEMP												B	
MARCTETT												B	
MARCTIEC												B	
MARCTOL												B	
MARCTOTD												B	

Table 6-11 PARAMeter Names in SOLs 115 Through 700 Part 6

PARAMeter Name	Solution Sequence Numbers (115 through 700)											
	115	116	118	129	144	145	146	153	159	200	400	600
MARCTOTL											B	
MARCTOTT											B	
MARCTUBE											B	
MARCTVL											B	
MARCUSUB											B	
MARCVERS											B	
MARCWDIS											B	
MARCWELD											B	
MARCWIND											B	
MARELSTO											B	
MARFACEA											B	
MARFACEB											B	
MARFATAL											B	
MARGPFOR											B	
MARIBOOC											B	
MARIProj											B	
MARMPCHK											B	
MARMPCID											C	
MARNOSET											B	
MARNOT16											B	
MARPLANE											B	
MARPROCS												
MARRBAR2											B	
MARROUTT											B	
MARSHRII											B	
MARUPDAT											B	
MAXIREVV											B	
MAXLP			E						E			
MAXRATIO	E		E	E	E	E	E	E	E	E	E	
MBENDCAP											B	
MCNLPARM											B	
MCON2D3D											B	

Table 6-11 PARAMeter Names in SOLs 115 Through 700 Part 6 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MCORDUPD												B	
MCSSHLC												C	
MCSSHORR												C	
MDAREAMD												B	
MDK4OPT											B		
MDOPT14											B		
MDOTM											B		
MDOTMFAC											B		
MDREDOPT											B		
MDUMLOAD												B	
MESH								B	B	B			
METHCMRS		E	E			E	E		E	E			
MEXTRNOD												B	
MEXTSEE												C	
MFASTCMP												B	
MFORCOR1												B	
MFSKIPPP												B	
MGAPINIT												B	
MGLUETOL												B	
MHEATSHL												B	
MHEATUNT												B	
MHOUBOLT												B	
MHRED	E		E	E	E	E	E			E			
MICRO													B
MINCLDSB												B	
MINIGOA	B		B	B	B	B	B	B	B	B	B	B	
MINRECCC												B	
MINSOUTT												B	
MINVASHF												B	
MINVCITR												B	
MINVCSHF												B	
MINVCTOL												B	
MINFMAX												B	

Table 6-11 PARAMeter Names in SOLs 115 Through 700 Part 6 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MINVNMOD												B	
MLSTRAIN												B	
MLSTRAN2												B	
MMEMDETT												B	

Table 6-12 PARAMeter Names in SOLs 115 Through 700 Part 7

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MNASTLDS												B	
MODACC						B	B						
MODEL	B		B	B	B	B	B	B	B	B	B	B	
MOFFCORE												B	
MOP2TITL												B	
MOUTRCOR												B	
MPCX	B		B	B	B	B	B	B	B	B	B	B	
MPERMPRT												B	
MPTUNIT							E						
MRAFFLOR												B	
MRAFFLOT												B	
MRAFFLOW												B	
MRALIAS												B	
MRBE2SNG												B	
MRBEAMB												B	
MRBEPARAM												B	
MRBIGMEM												B	
MRBUKMTTH												B	
MRC2DADD												B	
MRCKBODY												B	
MRCKLOAD												B	
MRCONADD												B	
MRCONTAB												B	
MRCONSE												C	

Table 6-12 PARAMeter Names in SOLs 115 Through 700 Part 7 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MRCNRES												B	
MRCCONVER												B	
MRCORDS												B	
MRCWANGL												B	
MRCWELD												B	
MRDELTTT												B	
MRDISCMB												B	
MRDYNOLD												B	
MREIGMTH												B	
MREL1103												B	
MRELRB												B	
MRELSPCD												B	
MRENUELE												B	
MRENUGRD												B	
MRENUMBR												B	
MRENUMMT												B	
MRESTALL												B	
MRESULTS												B	
MREVPLST												B	
MRFINITE												B	
MRFOLLO2												B	
MRFOLLOW												B	
MRFOLLOW1												B	
MRFOLLOW3												B	
MRFOLLOW4												B	
MRGAPUSE												B	
MRHYPMID												B	
MRITTYPE												B	
MRMAT8A3												B	
MRMAT8E3												B	
MRMAT8N1												B	
MRMAT8N3												B	
MRMAXISZ												B	

Table 6-12 PARAMeter Names in SOLs 115 Through 700 Part 7 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MRMAXNUM												B	
MRMEMSUM												B	
MRMTXKGG												B	
MRNOCOR												B	
MRORINTS												B	
MRPARALL												B	
MRELAST												B	
MRPBUSHT												B	

Table 6-13 PARAMeter Names in SOLs 115 Through 700 Part 8

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MRPIDCHK												B	
MRPLLOAD4												B	
MRPLLOAD4R												B	
MRPLSUPD												B	
MRPREFER												B	
MRPSHCMP												B	
MRPSHELL												B	
MRRELNOD												B	
MRRRCFILE												B	
MRSETNAM												B	
MRSPAWN2												B	
MRSPLINE												B	
MRSPRING												B	
MRT16STP												B	
MRTABLIM												B	
MRTABLS1												B	
MRTABLS2												B	
MRTABPRC												B	
MRTFINAL												B	
MRTIMING												B	

Table 6-13 PARAMeter Names in SOLs 115 Through 700 Part 8 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MRTTHREAD												B	
MRTSHEAR												B	
MRUSDATA												B	
MSOLMEM												B	
MSPCCCHK												B	
MSPCKEEP												B	
MSPEEDCB												B	
MSPEEDCW												B	
MSPEEDOU												B	
MSPEEDP4												B	
MSPEEDSE												B	
MSOLMEM, MBYTE												B	
MSTFBEAM												B	
MTABINGNR												B	
MTABLD1M												B	
MTABLD1T												B	
MTEMPDWN												B	
MTLD1SID												B	
MULRFORC												B	
MUMPSOLV												B	
MUSBKEEP												B	
MWINQUOT												B	
MWLDGSGA												B	
MXLAGM1												B	
NASPRT												B	
NDAMP			E							E		E	
NDAMPM												E	
NEWSET													
NINTPTS	E			E			E		E		E		
NLAYERS			E									E	
NLHTLS							E					E	
NLMAX												E	

Table 6-13 PARAMeter Names in SOLs 115 Through 700 Part 8 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
NLMIN									E	E			
NMLOOP													
NOCOMPS	E				E			E		E	E		
NOELOF	E				E			E		E			
NOELOP	E				E			E		E			
NOGPF	E				E			E		E			
NOMSGSTR	E			E			E		E				
NONCUP									B				
NQSET	E		E	E	E	E	E		E				
NLTOL									E		B		
NUMOUT	E				E			E		E			
NUMOUT1	E			E			E		E				

Table 6-14 PARAMeter Names in SOLs 115 Through 700 Part 9

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
NUMOUT2	E				E			E		E			
OCMP	E									E	E		
OEE	E									E	E	E	
OEF	E		E	E	E	E	E	E	E	E	E	E	
OEFX	E		E	E	E	E	E	E	E	E	E	E	
OELMOPT	E	E	E	E	E	E	E	E	E	E	E	E	
OELMSET	E	E	E	E	E	E	E	E	E	E	E	E	
OES	E		E	E	E	E	E	E	E	E	E	E	
OESE	E		E	E	E	E	E	E	E	E	E	E	
OESX	E		E	E	E	E	E	E	E	E	E	E	
OG	E				E			E		E	E	E	
OGEM	E		E	E	E	E	E	E	E	E	E	E	
OGPF	E				E					E	E	E	
OGPS	E				E					E			
OGRDOPT	E	E	E	E	E	E	E	E	E	E	E	E	
OGRDSET	E	E	E	E	E	E	E	E	E	E	E	E	

Table 6-14 PARAMeter Names in SOLs 115 Through 700 Part 9 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
OLDSEQ	B		B	B	B	B	B	B	B	B	B	B	
OMAXR	E		E	E	E	E	E	E	E	E	E	E	
OMID	E	E	E	E	E	E	E	E	E	E	E	E	
OMSGLVL	E	E	E	E	E	E	E	E	E	E	E	E	
OPCHSET	E	E	E	E	E	E	E	E	E	E	E	E	
OPG			E	E		E	E		E	E	E	E	E
OPGEOM					B	B	B			B			
OPGPKG					B	B	B			B			
OPPHIB						B	B			B			
OPPHIPA						B	B			B			
OPTEXIT										B			
OPTION	B												
OQG	E		E	E	E	E	E	E	E	E	E	E	E
OSETELE	E	E	E	E	E	E	E	E	E	E	E	E	
OSETGRD	E	E	E	E	E	E	E	E	E	E	E	E	
OSWELM	B	B	B	B	B	B	B			B	B	B	
OSWPPT	B	B	B	B	B	B	B			B	B	B	
OUG	E		E	E	E	E	E	E	E	E	E	E	E
OUGCORD	E		E	E	E	E	E	E	E	E	E	E	
OUGSPLIT			E	E			E		E	E	E	E	
OUMU	E									E			
OUNIT1	E		E	E	E	E	E	E	E	E	E	E	
OUNIT2	E		E	E	E	E	E	E	E	E	E	E	
OUTOPT	E			E				E		E	E	E	
PARTMEM													
PATPLUS	E		E	E	E	E	E	E	E	E	E	E	
PATVER	E		E	E	E	E	E	E	E	E	E	E	
PDRMSG	E		E	E	E	E	E	E	E	E	E	E	
PEDGEP													
PENFN											B		
PH2OUT											B		
PLTMSG	E		E	E	E	E	E	E	E	E	E	E	
POST	E		E	E	E	E	E	E	E	E	E	E	E*

Table 6-14 PARAMeter Names in SOLs 115 Through 700 Part 9 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
POSTEXT	E	E	E	E	E	E	E	E	E	E	E		
POSTU	E		E	E	E	E	E	E	E	E	E		
PREFDB												E	
PRINT						B					B		
PROUT	E		E	E	E	E	E	E	E	E	E		
PRPA					E			E			E		
PRPHIVZ	E		E	E	E	E	E		E	E			
PRPJ			E		E		E	E	E	E			
PRTCSTM	E		E	E	E	E	E	E	E	E	E	E	
PRTEQXIN	E		E	E	E	E	E	E	E	E	E	E	
PRTGPDT	E		E	E	E	E	E	E	E	E	E	E	
PRTGPL	E		E	E	E	E	E	E	E	E	E	E	
PRTGPTT	E		E	E	E	E	E	E	E	E	E	E	
PRTMAXIM	E		E	E	E	E	E	E	E	E			
PRTMGG	E		E	E	E	E	E	E	E	E	E	E	
PRTPG	E		E	E	E	E	E	E	E	E	E		
PSENPCH										B			

Table 6-15 PARAMeter Names in SOLs 115 Through 700 Part 10

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
PVALINIT											B		
Q							B						
RADMOD								B	B		B		
RBSETPRT	E	E	E										
RCONTACT												B	
RESLOPT	E	E	E	E	E	E	E	E	E	E	E		
RHOCP													B
RKSCHEME													B
RMSINT													
ROHYDRO													B
ROMULTI													B

Table 6-15 PARAMeter Names in SOLs 115 Through 700 Part 10 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
ROSTR													B
RSPECTRA													
RSPRINT													
RSTTEMP													
S1	E				E			E		E	E		
S1A	E				E			E		E	E		
S1AG	E				E			E		E	E		
S1AM	E				E			E		E	E		
S1G	E				E			E		E	E		
S1M	E				E			E		E	E		
SCRSPEC	B												
SEMAP	B		B	B	B	B	B	B	B	B	B	B	
SEMAPOPT	B		B	B	B	B	B	B	B	B	B	B	
SEMAPPRT	B		B	B	B	B	B	B	B	B	B	B	
SENSUOO													B
SEP1XOVR	B		B	B	B	B	B	B	B	B	B	B	
SEQOUT	B		B	B	B	B	B	B	B	B	B	B	
SERST	B		B	B	B	B	B	B	B	B	B	B	
SESDAMP	E	E	E	E		E							
SESEF	E												
SHLDAMP													E
SIGMA													B
SKINOUT													E
SKPAMP						B	B						B
SLOOPID				E									
SMALLQ	B		B	B	B	B	B	B	B	B	B	B	
SNORM	B		B	B	B	B	B	B	B	B	B	B	
SNORMPRT	B		B	B	B	B	B	B	B	B	B	B	
SOFTEXIT													B
SOLADJC													B
SOLID	B		B	B	B	B	B	B	B	B	B	B	
SPARSEDM					E								E
SPARSEDRL					E								E

Table 6-15 PARAMeter Names in SOLs 115 Through 700 Part 10 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
SPARSEPH						E	E			E			
SPDMRAT										E			
SPDRRAT					E					E			
SQSETID	E	E	E	E	E	E	E	E	E	E	E	E	
STEPFCT													B
STEPFCTL													B
SRTELTYP	E				E			E		E			
SRTOPT	E				E			E		E			
START	B		B		B	B	B	B	B	B	B	B	
STIME				E					E				
STRUCTMP										E			
SUBCASID												E	
SUBID								E				E	
SUPAERO					E	E	E				E		
SUPDOF					B								
SUPER	B		B	B	B	B	B	B	B	B	B	B	
TABID													
TABS									E	E	E		
TESTNEG									E			E	
TFSYMFAC			E	E		E	E	E	E	E	E	E	

Table 6-16 PARAMeter Names in SOLs 115 Through 700 Part 11

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
TINY	E				E			E		E			
TOLRSC	B		B	B	B	B	B	B	B	B	B		
TOPOCONV												E	
TSTATIC				E					E		E	E	
UNSYMFM											B		
UNSYMKB	B		B	B		B	B			B	B		
UPDTBSH										E			
USETPRT	E		E	E	E	E	E	E	E	E	E		

Table 6-16 PARAMeter Names in SOLs 115 Through 700 Part 11 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
VARPHI										C			
VELCUT												B	
VMOPT	E		E	E	E	E	E	E	E	E	E		
VREF						B				B			
VUBEAM													
VUELJUMP													
VUGJUMP													
VUHEXA													
VUPENTA													
VUQUAD4													
VUTETRA													
WTMASS	E		E	E	E	E	E	E	E	E	E	B	
W3				B					E	B	E	B	
W3FL										B			
W4				B					E	B	E	B	
W4FL										B			
XFLAG					E						E		
XYUNIT					E						E		
ZROCMAS	E		E	E	E	E	E			E	E		
ZROVEC	E		E	E	E	E	E			E	E		

7

Item Codes

- Item Code Description
- Element Stress (or Strain) Item Codes
- Element Force Item Codes
- Fluid Virtual Mass Pressure Item Codes
- 2D Slideline and 3D Surface Contact Item Codes
- Element Strain Energy Item Codes
- Fatigue Item Codes for LOC=ELEM on FTGPARM
- Fatigue Item Codes for LOC=NODE or NODA on FTGPARM
- Item Codes for Fatigue Analysis of Spot Welds
- Item Codes for Fatigue Analysis of Seam Welds
- Equivalent Radiated Power (ERP) Item Codes

Item Code Description

Item codes are integer numbers assigned to specific output quantities; such as, the third translational component of displacement, the major principal stress at Z1 in a CQUAD4 element, or the torque in a CBAR element. Item codes are specified on the following input statements:

- DRESP1 entry for Design Sensitivity and Optimization (SOL 200).
- X-Y Plotting commands. See [Plotting](#) in the *MSC Nastran Reference Guide*.
- DTI,INDTA entry for stress sorting.

The following tables provide item codes for:

- [Table 7-1](#). Element Stress or Strain.
- [Table 7-6](#). Element Force.
- [Table 7-8](#). Fluid Virtual Mass Pressure.
- [Table 7-9](#). Heat Transfer Flux.
- [Table 7-10](#). Slideline Contact Output.
- [Table 7-13](#). Element Strain Energy Item Codes.
- [Table 7-14](#). Fatigue Item Codes (LOC=ELEM)
- [Table 7-15](#) Fatigue Item Codes (LOC=NODE or NODA)
- [Table 7-16](#). Item Codes for Fatigue Analysis of Spot Welds
- [Table 7-17](#). Item Codes for Fatigue Analysis of Seam Welds
- [Table 7-18](#) Equivalent Radiated Power (ERP) Item Codes

The following superscripts appear in the tables and indicate:

1. Data for components marked with the symbol (1) are included in the data block MES and MEF outputs from modules DRMH1/DRMH3 and DRMS1. DRMH1/DRMH3 are used in EXTSEOUT Case Control processing. (See [MSC Nastran DMAP Programmer's Guide](#).)
2. Composite Element Stresses and Failure Indices.

Element Stress (or Strain) Item Codes

All item codes refer to stresses (or strains) unless otherwise denoted.

If output is magnitude/phase, the magnitude replaces the real part, and the phase replaces the imaginary part. Strain item codes are equivalent to stress item codes. However, strain is computed for only some elements. See [Table 3-1](#) in the *MSC Nastran Reference Guide*.

Table 7-1 Element Stress-Strain Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CAXIF2 (47)	2 ¹	Radial axis	2 ¹	Radial axis	RM
	3 ¹	Axial axis	3 ¹	Axial axis	RM
	4 ¹	Tangential edge	4 ¹	Tangential edge	RM
	5 ¹	Circumferential edge	5 ¹	Circumferential edge	RM
			6 ¹	Radial axis	IP
			7 ¹	Axial axis	IP
			8 ¹	Tangential edge	IP
			9 ¹	Circumferential edge	IP
CAXIF3 (48)	2 ¹	Radial centroid	2 ¹	Radial centroid	RM
	3 ¹	Circumferential centroid	3 ¹	Circumferential centroid	RM
	4 ¹	Axial centroid	4 ¹	Axial centroid	RM
	5 ¹	Tangential edge 1	5 ¹	Tangential edge 1	RM
	6 ¹	Circumferential edge 1	6 ¹	Circumferential edge 1	RM
	7 ¹	Tangential edge 2	7 ¹	Tangential edge 2	RM
	8 ¹	Circumferential edge 2	8 ¹	Circumferential edge 2	RM
	9 ¹	Tangential edge 3	9 ¹	Tangential edge 3	RM
	10 ¹	Circumferential edge 3	10 ¹	Circumferential edge 3	RM
			11 ¹	Radial centroid	IP
			12 ¹	Circumferential centroid	IP
			13 ¹	Axial centroid	IP
			14 ¹	Tangential edge 1	IP
			15 ¹	Circumferential edge 1	IP
			16 ¹	Tangential edge 2	IP
			17 ¹	Circumferential edge 2	IP
			18 ¹	Tangential edge 3	IP
			19 ¹	Circumferential edge 3	IP
CAXIF4 (49)	2 ¹	Radial centroid	2 ¹	Radial centroid	RM
	3 ¹	Circumferential centroid	3 ¹	Circumferential centroid	RM
	4 ¹	Axial centroid	4 ¹	Axial centroid	RM
	5 ¹	Tangential edge 1	5 ¹	Tangential edge 1	RM
	6 ¹	Circumferential edge 1	6 ¹	Circumferential edge 1	RM

Table 7-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	7 ¹	Tangential edge 2	7 ¹	Tangential edge 2	RM
	8 ¹	Circumferential edge 2	8 ¹	Circumferential edge 2	RM
	9 ¹	Tangential edge 3	9 ¹	Tangential edge 3	RM
	10 ¹	Circumferential edge 3	10 ¹	Circumferential edge 3	RM
	11 ¹	Tangential edge 4	11 ¹	Tangential edge 4	RM
	12 ¹	Circumferential edge 4	12 ¹	Circumferential edge 4	RM
			13	Radial centroid	IP
			14	Circumferential centroid	IP
			15	Axialcentroid	IP
			16	Tangential edge 1	IP
			17	Circumferential edge 1	IP
			18	Tangential edge 2	IP
			19	Circumferential edge 2	IP
			20	Tangential edge 3	IP
			21	Circumferential edge 3	IP
			22	Tangential edge 4	IP
			23	Circumferential edge 4	IP
CAXISYM (241)	2	Z1-Fiber distance 1			
Nonlinear	3	Stress-X			
	4	Stress-Y			
	5	Stress-XY			
	6	Strain-X			
	7	Strain-Y			
	8	Strain XY			
	9-15	Item 9 through 15 repeated for fiber distance Z2			
CBAR (34)	2 ¹	End A-Point C	2 ¹	End A-Point C	RM
	3 ¹	End A-Point D	3 ¹	End A-Point D	RM
	4 ¹	End A-Point E	4 ¹	End A-Point E	RM
	5 ¹	End A-Point F	5 ¹	End A-Point F	RM
	6 ¹	Axial	6 ¹	Axial	RM
	7	End A maximum	7 ¹	End A-Point C	IP
	8	End A minimum	8 ¹	End A-Point D	IP

Table 7-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	9	Safety margin in tension	9 ¹	End A-Point E	IP
	10 ¹	End B-Point C	10 ¹	End A-Point F	IP
	11 ¹	End B-Point D	11 ¹	Axial	IP
	12 ¹	End B-Point E	12 ¹	End B-Point C	RM
	13 ¹	End B-Point F	13 ¹	End B-Point D	RM
	14	End B maximum	14 ¹	End B-Point E	RM
	15	End B minimum	15 ¹	End B-Point F	RM
	16	Safety margin in compression	16 ¹	End B-Point C	IP
			17 ¹	End B-Point D	IP
			18 ¹	End B-Point E	IP
			19 ¹	End B-Point F	IP
CBAR (100)	2	Station Distance/Length	2	Station Distance/Length	RM
Intermediate Stations	3	Point C	3	Point C	RM
	4	Point D	4	Point D	RM
	5	Point E	5	Point E	RM
	6	Point F	6	Point F	RM
	7	Axial	7	Axial	RM
	8	Maximum	8	Maximum	RM
	9	Minimum	9	Minimum	RM
	10	Margin of Safety	10	Point C	IP
		(Item codes above are given for End A. For codes 2 through 10 at intermediate stations add $(K-1)*9$ where K is the station number, and for codes at End B, K=number of stations plus 1.)	11	Point D	IP
			12	Point E	IP
			13	Point F	IP
			14	Axial	IP
			15	Maximum	IP
			16	Minimum	
				(Item codes above are given for End A. For codes 2 through 16 at intermediate stations add $(K-1)*15$ where K is the station number, and for codes at End B, K=number of stations plus 1.)	

Table 7-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CBAR (238) Arbitrary Cross Section	2	Station Distance/Length			
	3	Max Axial Stress			
	4	Min Axial Stress			
	5	Max Shear Stress in xy			
	6	Min Shear Stress in xy			
	7	Max Shear Stress in zx			
	8	Min Shear Stress in zx			
	9	Max vonMises Stress		(Not Supported)	
CBEAM (2) Linear	2	External grid point ID	2	External grid point ID	
	3	Station distance/length	3	Station distance/length	
	4 ¹	Long. Stress at Point C	4 ¹	Long. Stress at Point C	RM
	5 ¹	Long. Stress at Point D	5 ¹	Long. Stress at Point D	RM
	6 ¹	Long. Stress at Point E	6 ¹	Long. Stress at Point E	RM
	7 ¹	Long. Stress at Point F	7 ¹	Long. Stress at Point F	RM
	8	Maximum stress	8 ¹	Long. Stress at Point C	IP
	9	Minimum stress	9 ¹	Long. Stress at Point D	IP
	10	Safety margin in tension	10 ¹	Long. Stress at Point E	IP
	11	Safety margin in compression	11 ¹	Long. Stress at Point F	IP
		(Item codes are given for end A. Addition of the quantity (K-1)10 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.)		(Item codes are given for end A. Addition of the quantity (K-1)10 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.)	
CBEAM (94) Nonlinear	2	External grid point ID		Not applicable	
	3	C (Character)			
	4	Long. Stress at point C			
	5	Equivalent stress			
	6	Total strain			
	7	Effective plastic strain			
	8	Effective creep strain			
		(Item codes 3 through 8 are repeated for points D, E, and F. Then the entire record (from 2 through N) is repeated for end B of the element.)			

Table 7-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CBEAM (239) Arbitrary Cross Section	2	Station Distance/Length			
	3	Max Axial Stress			
	4	Min Axial Stress			
	5	Max Shear Stress in xy			
	6	Min Shear Stress in xy			
	7	Max Shear Stress in xz			
	8	Min Shear Stress in xz			
	9	Max von Mises Stress		(Not supported)	
	2	External grid point ID	2	External grid point ID	
	3	Normal stress at point C	3	Normal stress at point C	RM
CBEAM3 (184) Linear	4	Normal stress at point D	4	Normal stress at point D	RM
	5	Normal stress at point E	5	Normal stress at point E	RM
	6	Normal stress at point F	6	Normal stress at point F	RM
	7	Maximum stress	7	Normal stress at point C	IP
	8	Minimum stress	8	Normal stress at point D	IP
	9	Safety margin in tension	9	Normal stress at point E	IP
	10	Safety margin in compression	10	Normal stress at point F	IP
	11	Shear stress in y-direction at point C	11	Shear stress in y-direction at point C	RM
	12	Shear stress in y-direction at point D	12	Shear stress in y-direction at point D	RM
	13	Shear stress in y-direction at point E	13	Shear stress in y-direction at point E	RM
	14	Shear stress in y-direction at point F	14	Shear stress in y-direction at point F	RM
	15	Maximum shear stress in y-direction	15	Shear stress in y-direction at point C	IP
	16	Minimum shear stress in y-direction	16	Shear stress in y-direction at point D	IP
	17	(Not applicable)	17	Shear stress in y-direction at point E	IP
	18	(Not applicable)	18	Shear stress in y-direction at point F	IP
	19	Shear stress in z-direction at point C	19	Shear stress in z-direction at point C	RM

Table 7-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	20	Shear stress in z-direction at point D	20	Shear stress in z-direction at point D	RM
	21	Shear stress in z-direction at point E	21	Shear stress in z-direction at point E	RM
	22	Shear stress in z-direction at point F	22	Shear stress in z-direction at point F	RM
	23	Maximum shear stress in z-direction	23	Shear stress in z-direction at point C	IP
	24	Minimum shear stress in z-direction	24	Shear stress in z-direction at point D	IP
	25	(Not applicable)	25	Shear stress in z-direction at point Z	IP
	26	(Not applicable) (Item codes are given for end A. they are repeated for end B and mid-node C, respectively)	26	Shear stress in z-direction at point F (Item codes are given for end A. They are repeated for end B and mid-node C, respectively)	IP
CBEND (69)	2	External grid point ID	2	External grid point ID	
	3	Circumferential angle	3	Circumferential angle	
	4 ¹	Long. Stress at Point C	4 ¹	Long. Stress at Point C	RM
	5 ¹	Long. Stress at Point D	5 ¹	Long. Stress at Point D	RM
	6 ¹	Long. Stress at Point E	6 ¹	Long. Stress at Point E	RM
	7 ¹	Long. Stress at Point F	7 ¹	Long. Stress at Point F	RM
	8	Maximum stress	8 ¹	Long. Stress at Point C	IP
	9	Minimum stress	9 ¹	Long. Stress at Point D	IP
	10	Safety margin in tension	10 ¹	Long. Stress at Point E	IP
	11	Safety margin in compression (Item codes are given for end A. Item codes 12 through 21 point to the same information for end B.)	11 ¹	Long. Stress at Point F (Item codes are given for end A. Item codes 12 through 21 point to the same information for end B.)	IP
CBUSH (102)	2 ¹	Translation-x	2 ¹	Translation-x	R
	3 ¹	Translation-y	3 ¹	Translation-y	R
	4 ¹	Translation-z	4 ¹	Translation-z	R
	5 ¹	Rotation-x	5 ¹	Rotation-x	R
	6 ¹	Rotation-y	6 ¹	Rotation-y	R
	7 ¹	Rotation-z	7 ¹	Rotation-z	R
			8 ¹	Translation-x	I

Table 7-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
			9 ¹	Translation-y	I
			10 ¹	Translation-z	I
			11 ¹	Rotation-x	I
			12 ¹	Rotation-y	I
			13 ¹	Rotation-z	I
CBUSH1D (40)	1	Element ID			
	2	Axial force			
	3	Axial displacement			
	4	Axial velocity		Not applicable	
	5	Axial stress			
	6	Axial strain			
	7				
	8				

Table 7-2 Element Stress-Strain Item Codes Part 2

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
CCONEAX (35)	2	Harmonic or point angle		Not applicable	
	3	Z1=Fiber Distance 1			
	4 ¹	Normal v at Z1			
	5 ¹	Normal u at Z1			
	6 ¹	Shear uv at Z1			
	7	Shear angle at Z1			
	8	Major principal at Z1			
	9	Minor principal at Z1			
	10	Maximum shear at Z1			
	11	Z2= Fiber Distance 2			
	12 ¹	Normal v at Z2			
	13 ¹	Normal u at Z2			
	14 ¹	Shear uv at Z2			
	15	Shear angle at Z2			

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
	16	Major principal at Z2			
	17	Minor principal at Z2			
	18	Maximum shear at Z2			
CDUM3	2 ¹	S1	2 ¹	S1	RM
thru	3 ¹	S2	3 ¹	S2	RM
CDUM9 (55-61)	4 ¹	S3	4 ¹	S3	RM
	5 ¹	S4	5 ¹	S4	RM
	6 ¹	S5	6 ¹	S5	RM
	7 ¹	S6	7 ¹	S6	RM
	8 ¹	S7	8 ¹	S7	RM
CELAS1 (11)	2 ¹	Stress	2 ¹	Stress	RM
			3 ¹	Stress	IP
CELAS2 (12)	2 ¹	Stress	2 ¹	Stress	RM
			3 ¹	Stress	IP
CELAS3 (13)	2 ¹	Stress	2 ¹	Stress	RM
			3 ¹	Stress	IP
CGAP (86)	2	Normal x		Not applicable	
	3	Shear y			
	4	Shear z			
	5	Axial u			
	6	Shear v			
	7	Shear w			
	8	Slip v			
	9	Slip w			
CHEXA (67)	2	Stress coordinate system	2	Stress coordinate system	
Linear	3	Coordinate type (Character)	3	Coordinate type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID (0=center)	5	External grid ID (0=center)	
	6 ¹	Normal x	6 ¹	Normal x	RM
	7 ¹	Shear xy	7 ¹	Normal y	RM
	8	First principal	8 ¹	Normal z	RM
	9	First principal x cosine	9 ¹	Shear xy	RM

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	10	Second principal x cosine	10 ¹	Shear yz	RM
	11	Third principal x cosine	11 ¹	Shear zx	RM
	12	Mean pressure	12 ¹	Normal x	IP
	13	von Mises or octahedral shear stress	13 ¹	Normal y	IP
	14 ¹	Normal y	14 ¹	Normal z	IP
	15 ¹	Shear yz	15 ¹	Shear xy	IP
	16	Second principal	16 ¹	Shear yz	IP
	17	First principal y cosine	17 ¹	Shear zx	IP
	18	Second principal y cosine	18-121	Items 5 through 17 repeated for 8 corners	
	19	Third principal y cosine			
	20 ¹	Normal z			
	21 ¹	Shear zx			
	22	Third principal			
	23	First principal z cosine			
	24	Second principal z cosine			
	25	Third principal z cosine			
	26-193	Items 5 through 25 repeated for 8 corners			
CHEXA (93) Nonlinear	2	Stress coordinate system			
	3	Grid/Gauss			
	4	Number of active points			
	5	External grid ID (0=center)			
	6	Stress-X			
	7	Stress-Y			
	8	Stress-Z			
	9	Stress-XY			
	10	Stress-YZ			
	11	Stress-ZX			
	12	Equivalent stress			
	13	Effective plastic strain		Not applicable	
	14	Effective creep strain			

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	15	Strain-X			
	16	Strain-Y			
	17	Strain-Z			
	18	Strain-XY			
	19	Strain-YZ			
	20	Strain-ZX			
	21-148	Items 5 through 20 Repeated for 8 corners			
CHEXA FD (202) Nonlinear Finite Deformation with 8 grid points	2	Grid/Gauss			
	3	Grid/Gauss ID (0=center)			
	4	Cauchy stress-X			
	5	Cauchy stress-Y			
	6	Cauchy stress-Z			
	7	Cauchy stress-XY			
	8	Cauchy stress-YZ			
	9	Cauchy stress-ZX			
	10	Pressure $p = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$		Not applicable	
	11	Volumetric strain J-1			
	12	Logarithmic strain-X			
	13	Logarithmic strain-Y			
	14	Logarithmic strain-Z			
	15	Logarithmic strain-XY			
	16	Logarithmic strain-YZ			
	17	Logarithmic strain-ZX			
	18-122	Items 3 through 17 repeated for 7 Gauss/grid points			
CHEXA FD (207) Nonlinear Finite Deformation with 20 grid points	2-17	Same as CHEXA FD (202)			
	18-407	Items 3 through 17 repeated for 26 Gauss points		Not applicable	
CIFHEX (65) Nonlinear	2	Grid/Gauss			
	3	External Grid ID (0 = Center)			
	4	Normal Stress			

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	5	Shear Stress 1			
	6	Shear Stress 2			
	7	Normal Strain			
	8	Shear Strain 1			
	9	Shear Strain 2			
	10	Damage Value			
	11-74	Items 3 through 10 repeated for 6 Corners			
CIPENT (66) Nonlinear	2	Grid/Gauss			
	3	External Grid ID (0 = Center)			
	4	Normal Stress			
	5	Shear Stress 1			
	6	Shear Stress 2			
	7	Normal Strain			
	8	Shear Strain 1			
	9	Shear Strain 2			
	10	Damage Value			
	11-58	Items 3 through 10 repeated for 6 Corners			
CIFQDX (73) Nonlinear	2	Grid/Gauss			
	3	External Grid ID (0 = Center)			
	4	Normal Stress			
	5	Shear Stress			
	6	Normal Strain			
	7	Shear Strain			
	8	Damage Value			
	9-32	Items 3 through 8 repeated for 4 Corners			
CIFQUAD (63) Nonlinear	2	Grid/Gauss			
	3	External Grid ID (0 = Center)			
	4	Normal Stress			
	5	Shear Stress			

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	6	Normal Strain			
	7	Shear Strain			
	8	Damage Value			
	9-32	Items 3 through 8 repeated for 4 Corners			
CONROD (10)	2 ¹	Axial stress	2 ¹	Axial stress	RM
Linear	3	Axial safety margin	3 ¹	Axial stress	IP
	4 ¹	Torsional stress	4 ¹	Torsional stress	RM
	5	Torsional safety margin	5 ¹	Torsional stress	IP
CONROD (92)	2	Axial stress			
Nonlinear	3	Equivalent stress			
	4	Total strain			
	5	Effective plastic strain		Not applicable	
	6	Effective creep strain			
	7	Linear torsional stress			
CPENTA (68)	2	Stress coordinate system	2	Stress coordinate system	
Linear	3	Coordinate type (Character)	3	Coordinate type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID	5	External grid ID	
		(0=center)		(0=center)	
	6 ¹	Normal x	6 ¹	Normal x	RM
	7 ¹	Shear xy	7 ¹	Normal y	RM
	8	First principal	8 ¹	Normal z	RM
	9	First principal x cosine	91	Shear xy	RM
	10	Second principal x cosine	10 ¹	Shear yz	RM
	11	Third principal x cosine	11 ¹	Shear zx	RM
	12	Mean pressure	12 ¹	Normal x	IP
	13	von Mises or Octahedral shear stress	13 ¹	Normal y	IP
	14 ¹	Normal y	14 ¹	Normal z	IP
	15 ¹	Shear yz	15 ¹	Shear xy	IP
	16	Second principal	16 ¹	Shear yz	IP
	17	First principal y cosine	17 ¹	Shear zx	IP

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	18	Second principal y cosine	18-95	Items 5 through 17 repeated for 6 corners	
	19	Third principal y cosine			
	20 ¹	Normal z			
	21 ¹	Shear zx			
	22	Third principal			
	23	First principal z cosine			
	24	Second principal z cosine			
	25	Third principal z cosine			
	26-151	Items 5 through 25 repeated for 6 corners			
CPENTA (91) Nonlinear	2	Stress coordinate system			
	3	Grid/Gauss			
	4	Number of active points			
	5	External grid ID (0=center)			
	6	Normal x stress			
	7	Normal y stress			
	8	Normal z stress			
	9	Shear xy stress			
	10	Shear yz stress			
	11	Shear zx stress			
	12	Equivalent stress		Not applicable	
	13	Effective plastic strain			
	14	Effective creep strain			
	15	Normal x strain			
	16	Normal y strain			
	17	Normal z strain			
	18	Shear xy strain			
	19	Shear yz strain			
	20	Shear zx strain			
	21-116	Items 5 through 20 Repeated for 6 corners			

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CPENTA FD (204) Nonlinear Finite Deformation with 6 grid points	2-17	Same as CHEXA FD (201)			
	18-92	Items 3 through 17 repeated for 5 Gauss points		Not applicable	
CPENTA FD (209) Nonlinear Finite Deformation with 15 grid points	2-17	Same as CHEXA FD (201)			
	18-317	Items 3 through 17 repeated for 20 Gauss points		Not applicable	

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
CPYRAM (242) Linear	2	Stress coordinate system	2	Stress coordinate system	
	3	Coordinate Type (Character)	3	Coordinate Type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID (0=centre)	5	External grid ID (0=centre)	
	6	Normal x	6	Normal x	RM
	7	Shear xy	7	Normal y	RM
	8	First principal	8	Normal z	RM
	9	First principal x cosine	9	Shear xy	RM
	10	Second principal x cosine	10	Shear yz	RM
	11	Third principal x cosine	11	Shear zx	RM
	12	Mean pressure	12	Normal x	IP
	13	von Mises or octahedral shear stress	13	Normal y	IP
	14	Normal y	14	Normal z	IP
	15	Shear yz	15	Shear xy	IP
	16	Second principal	16	Shear yz	IP
	17	First principal y cosine	17	Shear zx	IP
	18	Second principal y cosine	18-82	Items 5 through 17 repeated for 5 corners	
	19	Third principal y cosine			
	20	Normal z			
	21	Shear zx			
	22	Third Principal			
	23	First principal z cosine			
	24	Second principal z cosine			
	25	Third principal z cosine			
	26-130	Items 5 through 25 repeated for 5 corners			

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
CPYRAM (242) Non Linear	2	Stress coordinate system	Not Applicable		
	3	Grid/Gauss			
	4	Number of active points			
	5	External grid ID (0=centre)			
	6	Stress X			
	7	Stress Y			
	8	Stress Z			
	9	Stress XY			
	10	Stress YZ			
	11	Stress ZX			
	12	Equivalent Stress			
	13	Effective plastic strain			
	14	Effective creep strain			
	15	Strain X			
	16	Strain Y			
	17	Strain Z			
	18	Strain XY			
	19	Strain YZ			
	20	Strain ZX			
	21-100	Items 5 through 20 repeated for 5 corners			

Table 7-3 Element Stress-Strain Item Codes Part 3

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
CQUAD4 (33) Linear	2	Z1=Fiber distance 1	2	Z1=Fiber distance 1	RM
	3 ¹	Normal x at Z1	3 ¹	Normal x at Z1	
	4 ¹	Normal y at Z1	4 ¹	Normal x at Z1	
	5 ¹	Shear xy at Z1	5 ¹	Normal y at Z1	
	6	Shear angle at Z1	6 ¹	Normal y at Z1	

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
	7	Major principal at Z1	7 ¹	Shear xy at Z1	RM
	8	Minor principal at Z1	8 ¹	Shear xy at Z1	IP
	9	von Mises or maximum shear at Z1	9	Z2=Fiber distance 2	
	10	Z2=Fiber distance 2	10 ¹	Normal x at Z2	RM
	11 ¹	Normal x at Z2	11 ¹	Normal x at Z2	IP
	12 ¹	Normal y at Z2	12 ¹	Normal y at Z2	RM
	13 ¹	Shear xy at Z2	13 ¹	Normal y at Z2	IP
	14	Shear angle at Z2	14	Shear xy at Z2	RM
	15	Major principal at Z2	15	Shear xy at Z2	IP
	16	Minor principal at Z2			
	17	von Mises or maximum shear at Z2			
CQUAD4 (90) Nonlinear	2	Z1=Fiber distance 1 (plane stress only)			
	3	Stress-X (at Z1, if plane stress)			
	4	Stress-Y (at Z1, if plane stress)			
	5	Stress-Z (plane strain only)			
	6	Stress-XY (at Z1, if plane stress)			
	7	Equivalent stress (at Z1, if plane stress)			
	8	Plastic strain (at Z1, if plane stress)			
	9	Creep strain (at Z1, if plane stress)		Not applicable	
	10	Strain-X (at Z1, if plane stress)			
	11	Strain-Y (at Z1, if plane stress)			
	12	Strain-Z (plane strain only)			
	13	Strain-XY (at Z1, if plane stress)			
	14-25	Items 2 through 13 repeated for fiber distance Z2 (plane stress only)			
CQUAD4 ² (95) Composite	2	Lamina Number	2	Lamina Number	
	3	Normal-1	3	Normal-1	RM
	4	Normal-2	4	Normal-2	RM

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
	5	Shear-12	5	Shear-12	RM
	6	Shear-1Z	6	Shear-1Z	RM
	7	Shear-2Z	7	Shear-2Z	RM
	8	Shear angle	8	Normal-1	IP
	9	Major principal	9	Normal-1	IP
	10	Minor principal	10	Shear-12	IP
	11	Maximum shear	11	Shear-1Z	IP
	12		12	Shear-2Z	IP
CQUAD4 (144) CORNER Output	1	EID	1	EID	
	2	CEN/	2	CEN/	
	3	4	3	4	
	4	Z1-Fiber distance	4	Z1-Fiber distance	
	5	Normal x at Z1	5	Normal x at Z1	RM
	6	Normal y at Z1	6	Normal x at Z1	IP
	7	Shear xy at Z1	7	Normal y at Z1	RM
	8	Shear angle at Z1	8	Normal y at Z1	IP
	9	Major principal at Z1	9	Shear xy at Z1	RM
	10	Minor principal at Z1	10	Shear xy at Z1	IP
	11	von Mises or maximum shear at Z1	11	Z2-Fiber distance	
	12	Z2-Fiber distance	12	Normal x at Z2	RM
	13	Normal x at Z2	13	Normal x at Z2	IP
	14	Normal y at Z2	14	Normal y at Z2	RM
	15	Shear xy at Z2	15	Normal y at Z2	IP
	16	Shear angle at Z2	16	Shear xy at Z2	RM
	17	Major principal at Z2	17	Shear xy at Z2	IP
	18	Minor principal at Z2	18	Grid 1	
	19	von Mises or maximum shear at Z2	19-32	Same as 4 through 17 for corner 1	
	20	Grid 1	33	Grid 2	
	21-36	Same as 4 through 19 for corner 1	34-47	Same as 4 through 17 for corner 2	
	37	Grid 2	48	Grid 3	

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
	38-53	Same as 4 through 19 for corner 2	49-62	Same as 4 through 17 for corner 3	
	54	Grid 3	63	Grid 4	
	55-70	Same as 4 through 19 for corner 3	64-77	Same as 4 through 17 for corner 4	
	71	Grid 4			
	72-87	Same as 4 through 19 for corner 4			
CQUAD8 (64)	5 ¹	Normal x at Z1	5 ¹	Normal x at Z1	RM
	6 ¹	Normal y at Z1	6 ¹	Normal x at Z1	IP
	7 ¹	Shear xy at Z1	7 ¹	Normal y at Z1	RM
	8	O Shear angle at Z1	8 ¹	Normal y at Z1	IP
	9	Major principal at Z1	9 ¹	Shear xy at Z1	RM
	10	Minor principal at Z1	10 ¹	Shear xy at Z1	IP
	11	von Mises or maximum shear at Z1	12 ¹	Normal x at Z2	RM
	13 ¹	Normal x at Z2	13 ¹	Normal x at Z2	IP
	14 ¹	Normal y at Z2	14 ¹	Normal y at Z2	RM
	15 ¹	Shear xy at Z2	15 ¹	Normal y at Z2	IP
	16	Q Shear angle at Z2	16 ¹	Shear xy at Z2	RM
	17	Major principal at Z2	17 ¹	Shear xy at Z2	IP
	18	Minor principal at Z2	20-32	Same as items 5 through 17 for corner 1	
	19	von Mises or maximum shear at Z2	35-47	Same as items 5 through 17 for corner 2	
	22-36	Same as items 5 through 19 for corner 1	50-62	Same as items 5 through 17 for corner 3	
CQUAD8 ² (96) Composite	39-53	Same as items 5 through 19 for corner 2	65-77	Same as items 5 through 17 for corner 4	
	56-70	Same as items 5 through 19 for corner 3			
	73-87	Same as items 5 through 19 for corner 4			
		Same as CQUAD4(95)		Same as CQUAD4(95)	

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
CQUADFD (201) Nonlinear Finite Deformation with 4 grid points	2	Grid/Gauss			
	3	Grid/Gauss ID (0=center)			
	4	Cauchy stress-X			
	5	Cauchy stress-Y			
	6	Cauchy stress-Z			
	7	Cauchy stress-XY			
	8	$\frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$		Not applicable	
	9	Volumetric strain J-1			
	10	Logarithmic strain-X			
	11	Logarithmic strain-Y			
	13	Logarithmic strain-XY			
	14-46	Items 3 through 13 repeated for 3 Gauss points			
CQUADFD (208) Nonlinear Finite Deformation with 8 or 9 grid points	2-13	Same as CQUADFD (201)			
	14-101	Items 3 through 13 repeated for 8 Gauss points		Not applicable	
CQUADR (82) Linear		Same as CQUAD4(144)		Same as CQUAD4(144)	
CQUADR (172) Nonlinear		Same as CQUAD4(90)		Not applicable	
CQUADR (232) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
CQUADX(18) Harmonic	1	(Elem ID)	1	(Elem ID)	
	2	Harmonic	2	Harmonic	
	3	CEN/grid	3	CEN/grid	
	4	Stress_xx	4	Stress_xx	RM
	5	Stress_yy	5	Stress_yy	RM
	6	Stress_tt	6	Stress_tt	RM
	7	Stress_xy	7	Stress_xy	RM
	8	Stress_yt	8	Stress_yt	RM
	9	Stress_tx	9	Stress_tx	RM
	10	VON MISES*	10	Stress_xx	IP
	11-18	1st corner	11	Stress_yy	IP
	19-26	2nd corner	12	Stress_tt	IP
	27-34	3rd corner	13	Stress_xy	IP
	35-42	4th corner	14	Stress_yt	IP
			15	Stress_tx	IP
			16-28	1st corner	
			29-41	2nd corner	
			42-54	3rd corner	
			55-67	4th corner	
CQUADXF(214) Nonlinear Finite Deformation with 4 grid points	2	Grid/Gauss			
	3	Gauss ID			
	4	Cauchy stress-X (radial)			
	5	Cauchy stress-Y (axial)			
	6	Cauchy stress-Z (circumferential)			
	7	Cauchy stress-XY			
	8	Pressure $p = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$		Not applicable	
	9	Volumetric strain J-1			
	10	Logarithmic strain-X (radial)			
	11	Logarithmic strain-Y (axial)			
	12	Logarithmic strain-Z (circumferential)			
	13	Logarithmic strain-XY			

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	14-46	Items 3 through 13 repeated for remaining 3 Gauss points			
CQUADXFD (215)	2-13	Same as QUADXFD (214)		Not applicable	
Nonlinear Finite Deformation with 8 or 9 grid points	14-101	Items 3 through 13 repeated for remaining 8 Gauss points			
CROD (1)		Same as CONROD(10)		Same as CONROD(10)	
Linear					
CROD (89)		Same as CONROD(92)		Not applicable	
Nonlinear					
CSHEAR (4)	2	Maximum shear	2	Maximum shear	RM
	3 ¹	Average shear	3	Maximum shear	IP
	4	Safety margin	4 ¹	Average shear	RM
			5 ¹	Average shear	IP
CSLOT3 (50)	2	Radial centroid	2	Radial centroid	RM
	3	Axial centroid	3	Axial centroid	RM
	4	Tangential edge 1	4	Tangential edge 1	RM
	5	Tangential edge 2	5	Tangential edge 2	RM
	6	Tangential edge 3	6	Tangential edge 3	RM
			7	Radial centroid	IP
			8	Axial centroid	IP
			9	Tangential edge 1	IP
			10	Tangential edge 2	IP
			11	Tangential edge 3	IP
CSLOT4 (51)	2	Radial centroid	2	Radial centroid	RM
	3	Axial centroid	3	Axial centroid	RM
	4	Tangential edge 1	4	Tangential edge 1	RM
	5	Tangential edge 2	5	Tangential edge 2	RM
	6	Tangential edge 3	6	Tangential edge 3	RM
	7	Tangential edge 4	7	Tangential edge 4	RM
			8	Radial centroid	IP
			9	Axial centroid	IP
			10	Tangential edge 1	IP
			11	Tangential edge 2	IP

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
			12	Tangential edge 3	IP
			13	Tangential edge 4	IP
CTETRA (39)	2	Stress coordinate system	2	Stress coordinate system	
Linear	3	Coordinate type (Character)	3	Coordinate type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID	5	External grid ID	
		(0=center)		(0=center)	
	6 ¹	Normal x	6 ¹	Normal x	RM
	7 ¹	Shear xy	7 ¹	Normal y	RM
	8	First principal	8 ¹	Normal z	RM
	9	First principal x cosine	9 ¹	Shear xy	RM
	10	Second principal x cosine	10 ¹	Shear yz	RM
	11	Third principal x cosine	11 ¹	Shear zx	RM
	12	Mean pressure	12 ¹	Normal x	IP
	13	von Mises or octahedral shear stress	13 ¹	Normal y	IP
	14 ¹	Normal y	14 ¹	Normal z	IP
	15 ¹	Shear yz	15 ¹	Shear xy	IP
	16	Second principal	16 ¹	Shear yz	IP
	17	First principal y cosine	17 ¹	Shear zx	IP
	18	Second principal y cosine	18-69	Items 5 through 17 repeated for four corners	
	19	Third principal y cosine			
	20 ¹	Normal z			
	21 ¹	Shear zx			
	22	Third principal			
	23	First principal z cosine			
	24	Second principal z cosine			
	25	Third principal z cosine			
	26-109	Items 5 through 25 repeated for four corners			

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CTETRA (85) Nonlinear	2	Stress coordinate system			
	3	Grid/Gauss			
	4	Number of active points			
	5	External grid ID (0=center)			
	6	Stress-X			
	7	Stress-Y			
	8	Stress-Z			
	9	Stress-XY			
	10	Stress-YZ			
	11	Stress-ZX			
	12	Equivalent stress			
	13	Effective plastic strain		Not applicable	
	14	Effective creep strain			
	15	Strain-X			
	16	Strain-Y			
	17	Strain-Z			
	18	Strain-XY			
	19	Strain-YZ			
	20	Strain-ZX			
	21-84	Items 5 through 20 Repeated for four corners			
CTETRAFD (205) Nonlinear Finite Deformation with 4 grid points	2-17	Same as CHEXA FD (202)		Not applicable	
CTETRAFD (210) Nonlinear Finite Deformation with 10 grid points	2-17	Same as CHEXA FD (202)			
	18-77	Items 3 through 17 repeated for 4 Gauss points		Not applicable	

Table 7-4 Element Stress-Strain Item Codes Part 4

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
CTRIA3 (74)		Same as CQUAD4(33)		Same as CQUAD4(33)	
CTRIA3 ² (97) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRIA3 (88) Nonlinear		Same as CQUAD4(90)		Not applicable	
CTRIA6 (75) Linear	5 ¹	Normal x at Z1	5 ¹	Normal x at Z1	RM
	6 ¹	Normal y at Z1	6 ¹	Normal x at Z1	IP
	7 ¹	Shear xy at Z1	7 ¹	Normal y at Z1	RM
	8	Q shear angle at Z1	8 ¹	Normal y at Z1	IP
	9	Major principal at Z1	9 ¹	Shear xy at Z1	RM
	10	Minor principal at Z1	10 ¹	Shear xy at Z1	IP
	11	von Mises or maximum shear at Z1	12 ¹	Normal x at Z2	RM
	13 ¹	Normal x at Z2	13 ¹	Normal x at Z2	IP
	14 ¹	Normal y at Z2	14 ¹	Normal y at Z2	RM
	15 ¹	Shear xy at Z2	15 ¹	Normal y at Z2	IP
	16	Q shear angle at Z2	16 ¹	Shear xy at Z2	RM
	17	Major principal at Z2	17 ¹	Shear xy at Z2	IP
	18	Minor principal at Z2	20-32	Same as items 5 through 17 for corner 1	
	19	von Mises or maximum shear at Z2	35-47	Same as items 5 through 17 for corner 2	
	22-36	Same as items 5 through 19 for corner 1	50-62	Same as items 5 through 17 for corner 3	
	39-53	Same as items 5 through 19 for corner 2			
	56-70	Same as items 5 through 19 for corner 3			
CTRIA6 ² (98)		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRIAFD (206) Nonlinear Deformation with 3 grid points	2-13	Same as CQUADFD(201)		Not applicable	

Table 7-4 Element Stress-Strain Item Codes Part 4 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
CTRIAFD (211) Nonlinear Finite Deformation with 6 grid points	2-13	Same as CQUADFD (201)			
	14-35	Items 3 through 12 repeated for 2 Gauss points		Not applicable	
CTRIAR (70) Linear		Same as CTRIA6(75)		Same as CTRIA6(75)	
CTRIAR (173) Nonlinear		Same as CQUAD4(90)		Not applicable	
CTRIAR (233) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRIAX(17) Harmonic	1	(Elem ID)	1	(Elem ID)	
	2	Harmonic	2	Harmonic	
	3	CEN/grid	3	CEN/grid	
	4	Stress_xx	4	Stress_xx	RM
	5	Stress_yy	5	Stress_yy	RM
	6	Stress_tt	6	Stress_tt	RM
	7	Stress_xy	7	Stress_xy	RM
	8	Stress_yt	8	Stress_yt	RM
	9	Stress_tx	9	Stress_tx	RM
	10	VON MISES*	10	Stress_xx	IP
	11-18	1st corner	11	Stress_yy	IP
	19-26	2nd corner	12	Stress_tt	IP
	27-34	3rd corner	13	Stress_xy	IP
			14	Stress_yt	IP
			15	Stress_tx	IP
			16-28	1st corner	
			29-41	2nd corner	
			42-54	3rd corner	
CTRIAX6 (53)	3 ¹	Radial	3 ¹	Radial	RM
	4 ¹	Azimuthal	4 ¹	Radial	IP
	5 ¹	Axial	5 ¹	Azimuthal	RM
	6 ¹	Shear stress	6 ¹	Azimuthal	IP

Table 7-4 Element Stress-Strain Item Codes Part 4 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	7	Maximum principal	7	Axial	RM
	8	Maximum shear	8	Axial	IP
	9	von Mises or octahedral	9	Shear	RM
			10	Shear	IP
	11-17	Same as items 3 through 9 for corner 1	12-19	Same as items 3 through 10 for corner 1	
	19-25	Same as Items 3 through 9 for corner 2	21-28	Same as items 3 through 10 for corner 2	
	27-33	Same as items 3 through 9 for corner 3	30-37	Same as items 3 through 10 for corner 3	
CTRIAXFD (212) Nonlinear Finite Deformation with 3 grid points	2-13	Same as CQUADXF D (214)		Not applicable	
CTRIAXFD (213) Nonlinear Finite Deformation with 6 grid points	2-13	Same as CQUADXF D (214)			
CTUBE (3) Linear	14-35	Items 3 through 13 repeated for 2 Gauss points		Not applicable	
CTUBE (87) Nonlinear		Same as CONROD(92)		Not applicable	

Table 7-4 Element Stress-Strain Item Codes Part 4 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CWELDP (118) if PARTPAT or ELPAT	2	Axial	2	Axial	RM
	3	Maximum End A	3	Maximum End A	RM
	4	Minimum End A	4	Minimum End A	RM
	5	Maximum End B	5	Maximum End B	RM
	6	Minimum End B	6	Minimum End B	RM
	7	Maximum Shear	7	Maximum Shear	RM
	8	Bearing Stress (Not applicable to Strain)	8	Bearing Stress (Not applicable to Strain)	RM
			9	Axial	IP
			10	Maximum End A	IP
			11	Minimum End A	IP
			12	Maximum End B	IP
			13	Minimum End B	IP
			14	Maximum Shear	IP
			15	Bearing Stress (Not applicable to Strain)	IP
CWELDC (117) if MSET=OFF and oldweld=yes	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	VU grid ID for corner 1			
	4	Normal x	4	Normal x	RM
	5	Normal y	5	Normal y	RM
	6	Normal z	6	Normal z	RM
	7	Shear xy	7	Shear xy	RM
	8	Shear yz	8	Shear yz	RM
	9	Shear zx	9	Shear zx	RM
	10	First principal	10	Normal x	IP
	11	Second principal	11	Normal y	IP
	12	Third principal	12	Normal z	IP
	13	Mean pressure	13	Shear xy	IP
	14	von Mises/ Octahedral	14	Shear yz	IP
CWELD (200) if MSET=ON or olddweld=no	15-26	Repeat items 3-14 for corner 2	15	Shear zx	IP
	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	VU grid ID for corner 1			
	4	Normal x	4	Normal x	RM
	5	Normal y	5	Normal y	RM
	6	Normal z	6	Normal z	RM
	7	Shear xy	7	Shear xy	RM
	8	Shear yz	8	Shear yz	RM
	9	Shear zx	9	Shear zx	RM
	10	First principal	10	Normal x	IP
	11	Second principal	11	Normal y	IP
	12	Third principal	12	Normal z	IP
	13	Mean pressure	13	Shear xy	IP
	14	von Mises/ Octahedral	14	Shear yz	IP
	15-26	Repeat items 3-14 for corner 2	15	Shear zx	IP

Table 7-4 Element Stress-Strain Item Codes Part 4 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
	27-38	Repeat items 3-14 for corner 3	16-28	Repeat items 3-15 for corner 2	
	39-50	Repeat items 3-14 for corner 4	29-41	Repeat items 3-15 for corner 3	
	51-62	Repeat items 3-14 for corner 5 (VUPENTA, VUHEXA)	42-54	Repeat items 3-15 for corner 4	
	63-74	Repeat items 3-14 for corner 6 (VUPENTA, VUHEXA)	55-67	Repeat items 3-15 for corner 5 (VUPENTA, VUHEXA)	
	75-86	Repeat items 3-14 for corner 7 (VUHEXA)	68-80	Repeat items 3-15 for corner 6 (VUPENTA, VUHEXA)	
	87-98	Repeat items 3-14 for corner 8 (VUHEXA)	81-93	Repeat items 3-15 for corner 7 (VUHEXA)	
			94-106	Repeat items 3-15 for corner 8 (VUHEXA)	
VUHEXA (145) VUPENTA (146) VUTETRA (147) for HEXAp, PENTAp, TETRAPt if SDRPOPT=OFp (no principals)	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	VU grid ID for corner 1			
	4	Normal x	4	Normal x	RM
	5	Normal y	5	Normal y	RM
	6	Normal z	6	Normal z	RM
	7	Shear xy	7	Shear xy	RM
	8	Shear yz	8	Shear yz	RM
	9	Shear zx	9	Shear zx	RM
	10-16	Repeat items 3-9 for corner 2	10	Normal x	IP
	17-23	Repeat items 3-9 for corner 3	11	Normal y	IP
	24-30	Repeat items 3-9 for corner 4	12	Normal z	IP
	31-37	Repeat items 3-9 for corner 5 (VUPENTA, VUHEXA)	13	Shear xy	IP
	38-44	Repeat items 3-9 for corner 6 (VUPENTA, VUHEXA)	14	Shear yz	IP
	45-51	Repeat items 3-9 for corner 7 (VUHEXA)	15	Shear zx	IP
	52-58	Repeat items 3-9 for corner 8 (VUHEXA)	16-28	Repeat items 3-15 for corner 2	
			29-41	Repeat items 3-15 for corner 3	

Table 7-4 Element Stress-Strain Item Codes Part 4 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
			42-54	Repeat items 3-15 for corner 4	
			55-67	Repeat items 3-15 for corner 5 (VUPENTA, VUHEXA)	
			68-80	Repeat items 3-15 for corner 6 (VUPENTA, VUHEXA)	
			81-93	Repeat items 3-15 for corner 7 (VUHEXA)	
			94-106	Repeat items 3-15 for corner 8 (VUHEXA)	

Table 7-5 Element Stress-Strain Item Codes Part 5

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
VUQUAD (189) VUTRIA (190) for QUADp and TRIAp if STRAIN= FIBER; 4th char. of ICORD= X Y, Z (local coordinate system); and SDRPORT =SDRP (with principals)	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat/curved etc.			
	5	THETA material angle			
	6	ITYPE strcur=0, fiber=1			
	7	VU grid ID for corner 1			
	8	Z1 fiber distance			
	9	Z2 fiber distance			
	10	Normal x at Z1	10	Normal x at Z1	RM
	11	Normal y at Z1	11	Normal y at Z1	RM
	12	Shear xy at Z1	12	Shear xy at Z1	RM
	13	Shear Angle at Z1	13	0.0	RM
	14	Major principal at Z1	14	0.0	RM
	15	Minor principal at Z1	15	0.0	RM
	16	vonMises/Max.Shear at Z1	16	Normal x at Z2	RM
	17	Normal x at Z2	17	Normal y at Z2	RM
	18	Normal y at Z2	18	Shear xy at Z2	RM

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	19	Shear xy at Z2	19	0.0	RM
	20	Shear Angle at Z2	20	0.0	RM
	21	Major principal at Z2	21	0.0	RM
	22	Minor principal at Z2	22	Normal x at Z1	IP
	23	vonMises/Max.Shear at Z2	23	Normal y at Z1	IP
			24	Shear xy at Z1	IP
			25	0.0	IP
			26	0.0	IP
			27	0.0	IP
			28	Normal x at Z2	IP
			29	Normal y at Z2	IP
			30	Shear xy at Z2	IP
			31	0.0	IP
			32	0.0	IP
			33	0.0	IP
	24-40	Repeat items 7-23 for corner 2	34-60	Repeat items 7-33 for corner 2	
	41-57	Repeat items 7-23 for corner 3	61-87	Repeat items 7-33 for corner 3	
	58-74	Repeat items 7-23 for corner 4 (VUQUAD)	88-104	Repeat items 7-33 for corner 4 (VUQUAD)	
VUQUAD (189) VUTRIA (190) for QUADp and TRIAp if STRAIN= FIBER; if 4th char. of ICORD=X Y, Z (local coordinate system); and SDRPOPT =OPP (no principals)	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat/curved etc.			
	5	THETA material angle			
	6	ITYPE strcur=0, fiber=1			
	7	VU grid ID for corner 1			
	8	Z1 fiber distance			
	9	Z2 fiber distance			
	10	Normal x at Z1	10	Normal x at Z1	RM
	11	Normal y at Z1	11	Normal y at Z1	RM
	12	Shear xy at Z1	12	Shear xy at Z1	RM

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
	13	0.0	13	0.0	RM
	14	0.0	14	0.0	RM
	15	0.0	15	0.0	RM
	16	0.0	16	Normal x at Z2	RM
	17	Normal x at Z2	17	Normal y at Z2	RM
	18	Normal y at Z2	18	Shear xy at Z2	RM
	19	Shear xy at Z2	19	0.0	RM
	20	0.0	20	0.0	RM
	21	0.0	21	0.0	RM
	22	0.0	22	Normal x at Z1	IP
	23	0.0	23	Normal y at Z1	IP
			24	Shear xy at Z1	IP
			25	0.0	IP
			26	0.0	IP
			27	0.0	IP
			28	Normal x at Z2	IP
			29	Normal y at Z2	IP
			30	Shear xy at Z2	IP
			31	0.0	IP
			32	0.0	IP
			33	0.0	IP
24-40	Repeat items 7-23 for corner 2		34-60	Repeat items 7-33 for corner 2	
41-57	Repeat items 7-23 for corner 3		61-87	Repeat items 7-33 for corner 3	
58-74	Repeat items 7-23 for corner 4 (VUQUAD)		88-104	Repeat items 7-33 for corner 4 (VUQUAD)	

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
VUQUAD (189) VUTRIA (190) for QUADp and TRIAp if STRAIN= FIBER and 4th char. of ICORD=F (fixed coordinate system)	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat/curved etc.			
	5	THETA material angle			
	6	ITYPE strcur=0, fiber=1			
	7	VU grid ID for corner 1			
	8	Z1 fiber distance			
	9	Z2 fiber distance			
	10	Normal x at Z1	10	Normal x at Z1	RM
	11	Normal y at Z1	11	Normal y at Z1	RM
	12	Shear xy at Z1	12	Shear xy at Z1	RM
	13	Shear yz at Z1	13	Shear yz at Z1	RM
	14	Shear zx at Z1	14	Shear zx at Z1	RM
	15	Normal z at Z1	15	Normal z at Z1	RM
	16	Normal x at Z2	16	Normal x at Z2	RM
	17	Normal y at Z2	17	Normal y at Z2	RM
	18	Shear xy at Z2	18	Shear xy at Z2	RM
	19	Shear yz at Z2	19	Shear yz at Z2	RM
	20	Shear zx at Z2	20	Shear zx at Z2	RM
	21	Normal z at Z2	21	Normal z at Z2	RM
	22	0.0	22	Normal x at Z1	IP
	23	0.0	23	Normal y at Z1	IP
			24	Shear xy at Z1	IP
			25	Shear yz at Z1	IP
			26	Shear zx at Z1	IP
			27	Normal z at Z1	IP
			28	Normal x at Z2	IP
			29	Normal y at Z2	IP
			30	Shear xy at Z2	IP
			31	Shear yz at Z2	IP
			32	Shear zx at Z2	IP

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
			33	Normal z at Z2	IP
	24-40	Repeat items 7-23 for corner 2	34-60	Repeat items 7-33 for corner 2	
	41-57	Repeat items 7-23 for corner 3	61-87	Repeat items 7-33 for corner 3	
	58-74	Repeat items 7-23 for corner 4 (VUQUAD)	88-104	Repeat items 7-33 for corner 4 (VUQUAD)	
VUQUAD (189) VUTRIA (190) for QUADp and TRIAp if STRAIN=STRCUR and 4th char. of ICORD=X Y, Z (local coordinate system)	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat/curved etc.			
	5	THETA material angle			
	6	ITYPE strcur=0, fiber=1			
	7	VU grid ID for corner 1			
	8	0.0			
	9	0.0			
	10	Membrane Strain x	10	Membrane Strain x	RM
	11	Membrane Strain y	11	Membrane Strain y	RM
	12	Membrane Strain xy	12	Membrane Strain xy	RM
	13	0.0	13	0.0	RM
	14	0.0	14	0.0	RM
	15	0.0	15	0.0	RM
	16	Bending Curvature x	16	Bending Curvature x	RM
	17	Bending Curvature y	17	Bending Curvature y	RM
	18	Bending Curvature xy	18	Bending Curvature xy	RM
	19	Shear yz	19	Shear yz	RM
	20	Shear zx	20	Shear zx	RM
	21	0.0	21	0.0	RM
	22	0.0	22	Membrane Strain x	IP
	23	0.0	23	Membrane Strain y	IP
			24	Membrane Strain xy	IP
			25	0.0	IP
			26	0.0	IP

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
			27	0.0	IP
			28	Bending Curvature x	IP
			29	Bending Curvature y	IP
			30	Bending Curvature xy	IP
			31	Shear yz	IP
			32	Shear zx	IP
			33	0.0	IP
	24-40	Repeat items 7-23 for corner 2	34-60	Repeat items 7-33 for corner 2	
	41-57	Repeat items 7-23 for corner 3	61-87	Repeat items 7-33 for corner 3	
	58-74	Repeat items 7-23 for corner 4 (VUQUAD)	88-104	Repeat items 7-33 for corner 4 (VUQUAD)	
VUQUAD (189) VUTRIA (190) for QUADp and TRIAp if STRAIN= STRCUR and 4th char. of ICORD=F (fixed coordinate system)	1	VU element ID *10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat/curved etc.			
	5	THETA material angle			
	6	ITYPE strcur=0, fiber=1			
	7	VU grid ID for corner 1			
	8	0.0			
	9	0.0			
	10	Membrane Strain x	10	Membrane Strain x	RM
	11	Membrane Strain y	11	Membrane Strain y	RM
	12	Membrane Strain xy	12	Membrane Strain xy	RM
	13	Membrane Strain yz	13	Membrane Strain yz	RM
	14	Membrane Strain zx	14	Membrane Strain zx	RM
	15	Membrane Strain z	15	Membrane Strain z	RM
	16	Bending Curvature x	16	Bending Curvature x	RM
	17	Bending Curvature y	17	Bending Curvature y	RM
	18	Bending Curvature xy	18	Bending Curvature xy	RM
	19	Bending Curvature yz	19	Bending Curvature yz	RM
	20	Bending Curvature zx	20	Bending Curvature zx	RM

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		Real/Mag. or Imag./Phase
	Item Code	Item	Item Code	Item	
	21	Bending Curvature z	21	Bending Curvature z	RM
	22	0.0	22	Membrane Strain x	IP
	23	0.0	23	Membrane Strain y	IP
			24	Membrane Strain xy	IP
			25	Membrane Strain yz	IP
			26	Membrane Strain zx	IP
			27	Membrane Strain z	IP
			28	Bending Curvature x	IP
			29	Bending Curvature y	IP
			30	Bending Curvature xy	IP
			31	Bending Curvature yz	IP
			32	Bending Curvature zx	IP
			33	Bending Curvature z	IP
	24-40	Repeat items 7-23 for corner 2	34-60	Repeat items 7-33 for corner 2	
	41-57	Repeat items 7-23 for corner 3	61-87	Repeat items 7-33 for corner 3	
	58-74	Repeat items 7-23 for corner 4 (VUQUAD)	88-104	Repeat items 7-33 for corner 4 (VUQUAD)	
VUBEAM (191) for BEAMp	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD (not used)			
	5	VU grid ID for end 1			
	6	x/L position			
	7	Y-coordinate of output point C			
	8	Z-coordinate of output point C			
	9	W-coordinate of output point C			
	10	Normal x at C	10	Normal x at C	RM
	11	Shear xy at C	11	Shear xy at C	RM
	12	Shear zx at C	12	Shear zx at C	RM

Table 7-5 Element Stress-Strain Item Codes Part 5 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
			13	Normal x at C	IP
			14	Shear xy at C	IP
			15	Shear zx at C	IP
	13-18	Repeat items 7-12 for output point D	16-24	Repeat items 7-15 for output point D	
	19-24	Repeat items 7-12 for output point E	25-33	Repeat items 7-15 for output point E	
	25-30	Repeat items 7-12 for output point F	34-42	Repeat items 7-15 for output point F	
	31	Max longitudinal			
	32	Min longitudinal			
	33-60	Repeat items 5-32 for end 2	43-80	Repeat items 5-42 for end 2	

Element Force Item Codes

All items are element forces (or moments) unless otherwise indicated.

Table 7-6 Element Force Item Codes Part 1

Element Name Code	Real Element Forces		Complex Element Forces		Real/Mag. or Imag/Phase
	Item Code	Item	Item Code	Item	
CBAR (34) Linear	2 ¹	Bending End A plane 1	2 ¹	Bending End A plane 1	RM
	3 ¹	Bending End A plane 2	3 ¹	Bending End A plane 2	RM
	4 ¹	Bending End B plane 1	4 ¹	Bending End B plane 1	RM
	5 ¹	Bending End B plane 2	5 ¹	Bending End B plane 2	RM
	6 ¹	Shear plane 1	6 ¹	Shear plane 1	RM
	7 ¹	Shear plane 2	7 ¹	Shear plane 2	RM
	8 ¹	Axial force	8 ¹	Axial force	RM
	9 ¹	Torque	9 ¹	Torque	RM
			10 ¹	Bending End A plane 1	IP
			11 ¹	Bending End A plane 2	IP
			12 ¹	Bending End B plane 1	IP
			13 ¹	Bending End B plane 2	IP
			14 ¹	Shear plane 1	IP
			15 ¹	Shear plane 2	IP
			16 ¹	Axial force	IP
			17 ¹	Torque	IP
CBAR (100) Intermediate Stations	2	Station Distance/Length	2	Station Distance/Length	
	3	Bending Moment Plane 1	3	Bending Moment Plane 1	RM
	4	Bending Moment Plane 2	4	Bending Moment Plane 2	RM
	5	Shear Force Plane 1	5	Shear Force Plane 1	RM
	6	Shear Force Plane 2	6	Shear Force Plane 2	RM
	7	Axial	7	Axial	RM
	8	Torque	8	Torque	RM

Table 7-6 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
		Item codes are given for end A. Addition of the quantity (K-1) * 8 to the item code points to the same information for other stations, where K is the station number. K=8 for end B and 2 through 7 for intermediate stations.	9	Bending Moment Plane 1	IP
			10	Bending Moment Plane 2	IP
			11	Shear Force Plane 1	IP
			12	Shear Force Plane 2	IP
			13	Axial	IP
			14	Torque	IP
				(Item codes above are given for End A. For codes 2 through 14 at intermediate stations add (K-1) * 13 where K is the station number, and for codes at End B, K+number of stations plus 1.)	
CBEAM (2)	2	External grid point ID	2	External grid point ID	
Linear	3	Station distance/length	3	Station distance/length	
	4 ¹	Bending moment plane 1	4 ¹	Bending moment plane 1	RM
	5 ¹	Bending moment plane 2	5 ¹	Bending moment plane 2	RM
	6 ¹	Web shear plane 1	6 ¹	Web shear plane 1	RM
	7 ¹	Web shear plane 2	7 ¹	Web shear plane 2	RM
	8 ¹	Axial force	8 ¹	Axial force	RM
	9 ¹	Total torque	9 ¹	Total torque	RM
	10 ¹	Warping torque	10 ¹	Warping torque	RM
		(Item codes are given for end A. Addition of the quantity (K-1) 9 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.)	11 ¹	Bending moment plane 1	IP
			12 ¹	Bending moment plane 2	IP
			13 ¹	Web shear plane 1	IP
			14 ¹	Web shear plane 2	IP
			15 ¹	Axial force	IP
			16 ¹	Total torque	IP
			17 ¹	Warping torque	IP
				(Item codes are given for end A. Addition of the quantity (K-1) 16 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.)	
CBEAM3 (184)	2	External grid point ID	2	External grid point ID	
Linear	3	Bending moment in y-direction	3	Bending moment in y-direction	RM

Table 7-6 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		Real/Mag. or Imag/Phase
	Item Code	Item	Item Code	Item	
	4	Bending moment in z-direction	4	Bending moment in z-direction	RM
	5	Shear force in y-direction	5	Shear force in y-direction	RM
	6	Shear force in z-direction	6	Shear force in z-direction	RM
	7	Axial force	7	Axial force	RM
	8	Total torque	8	Total torque	RM
	9	Bi-shear force	9	Bi-shear force	RM
	10	Bi-moment (Item codes are given for end A. They are repeated for end B and mid-node C, respectively)	10	Bi-moment	RM
			11	Bending moment in y-direction	IP
			12	Bending moment in z-direction	IP
			13	Shear force in y-direction	IP
			14	Shear force in z-direction	IP
			15	Axial force	IP
			16	Total torque	IP
			17	Bi-shear force	IP
			18	Bi-moment (Item codes are given for end A. They are repeated for end B and mid-node C, respectively)	IP
CBEND (69)	2	External grid point ID	2	External grid point ID	
	3 ¹	Bending moment plane 1	3 ¹	Bending moment plane 1	RM
	4 ¹	Bending moment plane 2	4 ¹	Bending moment plane 2	RM
	5 ¹	Shear plane 1	5 ¹	Shear plane 1	RM
	6 ¹	Shear plane 2	6 ¹	Shear plane 2	RM
	7 ¹	Axial force	7 ¹	Axial force	RM
	8 ¹	Torque (Item codes are given for end A. Item codes 9 through 15 point to the same information for end B.)	8 ¹	Torque	RM
			9 ¹	Bending moment plane 1	IP
			10 ¹	Bending moment plane 2	IP
			11 ¹	Shear plane 1	IP
			12 ¹	Shear plane 2	IP
			13 ¹	Axial force	IP

Table 7-6 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
			14 ¹	Torque (Item codes are given for end A. Item codes 15 through 27 point to the same information for end B.)	IP
CBUSH (102)	2 ¹	Force-x	2 ¹	Force-x	RM
	3 ¹	Force-y	3 ¹	Force-y	RM
	4 ¹	Force-z	4 ¹	Force-z	RM
	5 ¹	Moment-x	5 ¹	Moment-x	RM
	6 ¹	Moment-y	6 ¹	Moment-y	RM
	7 ¹	Moment-z	7 ¹	Moment-z	RM
			8 ¹	Force-x	IP
			9 ¹	Force-y	IP
			10 ¹	Force-z	IP
			11 ¹	Moment-x	IP
			12 ¹	Moment-y	IP
			13 ¹	Moment-z	IP
CCONEAX (35)	2	Harmonic or point angle		Not applicable	
	3	Bending moment v			
	4	Bending moment u			
	5	Twist moment			
	6	Shear v			
	7	Shear u			
CDAMP1 (20)		Same as CELAS1		Same as CELAS1	
CDAMP2 (21)		Same as CELAS1		Same as CELAS1	
CDAMP3 (22)		Same as CELAS1		Same as CELAS1	
CDAMP4 (23)		Same as CELAS1		Same as CELAS1	
CDUM3 thru CDUM9 (55 - 61)	2 ¹	F1	2 ¹	F1	RM
	3 ¹	F2	3 ¹	F2	RM
	4 ¹	F3	4 ¹	F3	RM
	5 ¹	F4	5 ¹	F4	RM
	6 ¹	F5	6 ¹	F5	RM
	7 ¹	F6	7 ¹	F6	RM
	8 ¹	F7	8 ¹	F7	RM

Table 7-6 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		Real/Mag. or Imag/Phase
	Item Code	Item	Item Code	Item	
	9 ¹	F8	9 ¹	F8	RM
	10 ¹	F9	10 ¹	F9	RM
			11 ¹	F1	IP
			12 ¹	F2	IP
			13 ¹	F3	IP
			14 ¹	F4	IP
			15 ¹	F5	IP
			16 ¹	F6	IP
			17 ¹	F7	IP
			18 ¹	F8	IP
			19 ¹	F9	IP
CELAS1 (11)	2 ¹	Force	2 ¹	Force	RM
	3	Force	3	Force	IP
CELAS2 (12)		Same as CELAS1		Same as CELAS1	
CELAS3 (13)		Same as CELAS1		Same as CELAS1	
CELAS4 (14)		Same as CELAS1		Same as CELAS1	
CGAP (38)	2	Normal x		Not applicable	
	3	Shear y			
	4	Shear z			
	5	Axial u			
	6	Shear v			
	7	Shear w			
	8	Slip v			
	9	Slip w			
CONROD (10)	2 ¹	Axial force	2 ¹	Axial force	RM
	3 ¹	Torque	3 ¹	Axial force	IP
			4 ¹	Torque	RM
			5 ¹	Torque	IP
CQUAD4 (33)	2 ¹	Membrane force x	2 ¹	Membrane force x	RM
Linear	3 ¹	Membrane force y	3 ¹	Membrane force y	RM
	4 ¹	Membrane force xy	4 ¹	Membrane force xy	RM
	5 ¹	Bending moment x	5 ¹	Bending moment x	RM

Table 7-6 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		Real/Mag. or Imag/Phase
	Item Code	Item	Item Code	Item	
	6 ¹	Bending moment y	6 ¹	Bending moment y	RM
	7 ¹	Bending moment xy	7 ¹	Bending moment xy	RM
	8 ¹	Shear x	8 ¹	Shear x	RM
	9 ¹	Shear y	9 ¹	Shear y	RM
			10 ¹	Membrane force x	IP
			11 ¹	Membrane force y	IP
			12 ¹	Membrane force xy	IP
			13 ¹	Bending moment x	IP
			14 ¹	Bending moment y	IP
			15 ¹	Bending moment xy	IP
			16 ¹	Shear x	IP
			17 ¹	Shear y	IP
CQUAD4 (95)	2-3	Theory or blank		Not applicable	
Composite	4	Lamina number			
	5	FP (failure index) /SP (strength ratio) for direct stresses			
	6	Failure mode for			
		Maximum strain theory			
	7	FB (failure index) /SB (strength ratio) or -1 for interlaminar shear-stress			
	8	MAX of FP, FB or -1 or MIN of SP, SB or -1			
	9	Failure flag			
CQUAD4 (144)	1	EID	1	EID	
Corner Output	2	CEN/	2	CEN/	
	3	4	3	4	
	4	Membrane x	4	Membrane x	RM
	5	Membrane y	5	Membrane y	RM
	6	Membrane xy	6	Membrane xy	RM
	7	Bending x	7	Bending x	RM
	8	Bending y	8	Bending y	RM
	9	Bending xy	9	Bending xy	RM

Table 7-6 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		Real/Mag. or Imag/Phase
	Item Code	Item	Item Code	Item	
	10	Shear x	10	Shear x	RM
	11	Shear y	11	Shear y	RM
	12	Grid 1	12	Membrane x	IP
	13-20	Same as 4 through 11 for corner 1	13	Membrane y	IP
	21	Grid 2	14	Membrane xy	IP
	22-29	Same as 4 through 11 for corner 2	15	Bending x	IP
	30	Grid 3	16	Bending y	IP
	31-38	Same as 4 through 11 for corner 3	17	Bending xy	IP
	39	Grid 4	18	Shear x	IP
	40-47	Same as 4 through 11 for corner 4	19	Shear y	IP
			20	Grid 1	
			21-36	Same as 4 through 19 for corner 1	
			37	Grid 2	
			38-53	Same as 4 through 19 for corner 2	
			54	Grid 3	
			55-70	Same as 4 through 19 for corner 3	
			71	Grid 4	
			71-87	Same as 4 through 19 for corner 4	
CQUAD8 (64)	1	EID	1	EID	
Linear	2	CEN/	2	CEN/	
	3	4	3	4	
	4 ¹	Membrane force x	4 ¹	Membrane force x	RM
	5 ¹	Membrane force y	5 ¹	Membrane force y	RM
	6 ¹	Membrane force xy	6 ¹	Membrane force xy	RM
	7 ¹	Bending moment x	7 ¹	Bending moment x	RM
	8 ¹	Bending moment y	8 ¹	Bending moment y	RM
	9 ¹	Bending moment xy	9 ¹	Bending moment xy	RM
	10 ¹	Shear x	10 ¹	Shear x	RM

Table 7-6 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		Real/Mag. or Imag/Phase
	Item Code	Item	Item Code	Item	
	11 ¹	Shear y	11 ¹	Shear y	RM
	12	Grid1	12 ¹	Membrane force x	IP
	13-20	Same as items 4 through 11 for corner 1	13 ¹	Membrane force y	IP
	21	Grid2	14 ¹	Membrane force xy	IP
	22-29	Same as items 4 through 11 for corner 2	15 ¹	Bending moment x	IP
	30	Grid3	16 ¹	Bending moment y	IP
	31-38	Same as items 4 through 11 for corner 3	17 ¹	Bending moment xy	IP
	39	Grid4	18 ¹	Shear x	IP
	40-47	Same as items 4 through 11 for corner 4	19 ¹	Shear y	IP
			20	Grid1	
			21-36	Same as items 4 through 19 for corner 1	
37			37	Grid2	
			38-53	Same as items 4 through 19 for corner 2	
			54	Grid3	
			55-70	Same as items 4 through 19 for corner 3	
			71	Grid4	
			72-87	Same as items 4 through 19 for corner 4	
CQUAD8 ² (96) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CQUADR (82)		Same as CQUAD8(64)		Same as CQUAD8(64)	
CQUADR (235) Center		Same as CQUAD4(33)		Same as CQUAD4(33)	

Table 7-7 Element Force Item Codes Part 2

Element Name Code	Real Element Forces		Complex Element Forces		Real/Mag. or Imag/Phase
	Item Code	Item	Item Code	Item	
CROD (1)		Same as CONROD(10)		Same as CONROD(10)	
CSHEAR (4)	2 ¹	Force 4 to 1	2 ¹	Force 4 to 1	RM
	3 ¹	Force 2 to 1	3 ¹	Force 2 to 1	RM
	4 ¹	Force 1 to 2	4 ¹	Force 1 to 2	RM
	5 ¹	Force 3 to 2	5 ¹	Force 3 to 2	RM
	6 ¹	Force 2 to 3	6 ¹	Force 2 to 3	RM
	7 ¹	Force 4 to 3	7 ¹	Force 4 to 3	RM
	8 ¹	Force 3 to 4	8 ¹	Force 3 to 4	RM
	9 ¹	Force 1 to 4	9 ¹	Force 1 to 4	RM
	10 ¹	Kick force on 1	10 ¹	Force 4 to 1	IP
	11 ¹	Shear 12	11 ¹	Force 2 to 1	IP
	12 ¹	Kick force on 2	12 ¹	Force 1 to 2	IP
	13 ¹	Shear 23	13 ¹	Force 3 to 2	IP
	14 ¹	Kick force on 3	14 ¹	Force 2 to 3	IP
	15 ¹	Shear 34	15 ¹	Force 4 to 3	IP
	16 ¹	Kick force on 4	16 ¹	Force 3 to 4	IP
	17 ¹	Shear 41	17 ¹	Force 1 to 4	IP
			18 ¹	Kick force on 1	RM
			19 ¹	Shear 12	RM
			20 ¹	Kick force on 2	RM
			21 ¹	Shear 23	RM
			22 ¹	Kick force on 3	RM
			23 ¹	Shear 34	RM
			24 ¹	Kick force on 4	RM
			25 ¹	Shear 41	RM
			26 ¹	Kick force on 1	IP
			27 ¹	Shear 12	IP
			28 ¹	Kick force on 2	IP
			29 ¹	Shear 23	IP
			30 ¹	Kick force on 3	IP
			31 ¹	Shear 34	IP

Table 7-7 Element Force Item Codes Part 2 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		Real/Mag. or Imag/Phase
	Item Code	Item	Item Code	Item	
			32 ¹	Kick force on 4	IP
			33 ¹	Shear 41	IP
CTRIA3 (74) Linear		Same as CQUAD4(33)		Same as CQUAD4(33)	
CTRIA3 ² (97) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRIA6 (75) Linear	4 ¹	Membrane force x	4 ¹	Membrane force x	RM
	5 ¹	Membrane force y	5 ¹	Membrane force y	RM
	6 ¹	Membrane force xy	6 ¹	Membrane force xy	RM
	7 ¹	Bending moment x	7 ¹	Bending moment x	RM
	8 ¹	Bending moment y	8 ¹	Bending moment y	RM
	9 ¹	Bending moment xy	9 ¹	Bending moment xy	RM
	10 ¹	Shear x	10 ¹	Shear x	RM
	11 ¹	Shear y	11 ¹	Shear y	RM
	13-20	Same as items 4 through 11 for corner 1	12 ¹	Membrane force x	IP
	22-29	Same as items 4 through 11 for corner 2	13 ¹	Membrane force y	IP
	31-38	Same as items 4 through 11 for corner 3	14 ¹	Membrane force xy	IP
			15 ¹	Bending moment x	IP
			16 ¹	Bending moment y	IP
			17 ¹	Bending moment xy	IP
			18 ¹	Shear x	IP
			19 ¹	Shear y	IP
			21-36	Same as items 4 through 19 for corner 1	
			38-53	Same as items 4 through 19 for corner 2	
			55-70	Same as items 4 through 19 for corner 3	
CTRIA6 ² (98) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRIAR (70)		Same as CTRIA6(75)		Same as CTRIA6(75)	

Table 7-7 Element Force Item Codes Part 2 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		Real/Mag. or Imag/Phase
	Item Code	Item	Item Code	Item	
CTRIAR (236) Center	Same as CQUAD4(33)		Same as CQUAD4(33)		
CTUBE (3)	Same as CONROD(10)		Same as CONROD(10)		
CVISC (24)	Not applicable		Same as CONROD(10)		
CWELDP (118) if PARTPAT or ELPAT	2	mz bending end A plane 1	2	mz bending end A plane 1	RM
	3	my bending end A plane 2	3	my bending end A plane 2	RM
	4	mz bending end B plane 1	4	mz bending end B plane 1	RM
	5	my bending end B plane 2	5	my bending end B plane 2	RM
	6	fy shear force plane 1	6	fy shear force plane 1	RM
	7	fz shear force plane 2	7	fz shear force plane 2	RM
	8	fx axial force	8	fx axial force	RM
CWELDC (117) if MSET = OFF	9	mx torque	9	mx torque	RM
			10	mz bending end A plane 1	IP
			11	my bending end A plane 2	IP
CWELD (200) if MSET = ON			12	mz bending end B plane 1	IP
			13	my bending end B plane 2	IP
			14	fy shear force plane 1	IP
			15	fz shear force plane 2	IP
			16	fx axial force	IP
			17	mx torque	IP
VUQUAD (189) VUTRIA (190) for QUADp and TRIAp if 4th char. of ICORD=X Y, Z (local coordinate system)	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat/curved etc.			
	5	THETA material angle			
	6	0.0			
	7	VU grid ID for corner 1			
	8	Membrane Force x	8	Membrane Force x	RM
	9	Membrane Force y	9	Membrane Force y	RM
	10	Membrane Force xy	10	Membrane Force xy	RM
	11	0.0	11	0.0	RM

Table 7-7 Element Force Item Codes Part 2 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		Real/Mag. or Imag/Phase
	Item Code	Item	Item Code	Item	
	12	0.0	12	0.0	RM
	13	0.0	13	0.0	RM
	14	Bending Moment x	14	Bending Moment x	RM
	15	Bending Moment y	15	Bending Moment y	RM
	16	Bending Moment xy	16	Bending Moment xy	RM
	17	Shear zx	17	Shear zx	RM
	18	Shear yz	18	Shear yz	RM
	19	0.0	19	0.0	RM
			20	Membrane Force x	IP
			21	Membrane Force y	IP
			22	Membrane Force xy	IP
			23	0.0	IP
			24	0.0	IP
			25	0.0	IP
			26	Bending Moment x	IP
			27	Bending Moment y	IP
			28	Bending Moment xy	IP
			29	Shear zx	IP
			30	Shear yz	IP
			31	0.0	IP
	20-32	Repeat items 7-19 for corner 2	32-56	Repeat items 7-31 for corner 2	
	33-45	Repeat items 7-19 for corner 3	57-81	Repeat items 7-31 for corner 3	
	46-58	Repeat items 7-19 for corner 4 (VUQUAD)	82-106	Repeat items 7-31 for corner 4 (VUQUAD)	
VUQUAD (189) VUTRIA (190) for QUADp and TRIAP if 4th char. of ICORD=F (fixed coordinate system)	1	VU element ID *10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			
	4	ICORD flat/curved etc.			
	5	THETA material angle			
	6	0.0			
	7	VU grid ID for corner 1			

Table 7-7 Element Force Item Codes Part 2 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		Real/Mag. or Imag/Phase
	Item Code	Item	Item Code	Item	
	8	Membrane Force x	8	Membrane Force x	RM
	9	Membrane Force y	9	Membrane Force y	RM
	10	Membrane Force xy	10	Membrane Force xy	RM
	11	Membrane Force yz	11	Membrane Force yz	RM
	12	Membrane Force zx	12	Membrane Force zx	RM
	13	Membrane Force z	13	Membrane Force z	RM
	14	Bending Moment x	14	Bending Moment x	RM
	15	Bending Moment y	15	Bending Moment y	RM
	16	Bending Moment xy	16	Bending Moment xy	RM
	17	Bending Moment yz	17	Bending Moment yz	RM
	18	Bending Moment zx	18	Bending Moment zx	RM
	19	Bending Moment z	19	Bending Moment z	RM
			20	Membrane Force x	IP
			21	Membrane Force y	IP
			22	Membrane Force xy	IP
			23	Membrane Force yz	IP
			24	Membrane Force zx	IP
			25	Membrane Force z	IP
			26	Bending Moment x	IP
			27	Bending Moment y	IP
			28	Bending Moment xy	IP
			29	Bending Moment yz	IP
			30	Bending Moment zx	IP
			31	Bending Moment z	IP
	20-32	Repeat items 7-19 for corner 2	32-56	Repeat items 7-31 for corner 2	
	33-45	Repeat items 7-19 for corner 3	57-81	Repeat items 7-31 for corner 3	
	46-58	Repeat items 7-19 for corner 4 (VUQUAD)	82-106	Repeat items 7-31 for corner 4 (VUQUAD)	
VUBEAM (191) for BEAMP	1	VU element ID * 10 + device code			
	2	Parent p-element ID			
	3	CID coordinate system ID			

Table 7-7 Element Force Item Codes Part 2 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		Real/Mag. or Imag/Phase
	Item Code	Item	Item Code	Item	
	4	ICORD (not used)			
	5	VU grid ID for end 1			
	6	x/L position			
	7	Force x	7	Force x	RM
	8	Shear Force y	8	Shear Force y	RM
	9	Shear Force z	9	Shear Force z	RM
	10	Torsional Moment x	10	Torsional Moment x	RM
	11	Bending Moment y	11	Bending Moment y	RM
	12	Bending Moment z	12	Bending Moment z	RM
			13	Force x	IP
			14	Shear Force y	IP
			15	Shear Force z	IP
			16	Torsional Moment x	IP
			17	Bending Moment y	IP
			18	Bending Moment z	IP
	13-20	Repeat items 5-12 for end 2	19-32	Repeat items 5-18 for end 2	

Fluid Virtual Mass Pressure Item Codes

Table 7-8 Fluid Virtual Mass Pressure Item Codes

Element Name	Real Fluid Pressure		Complex Fluid Pressure		
	Code	Item	Code	Item	Real/Mag. or Imag./Phase
Plate	2	Fluid pressure	2	Pressure	RM
Family			3	Pressure	IP

Table 7-9 Heat Transfer Item Codes (Curve type is FLUX.)

Element Name (Code)	Code	Item
Heat Transfer Elements	2	Element type
	3 ¹	
	4 ¹	x gradient
	5 ¹	y gradient
	6 ¹	z gradient
	7	x flux
	8	y flux
	9	z flux
CHBDYE	4	Applied load
(107)	5	Free convection
	6	Forced convection
	7	Radiation
	8	Total
CHBDYG	Same as	Same as CHBDYE
(108)	CHBDYE	
CHBDYP	Same as	Same as CHBDYE
(109)	CHBDYE	

2D Slideline and 3D Surface Contact Item Codes

Table 7-10 Contact Item Codes

Element Name (Code)	Real Element Data		
	Item Code	Slideline Item	3D Surface Item (SOL 600)
CSLIFID (116)	1	Slave grid point	GRID ID
	2	Contact region identification number	(39) Contact Touched Body
	3	Master grid 1	
	4	Master grid 2	
	5	Surface coordinate	(38) Contact Status
	6	Normal force	(35) Normal Force
	7	Shear force	(37) Friction Force
	8	Normal stress	(34) Normal Stress
	9	Shear stress	(36) Friction Stress
	10	Normal gap	
	11	Slip	
	12	Slip ratio (Shear force/u*normal force)	
	13-14	Slip code (Character)	

1. Numbers in parenthesis refer to MARCOUT nodal post codes.
2. 3D Surface contact is available in SOL 600 only.

Table 7-11 Contact Item Code for Nodal Data (SOL 400 only)

Element Code	Item	Real Nodal Data 3D Surface Item (SOL 400 only)
203	1	Grid point ID
	2	First touched body
	3	Second touched body
	4	Third touched body
	5	Contact status (Remark 7)
	6	Normal contact force magnitude
	7	Friction contact force magnitude
	8	Normal stress
	9	Friction stress 1

Table 7-11 Contact Item Code for Nodal Data (SOL 400 only)

Element Code	Item	Real Nodal Data 3D Surface Item (SOL 400 only)
	10	Friction stress 2
	11	Contact normal force component F_{nx}
	12	Contact normal force component F_{ny}
	13	Contact normal force component F_{nz}
	14	Contact friction force component F_{fx}
	15	Contact friction force component F_{fy}
	16	Contact friction force component F_{fz}

Table 7-12 Contact Item Code for Body Data (SOL 400 only)

Element Code	Item	Rigid Body Data 3D Surface Item (SOL 400 Only)
203	1	BCBODY ID
	2-17	[XMAT] _{4x4}
	18	Contact force component RF_x
	19	Contact force component RF_y
	20	Contact force component RF_z
	21	Contact moment component RM_x
	22	Contact moment component RM_y
	23	Contact moment component RM_z

Remarks:

1. F_{nx}, F_{ny}, F_{nz} are normal force components in the global coordinate system.
2. F_{fx}, F_{fy}, F_{fz} are friction force components in the global coordinate system.
3. Normal force magnitude is square root of sum of squares of F_{nx}, F_{ny}, F_{nz} .
4. Friction force magnitude is square root of sum of squares of F_{fx}, F_{fy}, F_{fz} with sign of largest absolute of these values.
5. Contact touched body applies to slave grids (touching node) only.
6. A slave grid may not contact more than 3 bodies.
7. Only slave grids have non-zero contact status or touched bodies.
8. $RF_x, RF_y, RF_z, RM_x, RM_y, RM_z$ are the contact force and moment in the global coordinate system that applied on the centroid of the corresponding rigid body.

9. [XMAT] is the translation matrix to compute the new location of the rigid body.
10. These contact outputs, both [Table 7-11](#) and [Table 7-12](#) are saved in datablock OFCON3D.

Element Strain Energy Item Codes

Table 7-13 Element Strain Energy Item Codes

Element Name	Real Element Data	
	Item Code	Item
Element groups A and B	2	Element strain energy
Element groups A and B	3	Percent of total energy
Element group A	4	Element strain energy density

Remark:

1. Element group A includes elements of CBAR, CBEAM, CBEND, CONROD, CHEXA, CPENTA, CPYRAM, CQUAD4, CQUADR, CROD, CSHEAR, CTETRA, CTRIA3, CTRIA6, CTRIAR, and CTUBE. Element group B includes elements of CELAS1, CELAS2, CELAS3, AND CGAP.

Fatigue Item Codes

Note that the codes of the [Table 7-14](#) through [Table 7-17](#) are limited to those that can be referenced on a DRESP1 entry with RTYPE= FATGUE.

Table 7-14 Fatigue Item Codes for LOC=ELEM on FTGPARM

Element Name	Item Code	Real Element Data Item
Element Group A and Element Group B (for Z1 layer - bottom)	4	Fatigue life in <i>Repeats</i> of the loading sequence
	(-4)	See remark 4.
	5 (-5)	Log of fatigue life in <i>Repeats</i> of the loading sequence
	6 (-6)	Fatigue life in user defined fatigue equivalent units
	7 (-7)	Log of fatigue life in user defined fatigue equivalent units
	8 (-8)	Fatigue damage
	9 (-9)	Log of fatigue damage
Element Group B (for Z2 layer - top)	12 (-12)	Scale factor from Factor of Safety analysis
	18	Fatigue life in <i>Repeats</i> of the loading sequence
	19	Log of fatigue life in <i>Repeats</i> of the loading sequence
	20	Fatigue life in user defined fatigue equivalent units
	21	Log of fatigue life in user defined fatigue equivalent units
	22	Fatigue damage
	23	Log of fatigue damage
	26	Scale factor from Factor of Safety analysis

Remark:

1. Element Group A consists of elements: CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3, CSHEAR
2. Element Group B consists of elements: CQUAD4, CQUAD8, CQUADR, CTRIA6, CTRIAR
3. For CTRIA3 and CSHEAR elements, LOC=ELEM and LOC=NODE or NODA options on FTGPARM use the same item codes in [Table 7-14](#).
4. For Element Group B (shell elements), if the item code is negative (e.g., -4) for layer Z1, this flags Nastran to automatically (and internally) create a DRESP1 with the corresponding item code for layer Z2. This is a convenient method to include responses from both top and bottom by defining only a single DRESP1 entry for any particular set of elements or property sets.

Note: SOL 200 only supports NODE and not NODA on the FTGPARM entry

Table 7-15 Fatigue Item Codes for LOC=NODE or NODA on FTGPARM

Element Name	Item Codes	Real Element Data Item
CTETRA	4,16,28,40	Life in <i>Repeats</i> of the loading sequence
	5,17,29,41	Log of life in <i>Repeats</i> of the loading sequence
	6,18,30,42	Life in user defined fatigue equivalent units
	7,19,31,43	Log of life in user defined equivalent units
	8,20,32,44	Fatigue damage
	9,21,33,45	Log of fatigue damage
	12,24,36,48	Scale factor from Factor of Safety analysis
CQUAD4	4,33,62,91	Life in <i>Repeats</i> of the loading sequence
	(-4,-33,-62,-91)	See remark 8.
	5,34,63,92 (-5,-34,-63,-92)	Log of life in <i>Repeats</i> of the loading sequence
	6,35,64,93 (-6,-35,-64,-93)	Life in user defined fatigue equivalent units
	7, 36, 65, 94 (-7,-36,-65, -94)	Log of life in user defined equivalent units
	8, 37, 66, 95 (-8,-37,-66,-95)	Fatigue damage
	9, 38, 67, 96 (-9,-38,-67-96)	Log of fatigue damage
(for Z2 layer - top)	12, 41, 70, 99 (-12,-41,-70,-99)	Scale factor from Factor of Safety analysis
	18, 47, 76, 105	Life in <i>Repeats</i> of the loading sequence
	19, 48, 77, 106	Log of life in <i>Repeats</i> of the loading sequence
	20, 49, 78, 107	Life in user defined fatigue equivalent units
	21, 50, 79, 108	Log of life in user defined equivalent units
	22, 51, 80, 109	Fatigue damage
	23, 52, 81, 110	Log of fatigue damage
	26, 55, 84, 113	Scale factor from Factor of Safety analysis

Table 7-15 Fatigue Item Codes for LOC=NODE or NODA on FTGPARM (continued)

Element Name	Item Codes	Real Element Data Item
CTRIA6	4, 33, 62	Life in <i>Repeats</i> of the loading sequence
CTRIAR (for Z1 layer - bottom)	(-4, -33, -62)	See remark 8.
	5, 34, 63 (-5, -34, -63)	Log of life in <i>Repeats</i> of the loading sequence
	6, 35, 64 (-6, -35, -64)	Life in user defined fatigue equivalent units
	7, 36, 6 (-7, -36, -6)	Log of life in user defined equivalent units
	8, 37, 66 (-8, -37, -66)	Fatigue damage
	9, 38, 67 (-9, -38, -67)	Log of fatigue damage
	12, 41, 70 (-12, -41, -70)	Scale factor from Factor of Safety analysis
(for Z2 layer - top)	18, 47, 76	Life in <i>Repeats</i> of the loading sequence
	19, 48, 77	Log of life in <i>Repeats</i> of the loading sequence
	20, 50, 79	Life in user defined fatigue equivalent units
	22, 51, 80	Log of life in user defined equivalent units
	23, 52, 81	Fatigue damage
	26, 55, 84	Log of fatigue damage
		Scale factor from Factor of Safety analysis
CPENTA	4,16,28,40,52,64	Life in <i>Repeats</i> of the loading sequence
	5,17,29,41,53,65	Log of life in <i>Repeats</i> of the loading sequence
	6,18,30,42,54,66	Life in user defined fatigue equivalent units
	7,19,31,43,55,67	Log of life in user defined equivalent units
	8,20,32,44,56,68	Fatigue damage
	9,21,33,45,57,69	Log of fatigue damage
	12,24,36,48,60,72	Scale factor from Factor of Safety analysis

Table 7-15 Fatigue Item Codes for LOC=NODE or NODA on FTGPARM (continued)

Element Name	Item Codes	Real Element Data Item
CPYRAM	4,16,28,40,52	Life in <i>Repeats</i> of the loading sequence
	5,17,29,41,53	Log of life in <i>Repeats</i> of the loading sequence
	6,18,30,42,54	Life in user defined fatigue equivalent units
	7,19,31,43,55	Log of life in user defined equivalent units
	8,20,32,44,56	Fatigue damage
	9,21,33,45,57	Log of fatigue damage
	12,24,36,48,60	Scale factor from Factor of Safety analysis
CHEXA	4,16,28,40,52,64,76,88	Life in <i>Repeats</i> of the loading sequence
	5,17,29,41,53,65,77,89	Log of life in <i>Repeats</i> of the loading sequence
	6,18,30,42,54,66,78,90	Life in user defined fatigue equivalent units
	7,19,31,43,55,67,79,91	Log of life in user defined equivalent units
	8,20,32,44,56,68,80,92	Fatigue damage
	9,21,33,45,57,69,81,93	Log of fatigue damage
	12,24,36,48,60,72,84,96	Scale factor from Factor of Safety analysis

Remarks:

1. CTRIA6 and CTRIAR item codes are listed for Grids 1-3, respectively.
2. CQUAD4, CQUAD8, CQUADR, CTETRA item codes are listed for Grids 1-4, respectively.
3. CPYRAM item codes are listed for Grids 1-5, respectively.
4. CPENTA item codes are listed for Grids 1-6, respectively.
5. CHEXA item codes are listed for Grids 1-8, respectively.
6. For CTRIA3 and CSHEAR elements, LOC=ELEM and LOC=NODE or NODA options on FTGPARM use the same item codes in [Table 7-14](#).
7. If it is desired to use a life response (or other item) for all nodes (using LOC=NODE or NODA on FTGPARM), all item codes for the particular item are required, e.g., 4, 16, 27, and 38 for CTETRA (one DRESP1 entry for each item code).
8. For shell elements with bottom (Z1) and top (Z2) layers, if the item code is negative (e.g., -4) for layer Z1, this flags Nastran to automatically (and internally) create a DRESP1 with the corresponding item code for layer Z2. This is a convenient method to include responses from both top and bottom by defining only a single DRESP1 entry for any particular response for a set of elements or property sets. So as an example, to specify the bottom and top response (Life in *Repeats*) for all the grids of the specified element or property set for a CQUAD4 element, use items codes: -4,-27,-50,-73
9. These item codes are used for LOC=NODE or NODA and all other element nodal values of RECOVER on the FTGPARM entry, namely, "SGAGE", "CORNER", "BILIN", and "CUBIC" also.

Table 7-16 Item Codes for Fatigue Analysis of Spot Welds

Element Name	Item Code	Real Element Data Item
(For Top Sheet at Angle = 0.0 degrees)	4 (-4) (-1004)	Fatigue life in <i>Repeats</i> of the loading sequence See remarks 2. and 3.
	5 (-5) (-1005)	Log of fatigue life in <i>Repeats</i> of the loading sequence
	6 (-6) (-1006)	Fatigue life in user defined fatigue equivalent units
	7 (-7) (-1007)	Log of fatigue life in user defined fatigue equivalent units
	8 (-8) (-1008)	Fatigue damage
	9 (-9) (-1009)	Log of fatigue damage
	13 (-13) (-1013)	Scale factor from Factor of Safety analysis
	4+12*NANGLE	Fatigue life in <i>Repeats</i> of the loading sequence
(For Weld Nugget at Angle = 0.0 degrees)	5+12*NANGLE	Log of fatigue life in <i>Repeats</i> of the loading sequence
	6+12*NANGLE	Fatigue life in user defined fatigue equivalent units
	7+12*NANGLE	Log of fatigue life in user defined fatigue equivalent units
	8+12*NANGLE	Fatigue damage
	9+12*NANGLE	Log of fatigue damage
	13+12*NANGLE	Scale factor from Factor of Safety analysis
	4+24*NANGLE	Fatigue life in <i>Repeats</i> of the loading sequence
(For Bottom Sheet at Angle = 0.0 degrees)	5+24*NANGLE	Log of fatigue life in <i>Repeats</i> of the loading sequence
	6+24*NANGLE	Fatigue life in user defined fatigue equivalent units
	7+24*NANGLE	Log of fatigue life in user defined fatigue equivalent units
	8+24*NANGLE	Fatigue damage
	9+24*NANGLE	Log of fatigue damage
	13+24*NANGLE	Scale factor from Factor of Safety analysis

Remarks:

1. Spot weld fatigue item code is related to NANGLE, the number of calculation angles around the weld circumference as defined on the FTGPARM entry. Item codes listed above are for the first angle at zero (0) degrees. For each subsequent angle, the item code should be increased by 12. For example, item code =16 is for "Fatigue life in *Repeats* of the loading sequence" at the Top Sheet for the second angle.

2. If item codes 4-9 or 13 are negative (e.g., -4), this flags Nastran to automatically (and internally) create a DRESP1 with the corresponding item code for the nugget and bottom sheet as well and for all specified angles. This is a convenient method to include responses from all locations and angles by defining only a single DRESP1 entry for any particular response for a set of spot weld elements or property sets. So as an example, to specify the response (Life in *Repeats*) for all the locations and angles, use a single DRESP1 entry with item code: -4.
3. In conjunction with the above remark, if the item code is negative and multiplied by 1000 (e.g., -1004), then the nugget responses are not generated. Only the top and bottom sheet responses are internally generated.

Table 7-17 Item Codes for Fatigue Analysis of Seam Welds

Element Name	Item Code	Real Element Data Item
CQUAD4 CQUAD8 CQUADR	4, 19, 34, 45	Fatigue life in <i>Repeats</i> of the loading sequence
	5, 20, 35, 46	Log of fatigue life in <i>Repeats</i> of the loading sequence
	6, 21, 36, 47	Fatigue life in user defined fatigue equivalent units
	7, 22, 37, 48	Log of fatigue life in user defined fatigue equivalent units
	8, 23, 38, 49	Fatigue damage
	9, 24, 39, 50	Log of fatigue damage
	12, 27, 42, 53	Scale factor from Factor of Safety analysis
CTRIA6 CTRIAR	4, 19, 34	Fatigue life in <i>Repeats</i> of the loading sequence
	5, 20, 35	Log of fatigue life in <i>Repeats</i> of the loading sequence
	6, 21, 36	Fatigue life in user defined fatigue equivalent units
	7, 22, 37	Log of fatigue life in user defined fatigue equivalent units
	8, 23, 38	Fatigue damage
	9, 24, 39	Log of fatigue damage
	12, 27, 42	Scale factor from Factor of Safety analysis

Remarks:

1. CQUAD4, CQUAD8, CQUADR item codes are listed for Grids 1-4, respectively.
2. CTRIA6 and CTRIAR item codes are listed for Grids 1-3, respectively.
3. Note that FTGDEF has a keyword “SEAMW” that may have the field NDSIDi, which lists seam line grids. If this is used, it means that some grids of the elements will not have fatigue life associated with them and are excluded from the analysis.

Equivalent Radiated Power (ERP) Item Codes

Table 7-18 Equivalent Radiated Power (ERP) Item Codes

Element Name	Item Code	Item
QUAD4, CQUADR, CTRIA3,CTRIAR	2	ERP value
QUAD4, CQUADR, CTRIA3,CTRIAR	3	ERP Fraction
QUAD4, CQUADR, CTRIA3,CTRIAR	4	ERP(DB)

8

Degree-of-Freedom Sets

- Degree-of-Freedom Set Definitions
- Degree-of-Freedom Set Bulk Data Entries

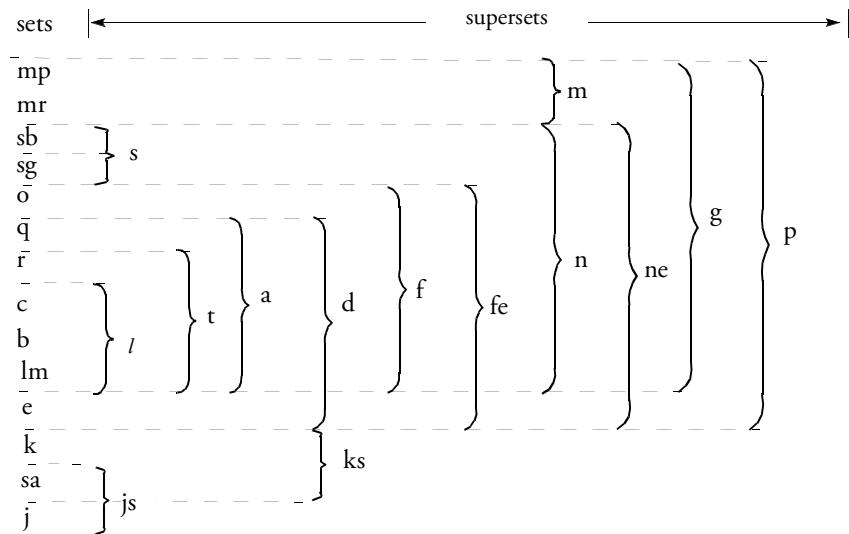
Degree-of-Freedom Set Definitions

Each degree-of-freedom is a member of one mutually exclusive set. Set names have the following definitions:

Set Name	Definition
mp	Degrees-of-freedom eliminated by <u>multipoint</u> constraints.
mr	Degrees-of-freedom eliminated by <u>multipoint</u> constraints created by the <u>rigid</u> elements using the LGELIM method on the Case Control command RIGID.
sb*	Degrees-of-freedom eliminated by <u>single-point</u> constraints that are included in <u>boundary</u> condition changes and by the AUTOSPC feature. (See the sz set)
sg*	Degrees-of-freedom eliminated by <u>single-point</u> constraints that are specified on the PS field on <u>GRID</u> Bulk Data entries.
sz	Degrees-of-freedom eliminated by the AUTOSPC feature.
o	Degrees-of-freedom <u>omitted</u> by structural matrix partitioning.
q	Generalized degrees-of-freedom assigned to component modes and residual vectors.
r	<u>reference</u> degrees-of-freedom used to determine free body motion.
c	Degrees-of-freedom that are free during component mode synthesis or dynamic reduction.
b	Degrees-of-freedom fixed during component mode analysis or dynamic reduction.
lm	<u>Lagrange</u> <u>multiplier</u> degrees-of-freedom created by the rigid elements using the LAGR method on the Case Control command, RIGID.
e	<u>extra</u> degrees-of-freedom introduced in dynamic analysis.
sa	Permanently constrained aerodynamic degrees-of-freedom.
k	Aerodynamic mesh point set for forces and displacements on the aero mesh.
j	Aerodynamic mesh collocation point set (exact physical interpretation is dependent on the aerodynamic theory).

*Strictly speaking, sb and sg are not exclusive with respect to one another. Degrees-of-freedom may exist in both sets simultaneously. Since these sets are not used explicitly in the solution sequences, this need not concern the user. However, those who use these sets in their own DMAPs should avoid redundant specifications when using these sets for partitioning or merging operations. That is, a degree-of-freedom should not be specified on both a PS field of a GRID entry (sg set) and on a selected SPC entry (sb set). Redundant specifications will cause UFM 2120 in the VEC module and behavior listed in [MSC Nastran DMAP Programmer's Guide](#) for the UPARTN module. These sets are exclusive, however, from the other mutually exclusive sets.

Each degree-of-freedom is also a member of one or more combined sets called “supersets.” Supersets have the following definitions:



Set Name	Meaning (+ indicates union of two sets)
$s = sb + sg$	all degrees-of-freedom eliminated by single point constraints
$l = b + c + lm$	the degrees-of-freedom remaining after the reference degrees-of-freedom are removed (degrees-of-freedom left over)
$t = l + r$	the total set of physical boundary degrees-of-freedom for superelements
$a = t + q$	the analysis set used in eigensolution
$d = a + e$	the set used in dynamic analysis by the direct method
$f = a + o$	unconstrained (free) structural degrees-of-freedom
$fe = f + e$	free degrees-of-freedom plus extra degrees-of-freedom
$n = f + s$	all degrees-of-freedom not constrained by multipoint constraints
$ne = n + e$	all degrees-of-freedom not constrained by multipoint constraints plus extra degrees-of-freedom
$m = mp + mr$	all degrees-of-freedom eliminated by multipoint constraints
$g = n + m$	all degrees-of-freedom including scalar degrees-of-freedom
$p = g + e$	all physical degrees-of-freedom including extra point degree-of-freedom
$ks = k + sa$	the union of k and the re-used s-set (6 dof per grid)
$js = j + sa$	the union of j and the re-used s-set (6 dof per grid)

table continued on the next page

Set Name	Meaning (+ indicates union of two sets)
$fr = o + l$	statically independent set minus the statically determinate supports ($fr = f - q - r$)
$v = o + c + r$	the set free to vibrate in dynamic reduction and component mode synthesis
$al = a - lm$	a-set without Lagrange multiplier degree-of-freedoms.
$dl = d - lm$	d-set without Lagrange multiplier degree-of-freedoms.
$gl = g - lm$	g-set without Lagrange multiplier degree-of-freedoms.
$ll = l - lm$	l-set without Lagrange multiplier degree-of-freedoms.
$nf = ne - lm$	ne-set without Lagrange multiplier degree-of-freedoms.
$pl = p - lm$	p-set without Lagrange multiplier degree-of-freedoms.
$tl = t - lm$	t-set without Lagrange multiplier degree-of-freedoms.
$nl = n - lm$	n-set without Lagrange multiplier degree-of-freedoms.
$fl = f - lm$	f-set without Lagrange multiplier degree-of-freedoms.
$ff = fe - lm$	fe-set without Lagrange multiplier degree-of-freedoms.

The a-set and o-set are created in the following ways:

1. If only OMITi entries are present, then the o-set consists of degrees-of-freedom listed explicitly on OMITi entries. The remaining f-set degrees-of-freedom are placed in the b-set, which is a subset of the a-set.
2. If ASETi or QSETi entries are present, then the a-set consists of all degrees-of-freedom listed on ASETi entries and any entries listing its subsets, such as QSETi, SUPPORTi, CSETi, and BSETi entries. Any OMITi entries are redundant. The remaining f-set degrees-of-freedom are placed in the o-set.
3. If there are no ASETi, QSETi, or OMITi entries present but there are SUPPORTi entries present, then the entire f-set is placed in the a-set and the o-set is not created.
4. There must be at least one explicitly ASETi, QSETi, or OMITi entry for the o-set to exist, even if the ASETi, QSETi, or OMITi entry is redundant.

In dynamic analysis, additional vector sets are obtained by a modal transformation derived from real eigenvalue analysis of the a-set. These sets are as follows:

ξ_o = rigid body (zero frequency) modal degrees-of-freedom

ξ_f = finite frequency modal degrees-of-freedom

ξ_i = $\xi_o + \xi_f$, the set of all modal degrees-of-freedom

One vector set is defined that combines physical and modal degrees-of-freedom:

u_h = $\xi_i + u_e$, the set of all modal degrees-of-freedom

The membership of each degree-of-freedom can be printed by use of the Bulk Data entries PARAM,USETPRT and PARAM,USETSEL.

Degree-of-Freedom Set Bulk Data Entries

Degrees-of-freedom are placed in sets as specified by the user on the following Bulk Data entries:

Name	Bulk Data Entry Name
m	MPC, MPCADD, MPCAX, POINTAX, RBAR, RBAR1, RBE1, RBE2, RBE3, RJOINT, RROD, RSPLINE, RSSCON, RTRPLT, RTRPLT1, GMBC, GMSPC*
sb	SPC, SPC1, SPCADD, SPCAX, FLSYM, GMSPC*, BNDGRID, PARAM,AUTOSPC,YES
sz	PARAM,AUTOSPC,YES
sg	GRID, GRIDB, GRDSET (PS field)
o	OMIT, OMIT1, OMITAX, GRID (SEID field), SESET
q	QSET, QSET1
r	SUPPORT, SUPPORT1, SUPAX
c	CSET, CSET1, BNDFREE, BNDFRE1
b	BSET, BSET1, BNDFIX, BNDFIX1
e	EPOINT
sa	CAEROi
k	CAEROi
a	ASET, ASET1, Superelement exterior degrees-of-freedom, CSUPEXT
ap	ACCSPT
rv	RVDOF and RVDOF1
u1 - u6	USET, USET1, SEUSET, and SEUSET1

*Placed in set only if constraints are not specified in the basic coordinate system.

In superelement analysis, the appropriate entry names are preceded by the letters SE, and have a field reserved for the superelement identification number. This identification is used because a boundary (exterior) grid point may be in one mutually exclusive set in one superelement and in a different set in the adjoining superelement. The SE-type entries are internally translated to the following types of entry for the referenced superelement:

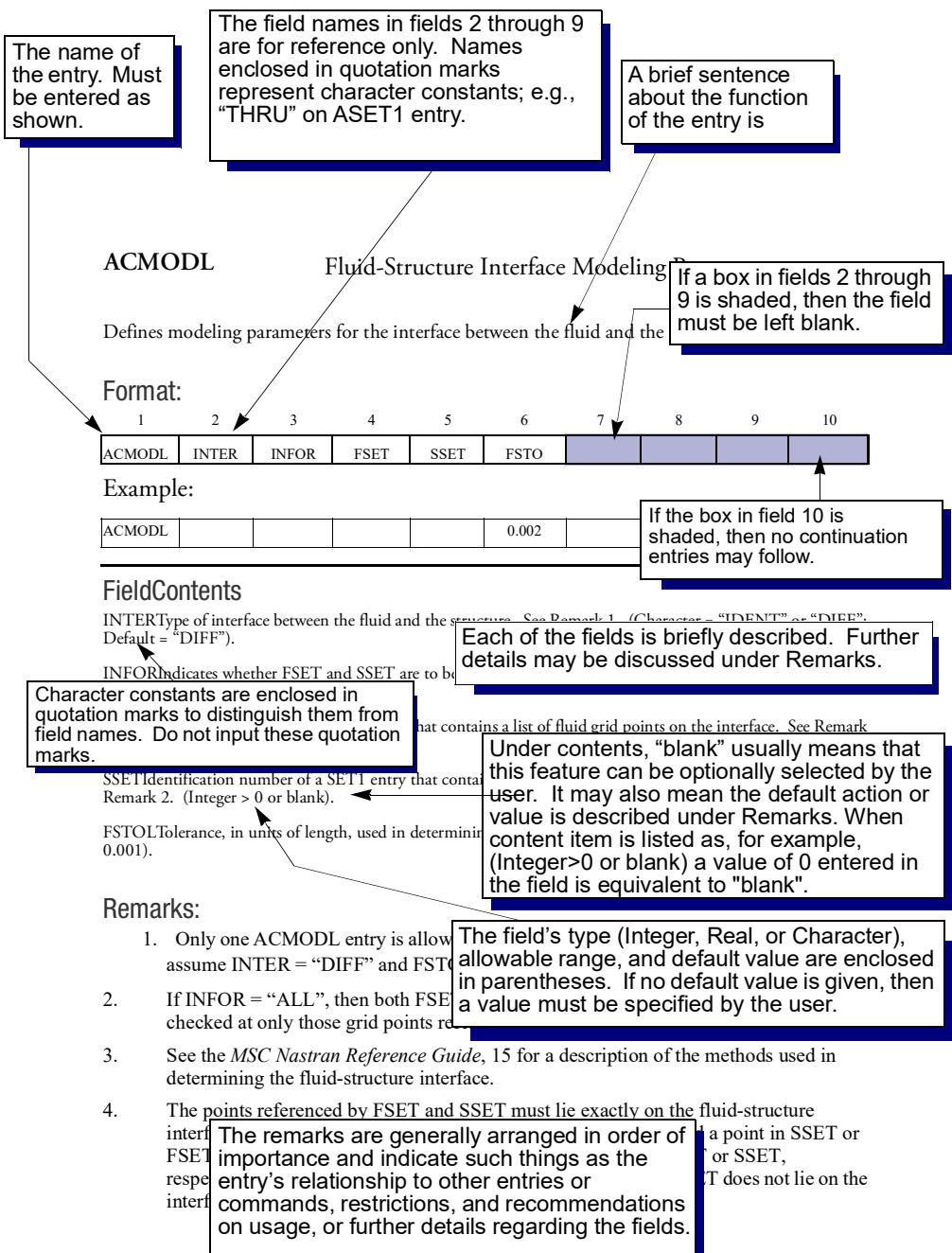
Entry Type	Equivalent Type
SEQSETi	QSETi
SESUP	SUPPORT
SECSETi	CSETi, BNDFREE, DNBDRE1
SEBSETi	BSETi, BNDFIXi

9

Bulk Data Entries

- Key to Descriptions
- Bulk Data Entry Descriptions

Key to Descriptions



The Bulk Data Section

The Bulk Data Section contains entries that specify model geometry, element connectivity, element and material properties, constraints (boundary conditions), and loads. Some entries, such as loads and constraints, are selected by an appropriate Case Control command.

Entries are prepared in either fixed or free field format. The descriptions in this section show only the fixed format. Entries that are used by the MSGMESH program are not included in this guide. For a description of the various format options and the MSGMESH entries, see the [Use of Parameters](#) in the *MSC Nastran Reference Guide*.

New Bulk Data Entries added after the MSC Nastran 2011 version are only valid with the default option IFPSTAR=YES.

Bulk Data Entry Descriptions

Each Bulk Data entry is described as follows:

Description

A brief sentence about the function of the entry is given.

Format

The name of the entry is given in the first field. The subsequent fields are described under the Field and Contents Section. Shaded fields must be left blank. If field 10 is shaded, then no continuation entries are permitted. Character strings enclosed in quotation marks must be specified without the quotation marks as shown in the example.

Example

A typical example is given.

Field and Contents

Each of the fields 2 through 9 that are named in the Format section is briefly described under Contents. The field's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. The field must be specified by the user if no default value is given.

Remarks

The remarks in the Remarks Section are generally arranged in order of importance and indicate such things as how the Bulk Data entry is selected in the Case Control Section, its relationship to other entries, restrictions and recommendations on its use, and further descriptions of the fields.

Format of Bulk Data Entries

Real, Integer, and Character Input Data

Nastran is quite particular about the input requirements for data entry. The three possible types of data entries are Integer, Real, and Character (sometimes called literal, or BCD-binary coded decimal). The three types of data are described as follows:

Integer	Cannot contain a decimal point.
Real	Must contain a decimal point. If however, system cell 444 is set to 9 and the only possible value format is real, then an integer representation (less decimal point) will be converted to real. A single decimal point is not be considered a real zero value. Either 0 or 0. (a digit) must be supplied.
Character	Can be alphanumeric, and with exceptions (listed on specific bulk data entries) should always start with an alpha character. Legal alpha characters are the English: A-Z and a-z. Legal numeric characters are 0-9, and in special circumstances in fields that allow user naming the dash (-) and the underscore (_). In user naming fields use of \$, &, *, =, +, period (.), comma (,) should be avoided as they often have special meaning to the nastran bulk data interpreter. Misspelled required words such as the name of an entry will result in a fatal message. Substitution of non English alphabetic characters will cause a fatal message to be issued.

Real numbers may be entered in a variety of ways. For example, the following are all acceptable versions of the real number seven:

7.0	.7E1	0.7+1
.70+1	7.E+0	70.-1

Free, Small, and Large Field Formats

Nastran has three different field formats for input data:

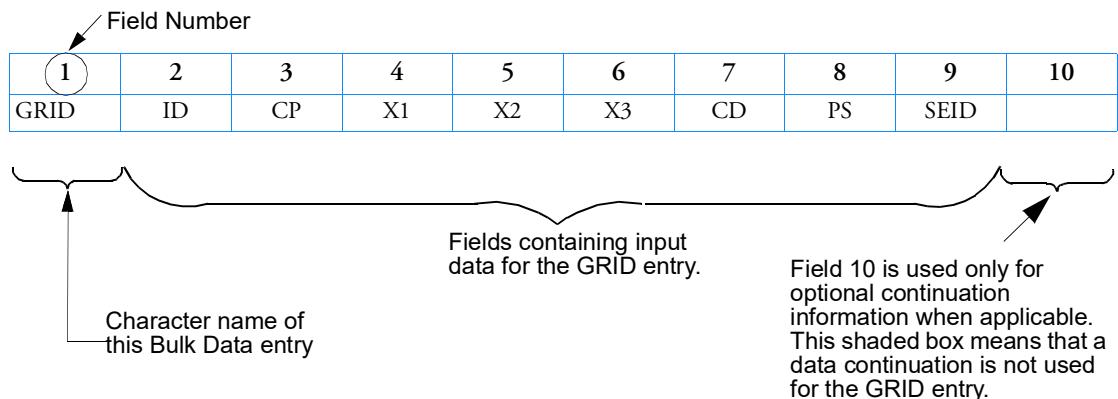
Free Field Format	Input data fields are separated by commas.
Small Field Format	Ten fields of eight characters each.
Large Field Format	Ten fields-eight fields containing actual data are sixteen characters each (fields 2-5 and 6-9). Large fields are used when greater numerical accuracy is required. See the Large Field Format, 1126 for additional information.

The NASTRAN statement, File Management Section, Executive Control Section, and Case Control Section use free field format. The Bulk Data Section allows the use of any of the three formats.

Nastran Bulk Data contains ten fields per input data entry. The first field contains the character name of the Bulk Data item (e.g., GRID, CBAR, MAT1, etc.). Fields two through nine contain data input information

for the Bulk Data entry. The tenth field never contains data—it is reserved for entry continuation information, if applicable.

Consider the format of a typical MSC Nastran Bulk Data entry, the GRID entry, which is used in Nastran to describe the geometry of the structural model.



Example:

1	2	3	4	5	6	7	8	9	10
GRID	2		1.0	-2.0	3.0		136		

We will now represent this example in free field, small field, and large field formats.

Free Field Format

In free field format, data fields are separated by commas or blanks (commas are strongly recommended). The following shows the GRID Bulk Data entry example in free field format:

GRID,2,,1.0,-2.0,3.0,,136

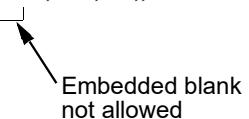
A bracket under the second and third commas is labeled 'Two consecutive commas indicate an empty field'.

The rules for free field format are as follows:

- Free field data entries must start in column 1.
- To skip one field, use two commas in succession. To skip two fields, use three commas in succession (and so on).
- Integer or character fields with more than eight characters cause a fatal error.
- Real numbers with more than eight characters are rounded off and lose some precision. For example, an entry of 1.2345678+2 becomes 123.4568. If more significant digits are needed, use the large field format. If however, NASTRAN IFPSTAR=YES is used, then the entry will automatically be converted to large field format.

- Free field data cannot contain embedded blanks to skip fields. An example of a free field embedded blank is shown:

GRID,2,,1 0,-2.0,3.0,,136



Embedded blank
not allowed

In a continuation line, if a comma is the first character and in the first 12 columns, this line is free format.

- A dollar sign \$ can be used within a line. All data following the \$ on the line will be for comments. An example of a comment is shown:

DISP(PUNCH)=ALL \$ request punched displacement output for all grids

The free field data entry capability in Nastran have been enhanced to support easy to use data input formats. The following examples illustrate the possible forms of the free field data input and the resulting translation to the fixed-field format.

Entry with or without user continuation mnemonics.

MATT9,1101,2 ,3 ,4 ,,,8 ,+P101
+P101,9 ,,,13

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1101	2	3	4				8	+P101
+P101	9				13				

GRID,100,,1.0,0.0,0.0,,456

Translates to:

GRID	100		1.0	0.0	0.0		456		
------	-----	--	-----	-----	-----	--	-----	--	--

The continuation mnemonics are not included because they are not required. This is illustrated by the entry with automatic continuation:

SPC1,100,12456,1,2,3,4,5,6,7,8,9,10

Translates to:

1	2	3	4	5	6	7	8	9	10
SPC1	100	12456	1	2	3	4	5	6	
	7	8	9	10					

If more than 80 characters of data are required, the free field may be continued in the next line provided that the next entry starts with a comma in the first column. The next entry will be a logical continuation of the first. For example, the free-field entry:

MATT9,1151,2 ,3 ,4 ,,,8
,9 ,,,13

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1151	2	3	4				8	
	9				13				

Which is equivalent to:

MATT9,1151,2 ,3 ,4 ,,,8 ,+
,9 ,,,13

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1151	2	3	4				8	
+	9				13				+

The free field data entry can be used to input mixed Small Field, Large Field continuations. Note that the plus (+) and asterisk (*) characters are used to indicate Small Field and Large Field input form respectively when free field data entry is used. For example, the entries:

MATT9*,1302,2 ,4 ,+
+,,,,,13

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9*		1302		2			4		+
+					13				

MATT9,1303,2 ,3 ,4 ,,,8 ,+
*,9 ,,,,+
*,13

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1303	2	3	4				8	+
*		9							
*		13							

MATT9,1355,2 ,3 ,5 ,,,8 ,+
*,10 ,,,+
+,17

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1355	2	3		5			8	+
*			10						+
+	17								

System cell 363 must be set to 1 (i.e., system(363)=1, or STRICTUAI=1) if more than 80 characters of data are required, then the free-field entry is continued by terminating the parent with a comma. The next entry will be a logical continuation of the first. It is not required to end the first entry at any specific point. This is illustrated by the entry:

CHEXA,200, 200, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16,
17, 18, 19, 20

Translates to:

1	2	3	4	5	6	7	8	9	10
CHEXA	200	200	1	2	3	4	5	6	
	7	8	9	10	11	12	13	14	
	15	16	17	18	19	20			

Because of the feature allowing more than 10 fields of data to be entered on one free field entry, IT IS NOT ALLOWED to terminate a single free field entry with a comma. For example:

CHEXA,200, 200, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16,

Small Field Format

Small field format separates a Bulk Data entry into ten equal fields of eight characters each:

8 character field									
1	2	3	4	5	6	7	8	9	10
11	12	13	14	15	16	17	18	19	20
80 characters									

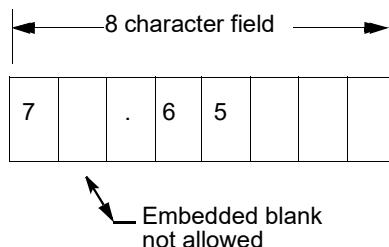
The following is an example of the GRID entry in small field format:

1	2	3	4	5	6	7	8	9	10
GRID	2		1.0	-2.0	3.0		136		

The rules for small field format are as follows:

- Fields 1 and 10 must be left justified.
- Fields 2 through 9 do not need to be either right or left justified, although aligning the data fields is good practice.

- Small field input data cannot contain any embedded blanks. An example of a small field embedded blank is shown:



Large Field Format

A high degree of numerical accuracy is required in some Nastran applications. Large field format is used when small field format does not provide enough significant digits (recall that a minus sign, decimal point, and the "E" in scientific notation count as characters).

Large field format requires (at least) two lines for each entry: the first and last field of each line contains eight columns, and the fields in between contain 16 columns. Short field becomes two lines. Large field entries are denoted by an asterisk (*) immediately following the character string in field 1A of the first line and immediately preceding the character string in field 1B of the second line.

The following is an example of the GRID Bulk Data entry example in large field format:

First Line: (Left half of single field)

Field	1A	2	3	4	5	6
	GRID*	2		1.0	-2.0	GRID10
	8	16	16	16	16	8

columns

Second Line: (Right half of single field)

Field	1B	6	7	8	9	10B
	*GRID10	3.0		136		
	8	16	16	16	16	8

columns

Continuations

Some Bulk Data entries require more than eight fields (72 columns) of data. Continuations are required in such cases. To do this, a parent entry (the first line) is followed by one or more continuation entries on subsequent lines. For example, consider the following PBAR simple beam property entry (do not worry about what each field represents-this will be explained later):

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						

Continuation Example:

PBAR	39	6	2.9	1.86	2.92	.48			+PB1
+PB1	0.	0.	0.	1.	1.	1.	1.	0.	+PB2
+PB2	.86	.86							

+PB1 in field 10 of the parent entry is an arbitrary (and unique) user-defined pointer to field 1 of the second line. +PB2 in the second line points the third line, and so on.

Continuation fields can also be generated automatically by Nastran (this approach is the recommended practice). To automatically generate a continuation, the continuation line (or lines) must immediately follow the parent Bulk Data entry. In addition, fields 1 and 10 of the continuation line (or lines) must be left blank. In the case of double-width generated continuations are not blank in field 1, but have an "*" in column 1. Nastran will then generate unique continuations for you. This process is illustrated in the following example:

Input (.DAT) file:

```
CHEXA,      1,      10,      3,      5,      7,      1,      15,      17,
,      19,      13,      4,      6,      8,      2,      10,      11,
```

Output (.F06) file:

```
GEAR TOOTH EXAMPLESEPTEMBER 29, 1993 MSC.Nastran 9/ 4/91 PAGE 3
S O R T E D B U L K D A T A E C H O
CARD
COUNT. 1 .. 2 .. 3 .. 4 .. 5 .. 6 .. 7 .. 8 .. 9 .. 10 ..
1- CHEXA      1      10      3      5      7      1      15      17      +000001
2-++000001    19      13      4      6      8      2      10      11      +000002
3-++000002    12      9      16      18      20      14      +000003
```

Blank lines are allowed if they are in between entries but not allowed if they are in an entry and further continuations are followed if ifpstar is yes, for example, the blank line in below entry is not allowed. To ignore those blank lines, set system(767) to 1.

CHEXA, 1, 10, 3, 5, 7, 1, 15, 17,

```
, 19, 13, 4, 6, 8, 2, 10, 11,
, 12, 9, 16, 18, 20, 14
```

Nastran Continuation fields (fields one and ten) are replicated using the following conventions:

- Only letters of the alphabet and integers may be used. They are coded into a base 36 number. That is, the sequence of numbers is 0,1, 2, ..., 8, 9, A, B, ...
- The first character in field one or ten is not incremented.
- The continuation fields are incremented by +1 regardless of the value specified by the user.

4. The number of characters in an incremented field will not be increased. For example, if the first field is "0", the thirty-seventh field will also be "0", resulting in an illegal entry. A method to solve this problem would be to start with a first field of "00". This will provide thirty-six squared unique fields.
5. At least one field in fields 2 through 8 of continuation entries must be non-blank.

Replication

Replication is a limited data generation capability which may be used in a fixed or free-field format and is not currently fully supported with NASTRAN SYSTEM(444)=1 and its use, therefore, is not recommended. Also replication is not at all currently supported with SimX reading a user-generated input file and strange models may be displayed.

1. Duplication of fields from the preceding entry is accomplished by coding the symbol =.
2. Duplication of all trailing fields from the preceding entry is accomplished by coding the symbol ==.
3. Incrementing a value from the previous entry is indicated by coding *x or *(x), where x is the value of the increment. "x" should be a real number for real fields or an integer for integer fields.
4. Repeated replication is indicated by coding =n or =(n), where n is the number of images to be generated using the values of the increments on the preceding entry.
5. Data items may be enclosed with parentheses or the parentheses may be deleted.
6. The MSGMESH capability includes the capabilities described here, plus the following capabilities as long as NASTRAN MESH is specified in the File Management Section.
 - Continuation entry fields may be incremented or decremented.
 - Repeated replication is indicated by coding =(n) in field 1, where n is number of entry images to be generated using the values of increments from the current or preceding replication entry.

Entered entries:

GRID, 101, 17, 1.0, 10.5,,17,3456

= , *1, =, *0.2, ==

=3

Generated entries:

GRID	101	17	1.0	10.5		17	3456		
GRID	102	17	1.2	10.5		17	3456		
GRID	103	17	1.4	10.5		17	3456		
GRID	104	17	1.6	10.5		17	3456		
GRID	105	17	1.8	10.5		17	3456		

- A blank in field 1 indicates immediate continuation entry replication. The default continuation entry increment is 1. Example:

```
BSET1,123,1,2,3,4,5,6,7
,,*7,*7,*7,*7,*7,*7
=(3)
```

Generated entries:

BSET1	123	1	2	3	4	5	6	7	+00001
++00001		8	9	10	11	12	13	14	+00002
++00002		15	16	17	18	19	20	21	+00003
++00003		22	23	24	25	26	27	28	+00004
++00004		29	30	31	32	33	34	35	+00005

- A “=(D)” in field 1 indicates delayed continuation entry replication. A maximum of 9 entries may be replicated as a group. The default continuation entry increment is 10. Example:

Entered entries:

```
CTRIA3,10,1,1,10,11/+C1
=(D),*(1),=,*(1),*(1)/*(20)
+C1,,,2.0,1.0,1.0
=(2),==
```

Generated entries:

CTRIA3	10	1	1	10	11				+C1
+C1			2.0	1.0	1.0				
CTRIA3	11	1	1	11	12				+C21
+C21			2.0	1.0	1.0				
CTRIA3	12	1	1	12	13				+C41
+C41			2.0	1.0	1.0				

- Parentheses are optional on replication entries and an equal sign may replace an asterisk.

The following is an example of the use of replication, automatic continuation field generation, and the free field format:

GRID, 101, 17, 1.0, 10.5,,17,3456

=,*1,=,*0.2, *(0.1), == \$ COMMENTS MAY APPEAR AFTER \$

=3

EIGR,13,GIV,,30.

,MASS

CBAR,1 ,1 ,101 ,102,0.,0.,1.,,+0

=,*1,=,*1,*1====*1

+0,56

*1,=\$

The above free-field entries will generate the following Bulk Data in the 8-column format, as seen in the SORTED BULK DATA ECHO:

Note: A “,” should always be used after the “*1” for the continuation increment even if fixed field format is being used.

CBAR	1	1	101	102	0.	0.	1.		+0
+0	56								
CBAR	2	1	102	103	0.	0.	1.		+1
+1	56								
EIGR	13	GIV		30.					+0000001
++000001	MASS								
GRID	101	17	1.0	10.5		17	3456		
GRID	102	17	1.2	10.6		17	3456		
GRID	103	17	1.4	10.7		17	3456		
GRID	104	17	1.6	10.8		17	3456		
GRID	105	17	1.8	10.9		17	3456		

The automatically generated continuation entries start with the number 1, are incremented by 1, and are padded with zeros and plus signs as shown above. If this feature is used, it is the user's responsibility not to enter continuation entries that also use this convention. In particular, data generated on another run and then written to the PUNCH file with the ECHO=PUNCH, will cause problems when introduced into other data with blank continuation fields.

Identifiers

MSC Nastran is designed to use Identifiers for many quantities, the most common include:

ID	Grid Identification number
EID	Element Identification number
PID	Property Identification number
MID	Material Identification number
CID	Coordinate System number

When these identifications are given they must be unique in their class. All IDs defined on GRID, GRIDB, GRIDF, GRIDS, SPOINT and EPOINT must be unique.

In the case of Element Identification (EID), elements are generally given by "C" entries, such as CBAR, CBEAM, CQUAD, CHEXA, but rigid elements such as RBAR, RBE1, RBE2 are also treated as elements with respect to the EID, and should also be part of this class. All EID must be unique.

All Identification numbers must be greater than zero (example EID > 0)

Furthermore Element Identification and Grid Identification must be less than 100,000,000 to insure data integrity.

There are two types of Property entries, primary ones and secondary ones. The property id on the primary property entries must be unique. These property identifier numbers are referenced on either "C" options defining element connectivity or "G" options for p-element technology. The correspondence between these options is given below.

Referenced by	Primary Property Names
CHBDYP	BDYOR
CHBDYG	BDYOR
CHBDYE	BDYOR
CAABSF	PAABSF
CACINF3	PACINF
CACINF4	PACINF
CAERO1	PAERO1
CAERO2	PAERO2
CAERO3	PAERO3
CAERO4	PAERO4
CAERO5	PAERO5
CAXISYM	PAXISYM
CBAR	PBAR,PBRL, PBRSECT
CBEAM	PBEAM, PBCOMP, PBEAML, PBMSECT, PDISCR, PBMARB6, PMMNUM6
CBEAM3	PBEAM3, PBMSECT
CBELT	PBELTD
CBEND	PBEND
CBUSH	PBUSH, PBUSHT
CBUSH1D	PBUSH1D
CBUSH2D	PBUSH2D
CCONEAX	PCONEAX
CDAMP1	PDAMP
CDAMP1D	PDAMP
CDAMP3	PDAMP
CDAMP5	PDAMP5
CDUMi	PDUMi
CELAS1	PELAS1
CELAS1D	PELAS1
CELAS3	PELAS
CFAST	PFAST
CGAP	PGAP

Referenced by	Primary Property Names
CHACAB	PACABS
CHACBR	PACBAR
CHBDYP	PHBDY
CHEXA	PSOLID, PLSOLID, PCOMPLS
CIFHEX	PCOHE
CIFPENT	PCOHE
CIFQDT	PCOHE
CIFQUAD	PCOHE
CMARKB2	PMARKER
CMARKN1	PMARKER
CMASS1	PMASS
CMASS3	PMASS
CMREBAI	PMREBAI
CMREBAR	PMREBAR
CONSPOT	PBSPOT
CPENTA	PSOLID, PLSOLID
CQUAD	PLPLANE,PLCOMP
CQUAD4	PSHELL,PCOMP,PCOMPG, PLPLANE,PSHELL1,PSHELLD,PLCOMP
CQUAD8	PSHELL,PCOMP,PCOMPG,PLPLANE,PLCOMP
CQUADR	PSHELL,PCOMP,PCOMPG,PLPLANE
CQUADX	PAXSYMH, PLPLANE,PLCOMP
CRAC2D	PRAC2D
CRAC3D	PRAC3D
CROD	PROD,PBELTD
CSEAM	PSEAM
CSHEAR	PSHEAR
CSPH	PSPH
CSPR	PSPRMAT
CSSHLL	PSSHL
CSSHLLH	PSSHL
CSSHLP	PSSHL
CTETRA	PSOLID, PLSOLID
CTQUAD	PTSHELL

Referenced by	Primary Property Names
CTRIA3	PSHELL,PCOMP,PCOMPG, PLPLANE, PSHELL1
CTRIA6	PSHELL,PCOMP,PCOMPG,PLPLANE
CTRIAR	PSHELL,PCOMP,PCOMPG, PLPLANE
CTRIAX	PAXSYMH
CTTRIA	PTSHELL
CTUBE	PTUBE
CVISC	PVISC
CWELD	PWELD
GMINTC	PINTC
GMINTS	PINTS

The secondary Property entries use the same property id as the primary property id. The relationship between these two is shown below.

Primary	Secondary
PSOLID	PSLDN1
PLPLANE	PSHLN2
PCOMP	PCOMPA
PBAR	PBARN1
PBEAM	PBEAM71
PBEAM	PBEMN1
PBEAML	PBEMN1
PCOMP	PCOMPF
PCOMPG	PCOMPF
PDAMP	PDAMPT
PELAS	PELAST
PROD	PROND1
PSHEAR	PSHEARN
PSHELL	PSHLN1
PCOMP	PSHLN1

There are two types of Material entries, primary ones and secondary ones. The material id on the primary material entries must be unique. The material id on the secondary Material entry must agree with the material

id on the associated primary Material entry. The correlation between primary and secondary Material entries is given below.

Material Model	Primary	Secondary											
Isotropic	MAT1	MATT1	MATS1	MATVP	MATEP	MATTEP	MATF	MATTF	MATF1	MATFTG	MATVE	MATTVE	
Shell Anisotropic	MAT2	MATT2		MATVP	MATEP	MATTEP	MATF	MATTF					
Planar Orthotropic	MAT3	MATT3	MATS3	MATVP	MATEP	MATTEP	MATF	MATTF					
Hypoelastic User Sub.	MATUSR	MATTUSR											
General Orthotropic	MATORT	MATTORT	MATSORT				MATF	MATTF					
Shell Orthotropic	MAT8	MATT8	MATS8				MATF	MATTF					
General Anisotropic	MAT9	MATT9											
Genel Hyperelastic	MATHE	MATTHE									MATVE	MATTVE	
Gasket	MATG	MATTG											
5th order Mooney-Rivlin	MATHP												
Advanced NLELAST	MATNLE												
Shape Memory	MATSMA												
Isotropic Poroelastic	MATPE1												
CZM Material SOL 400	MCOHE												
CZM Material SOL 600	MDELAM												
Composite Mixture	MIXTURE												
NLELAST	MATNLE												
Digimat Composite	MATDIGI												
Isotropic Heat Transfer	MAT4	MATT4											
Anisotropic Heat Transfer	MAT5	MATT5											

Bulk Data Entry Summary

This section contains a summary of all Bulk Data entries. The entries are categorized as Geometry, Elements, Material Properties, Constraints, Loads, Solution Control, and Miscellaneous. Entries that are exclusive to SOL 600 have been grouped together at the end of the summary. Entries that are exclusive to Explicit Nonlinear analysis (SOL 700) have also been grouped together at the end of the summary.

Constraints and Partitioning

Component Mode Boundary Conditions

BNDFIX	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.
BNDFIX1	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.
BNDFREE	Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.
BNDFRE1	Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.
BSET	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
BSET1	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
CSET	Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.
CSET1	Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.
QSET	Defines generalized degrees-of-freedom (q-set) to be used for component mode synthesis.
QSET1	Defines generalized degrees-of-freedom (q-set) to be used for component mode synthesis.
SEBSET	Defines boundary degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
SEBSET1	Defines fixed boundary points for superelement.
SECSET	Defines boundary degrees-of-freedom to be free (c-set) during component mode synthesis calculations.
SECSET1	Defines boundary degrees-of-freedom to be free (c-set) during component mode synthesis calculations.
SENQSET	Defines number of internally generated scalar points for superelement dynamic reduction.
SEQSET	Defines the generalized degrees-of-freedom of the superelement to be used in component mode synthesis.
SEQSET1	Defines the generalized degrees-of-freedom of the superelement to be used in component mode synthesis.
SESUP	Defines determinate reaction superelement degrees-of-freedom in a free-body analysis.

Free Body Supports

CYSUP	Defines fictitious supports for cyclic symmetry analysis.
SUPAX	Defines determinate reaction degrees-of-freedom in free bodies for conical shell analysis.
SUPPORTi	Defines degrees-of-freedom for determinate reactions.

Multipoint Constraints

MONSUM	Defines a new monitor result that is the weighted sum of existing monitor results.
MPC	Defines a linear relationship for two or more degrees-of-freedom.
MPCADD	Defines a multipoint constraint set as a union of multipoint constraint sets defined via MPC entries.
MPCD	Defines a load selectable value for nonhomogeneous multi-point constraint.
MPCY	Defines a linear nonhomogeneous relationship for two or more degrees-of-freedom.
MPCAX	Defines multipoint constraints for conical shell problems.
POINTAX	Defines multipoint constraints for point on conical shell.
RBAR	Defines multipoint constraints for rigid bar.
RBEi	Defines multipoint constraints for RBE1, RBE2, RBE3.
RROD	Defines multipoint constraints for rigid rod.
RSPLINE	Defines multipoint constraints for spline element.
RTRPLT	Defines multipoint constraints for rigid triangular plate.

p-element Geometry Constraints

GMBC	Defines enforced displacements for GRID, FEEDGE, FEFACE, GMCURV, and GMSURF entries.
GMSPC	Defines constraints for entries.

Partitioning

ASET	Defines degrees-of-freedom in the analysis set (a-set).
ASET1	Defines degrees-of-freedom in the analysis set (a-set).
CSUPEXT	Assigns exterior points to a superelement.
GRID	Defines interior points for a superelement.
OMIT	Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).
OMIT1	Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).
OMITAX	Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).
RELEASE	Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.

SEELT	Reassigns superelement boundary elements to an upstream superelement.
SESET	Defines interior grid points for a superelement.

Single Point Constraints

FLSYM	Symmetry control for boundary in axisymmetric fluid problem.
GRID	Includes single point constraint definition.
GRIDB	Includes single point constraint definition.
GRDSET	Includes default for single point constraints.
SPC	Defines a set of single-point constraints and enforced motion (enforced displacements in static analysis and enforced displacements, velocities or acceleration in dynamic analysis).
SPC1	Defines a set of single point constraints.
SPCADD	Defines a single-point constraint set as a union of single-point constraint sets defined on SPC or SPC1 entries.
SPCAX	Defines a set of single-point constraints or enforced displacements for conical shell coordinates.
SPCOFF	Defines degrees-of-freedom to be excluded from the AUTOSPC operation.
SPCOFF1	Defines a set of degrees-of-freedom to be excluded from the AUTOSPC operation.

User Sets

DEFUSER	Defines new names for degree-of-freedom sets.
SEUSER	Defines a degree-of-freedom set for a superelement.
SEUSER1	Defines a degree-of-freedom set for a superelement.
USER	Defines a degree-of-freedom set.
USER1	Defines a degrees-of-freedom set.

Elements

A summary of the capabilities and characteristics of the small strain elements is available in [Element Summary -- Conventional MSC Nastran Elements](#) in the *MSC Nastran Reference Guide*.

Aerodynamic Elements

AEFACT	Defines real numbers for aeroelastic analysis.
AELINK	Defines relationships between or among AESTAT and AESURF entries.
AELIST	Defines a list of aerodynamic elements to undergo the motion prescribed with the AESURF Bulk Data entry for static aeroelasticity. Also defines server specific integer data for external spline methods.
AEQUAD4	Defines the connectivity of a quadrilateral aerodynamic element.

AESTAT	Specifies rigid body motions to be used as trim variables in static aeroelasticity.
AESURF	Specifies an aerodynamic control surface as a member of the set of aerodynamic extra points.
AESURFS	Optional specification of the structural nodes associated with an aerodynamic control surface that has been defined on an AESURF entry.
AETRIA3	Defines the connectivity of a triangular aerodynamic element.
CAERO1	Defines an aerodynamic macro element (panel) in terms of two leading edge locations and side chords.
CAERO2	Defines aerodynamic slender body and interference elements for Doublet-Lattice aerodynamics.
CAERO3	Defines the aerodynamic edges of a Mach Box lifting surface. If no cranks are present, this entry defines the aerodynamic Mach Box lifting surface.
CAERO4	Defines an aerodynamic macro element for Strip theory.
CAERO5	Defines an aerodynamic macro element for Piston theory.
CSSCHD	Defines a scheduled control surface deflection as a function of Mach number and angle of attack.
PAERO1	Defines associated bodies for the panels in the Doublet-Lattice method.
PAERO2	Defines the cross-sectional properties of aerodynamic bodies.
PAERO3	Defines the number of Mach boxes in the flow direction and the location of cranks and control surfaces of a Mach box lifting surface.
PAERO4	Defines properties of each strip element for Strip theory.
PAEROS	Defines properties of each strip element for Piston theory.

Aerodynamic to Structure Interconnection

AELISTC	Defines a list of 8-character strings.
SET1	Defines a list of structural grid points.
SET2	Defines a list of structural grid points in terms of aerodynamic macro elements.
SET3	Defines a list of grids, elements or points.
SPBLND1	Defines a strip based blending of two splines.
SPBLND2	Defines a curve based blending of two splines.
SPLINE1	Defines a surface spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.
SPLINE2	Defines a beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.
SPLINE3	Defines a constraint equation for aeroelastic problems. Useful for control surface constraints.

SPLINE4	Defines a curved surface spline for interpolating motion and/or forces for aeroelastic problems on general aerodynamic geometries using either the Infinite Plate, Thin Plate or Finite Plate splining method.
SPLINE5	Defines a 1D beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by irregular arrays of aerodynamic points.
SPLINE6	Defines a 6DOF or 3DOF finite surface spline for interpolating motion and/or forces between two meshes.
SPLINE7	Defines a 6DOF finite beam spline for interpolating motion and/or forces between two meshes.
SPLEXIN	Defines the input for a spline that will be evaluated with a user-supplied procedure.
SPRINRB	Defines a rigid body spline for interpolating motion or forces for aeroelastic problems on general aerodynamic geometries.
SPRELAX	Defines relaxation of a spline based on an adjacent spline.

Axisymmetric Elements

CAXISYM	Defines an axisymmetric shell.
CCONEAX	Defines a conical shell element.
CQUADX	Defines an axisymmetric quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.
CTRIAX	Defines an axisymmetric triangular element with up to 6 grid points for use in fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.
CTRIAX6	Defines an isoparametric and axisymmetric triangular cross section ring element with midside grid points.
MREVERS	Defines which elements, if any, required node numbering to be reversed in SOL 600.
PCONEAX	Defines the properties of a conical shell element described on a CCONEAX entry.

Cohesive Zone Modeling Elements

CIFHEX	Defines a solid interface cohesive zone modeling element in SOL 400.
CIFPENT	Defines a solid interface cohesive zone modeling element in SOL 400.
CIFQDX	Defines an axisymmetric interface cohesive zone modeling element in SOL 400.
CIFQUAD	Defines a planar interface element for cohesive zone modeling element in SOL 400.
MCOHE	Defines damage and cohesive energy for interface zone modeling elements in SOL 400.
PCOHE	Defines the properties of a cohesive interface zone modeling elements in SOL 400.

Connector Elements

CFAST	Defines a fastener with material orientation connecting two surface patches.
CSEAM	Defines a seam-line connecting two surfaces.

CSLOT3	Defines an element connecting three points that solve the wave equation in two dimensions.
CSLOT4	Defines an element connecting four points that solve the wave equation in two dimensions.
CWELD	Defines a weld or fastener connecting two surface patches or points.
PFAST	Defines the CFAST fastener property values.
PSEAM	Defines the CSEAM fastener property values.
PWELD	Defines the property of connector (CWELD) elements.
SWLDPRM	Overrides default values of parameters for connector search.

Slidelines Contact (SOL 106, 129)

BCOMP	Defines the parameters for a contact region and its properties.
BFRIC	Defines frictional properties between two bodies in contact.
BLSEG	Defines a curve that consists of a number of line segments via grid numbers that may come in contact with another body.
BWIDTH	Defines widths or thicknesses for line segments in 3-D or 2-D slideline contact defined in the corresponding BLSEG Bulk Data entry.

Gap Elements

CGAP	Defines a gap or friction element.
PGAP	Defines the properties of the gap element (CGAP entry).

Crack Tip Elements

CRAC2D	Defines a two-dimensional crack tip element.
CRAC3D	Defines a three-dimensional crack tip element.
PRAC2D	Defines the properties and stress evaluation techniques to be used with the CRAC2D structural element.
PRAC3D	Defines the properties of the CRAC3D structural element.

Damping Elements

CBUSH1D	See line elements.
CDAMP1	Defines a scalar damper element.
CVISC	Defines a viscous damper element.
DAMPING	Specifies the values for parameter damping and/or selects optional HYBRID damping.
HYBDAMP	Specifies hybrid damping parameters.
PBUSH1D	See line elements.

PDAMP	Specifies the damping value of a scalar damper element using defined CDAMP1 or CDAMP3 entries.
PDAMP5	Defines the damping multiplier and references the material properties for damping. CDAMP5 is intended for heat transfer analysis only.
PDAMPT	Defines the frequency-dependent properties for a PDAMP Bulk Data entry.
PVISC	Defines properties of a one-dimensional viscous damping element (CVISC entry).
ROTHYBD	Defines Hybrid damping for rotordynamics.

Dummy Elements

ADUMi	Defines attributes of the dummy elements ($1 \leq i \leq 9$)
CDUMi	Defines a dummy element ($1 \leq i \leq 9$)
PDUMi	Defines the properties of a dummy element ($1 \leq i \leq 9$). Referenced by the CDUMi entry.
POTEL	Defines a one-dimensional dummy element for use in plotting.

Fluid and Acoustic Elements

CAABSF	Defines a frequency-dependent acoustic absorber element in coupled fluid-structural analysis.
CACINFi	Defines few types of acoustic infinite elements.
CAXIFI	Defines an axisymmetric fluid element that connects $i = 2, 3$, or 4 fluid points.
CAXISYM	Defines two or three node axisymmetric thick shell elements.
CFLUIDi	Defines three types of fluid elements for an axisymmetric fluid model.
CHACAB	Defines the acoustic absorber element in coupled fluid-structural analysis.
CHACBR	Defines the acoustic barrier element.
CHEXA	Connection definition for a pentahedron element in coupled fluid-structural analysis.
CPENTA	Connection definition for a tetrahedron element in coupled fluid-structural analysis.
CSLOTi	Defines slot element for acoustic cavity analysis.
CTETRA	Defines the connections of the four-sided solid element with four to ten grid points.
ELIST	Defines a list of structural elements for virtual fluid mass.
MAT10	Fluid Material Property Definition.
PAABSF	Defines the properties of a frequency-dependent acoustic absorber element.
PACABS	Defines the properties of the acoustic absorber element.
PACBAR	Defines the properties of the acoustic barrier element.
PACINF	Defines the properties of acoustic infinite elements.
PANEL	Selects the set of structural grid points that define one or more panels.

PSOLID	Defines the fluid properties of solid elements (CHEXA, CPENTA, and CTETRA entries).
SET1	Defines a list of structural grid points for aerodynamic analysis, XY-plots for SORT1 output, and the PANEL entry.

Equivalent Radiated Power

ERPPNL	Equivalent Radiated Power Definition
------------------------	--------------------------------------

Poroelasticity (PEM)

ACTRIM	ACTRAN Trimmed Material Matrices for SOL 108/111
ACLOAD	ACTRAN Acoustic Pressure Load Matrices for SOL 108/111
MATPE1	Isotropic Poroelastic Material Property definition
MATF1	Frequency Dependent Isotropic Material Definition

Heat Transfer Elements

BDYOR	Defines default values for the CHBDYP, CHBDYG, and CHBDYE entries.
CHBDYi	Connection definition for surface element (CHBDYE, CHBDYG, CHBDYP).
CONTRLT	Thermal control element for heat transfer analysis.
PHBDY	A property entry referenced by CHBDYP entries to give auxiliary geometric information for boundary condition surface elements.

The PRODN1, PSHLN1, PSHLN2, PSLDN1, PSHEARN, PLCOMP, and PCOMPLS may be used to extend the nonlinear capabilities of heat transfer elements in SOL 400. PCOMPLS may also be used to extend the nonlinear capabilities of heat transfer elements in all linear solutions between SOL101 and SOL112.

The following elastic elements may also be used as heat conduction elements:

Linear:	CBAR, CROD, CONROD, CTUBE, CBEAM, CBEND.
Membrane:	CTRIA3, CTRIA6, CQUAD4, CQUAD8.
Axisymmetric:	CTRIAX6.
Solid:	CTETRA, CHEXA, CPENTA.

Line Elements

BAROR	Default for orientation and property for CBAR.
BEAMOR	Defines default values for field 3 and fields 6 through 8 of the CBEAM entry.
CBAR	Defines a simple beam element.
CBEAM	Defines a beam element.

CBEAM3	Defines a three-node beam element.
CBEND	Defines a curved beam, curved pipe, or elbow element.
CBUSH1D	Defines the connectivity of a one-dimensional spring and viscous damper element.
CFAST	Defines a fastener with material orientation connecting two surface patches.
CINTC	Defines a line interface element with specified boundaries.
CMREBAI	Defines Rebar elements and matching “Matrix” solid element using the Marc REBAR with INSERT Method. (SOL 600)
CMREBAR	Defines Rebar elements with matching “Matrix” solid elements using the Marc REBAR without INSERT Method. (SOL 600)
CONROD	Defines a rod element without reference to a property entry.
CROD	Defines a tension-compression-torsion element.
CTUBE	Defines a tension-compression-torsion tube element.
CWELD	Defines a weld or fastener connecting two surface patches or points.
PBAR	Defines the properties of a simple beam element (CBAR entry).
PBARL	Defines the properties of a simple beam element (CBAR entry) by cross-sectional dimensions.
PBARN1	Specifies additional nonlinear properties for elements that point to a PBAR or PBARL entry.
PBCOMP	Alternate form of the PBEAM entry to define properties of a uniform cross-sectional beam referenced by a CBEAM entry.
PBEAM	Defines the properties of a beam element (CBEAM entry). This element may be used to model tapered beams.
PBEAM3	Defines the properties of a three-node beam element (CBEAM3 entry).
PBEAML	Defines the properties of a beam element by cross-sectional dimensions.
PBEMN1	Specifies additional nonlinear properties for elements that point to a PBEAM or PBEAML entry.
PBEND	Defines the properties of a curved beam, curved pipe, or elbow element (CBEND entry).
PFAST	Defines the CFAST fastener property values.
PBMSECT	Defines the shape of arbitrary cross-section for CBEAM element.
PBRSECT	Defines the shape of arbitrary cross-section for CBAR element.
PBUSH1D	Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).
PBUSH2D	Defines linear and nonlinear properties of a two-dimensional element (CBUSH2D entry).
PMREBAI	Defines Rebar property information for CMREBAI elements. (SOL 600)
PMREBAR	Defines Rebar property information for CMREBAR elements. (SOL 600)

PROD	Defines the properties of a rod element (CROD entry).
PRODN1	Defines nonlinear property extensions for the PROD in SOL 400.
PTUBE	Defines the properties of a thin-walled cylindrical tube element (CTUBE entry).
PWELD	Defines the properties of connector (CWELD) elements.

Mass Elements

CMASSi	Connection definition for scalar mass, also property definition for i=2 or 4.
CONM1	Defines a 6 x 6 symmetric mass matrix at a geometric grid point.
CONM2	Defines concentrated mass at a grid point.
PMASS	Specifies the mass value of a scalar mass element (CMASS1 or CMASS3 entries).
NSM	Non Structural Mass by ID.
NSM1	Non Structural Mass (alternate form).
NSMADD	Non Structural Mass Set Combination.
NSML	Lumped Non Structural Mass by ID.
NSML1	Lumped Non Structural Mass (alternate form).

p-element Interface Elements

GMINTC	Defines a p-interface element along a curve.
GMINTS	Defines a p-interface element along a surface.
PINTC	Property definition for GMINTC.
PINTS	Property definition for GMINTS.

Rigid Elements

RBAR	Defines a rigid bar with six degrees-of-freedom at each end.
RBAR1	Alternative format for RBAR.
RBE1	Defines a rigid body connected to an arbitrary number of grid points.
RBE2	Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points.
RBE3	Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.
RJOINT	Defines a rigid joint element connecting two coinciding grid points.
RROD	Defines a pin-ended element that is rigid in translation.
RSPLINE	Defines multipoint constraints for the interpolation of displacements at grid points.
RSSCON	Defines multipoint constraints to model clamped connections of shell-to-solid elements.

- RTRPLT** Defines a rigid triangular plate.
RTRPLT1 Defines a rigid triangular plate (alternate).

Scalar and Bushing Elements

- CBUSH** Defines a generalized spring-and-damper structural element that may be nonlinear or frequency dependent.
- CBUSH1D** Defines the connectivity of a one-dimensional spring and viscous damper element.
- CBUSH2D** Defines the connectivity of a two-dimensional Linear-Nonlinear element.
- CELASi** Connection definition for scalar spring, also property definition for i=2 or 4.
- GENEL** Defines a general element.
- MGRSPR** Defines grids to add soft spring to ground. (SOL 600)
- PBUSH** Defines the nominal property values for a generalized spring-and-damper structural element.
- PBUSHT** Defines the frequency dependent properties or the stress dependent properties for a generalized spring and damper structural element.
- PBUSH1D** Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).
- PELAS** Specifies the stiffness, damping coefficient, and stress coefficient of a scalar elastic (spring) element (CELAS1 or CELAS3 entry).
- PELAST** Defines the frequency dependent properties for a PELAS Bulk Data entry.

Solid Elements

- CHEXA** Defines the connections of the six-sided solid element with eight to twenty grid points.
- CPENTA** Defines the connections of a five-sided solid element with six to fifteen grid points.
- CTETRA** Defines the connections of the four-sided solid element with four to ten grid points.
- PCOMPLS** Defines the linear/nonlinear properties of an n-ply composite material laminate for a layered solid (CHEXA) element in SOL 400 and all linear solution sequences between SOL101 and SOL112.
- PLSOLID** Defines a fully nonlinear (i.e., large strain and large rotation) hyperelastic solid element.
- PSLDN1** Defines the nonlinear property extensions for a PSOLID entry in SOL 400.
- PSOLID** Defines the properties of solid elements (CHEXA, CPENTA, and CTETRA entries).

Surface Elements

- CQUAD** Defines a plane strain quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis.
- CQUAD4** Defines an isoparametric membrane-bending or plane strain quadrilateral plate element.

CQUAD8	Defines a curved quadrilateral shell or plane strain element with eight grid points.
CQUADR	Defines an isoparametric membrane and bending quadrilateral plate element.
CSHEAR	Defines the properties of a shear panel (CSHEAR entry).
CTRIA3	Defines an isoparametric membrane-bending or plane strain triangular plate element.
CTRIA6	Defines a curved triangular shell element or plane strain with six grid points.
CTRIAR	Defines an isoparametric membrane-bending triangular plate element. However, this element does not include membrane-bending coupling. It is a companion to the CQUADR element.
PCOMP	Defines the properties of an n-ply composite material laminate.
PCOMPF	Defines the integration procedure for through the thickness integration of composite shells. (SOLs 400/600 only)
PCOMPG	Defines global (external) ply IDs and properties for a composite material laminate.
PLCOMP	Defines the linear/nonlinear properties of an n-ply composite material laminate of a plane stress, plane strain, or axisymmetric (CQUAD or CQUADX entry) element in SOL 400.
PLPLANE	Defines the properties of a fully nonlinear (i.e., large strain and large rotation) hyperelastic plane strain or axisymmetric element.
PSHELL	Defines the membrane, bending, transverse shear, and coupling properties of thin shell elements.
PSHLN1	Defines nonlinear property extensions for a PSHELL or PCOMP or PCOMPG entry in SOL 400.
PSHLN2	Defines nonlinear property extensions for a PLPLANE entry in SOL 400.
PSHEAR	Defines the properties of a shear panel (CSHEAR entry).
PSHEARN	Defines nonlinear property extensions for a PSHEAR in SOL 400.
SNORM	Defines a surface normal vector at a grid point for CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements.

Geometry

Adaptive Meshing

HADACRI	Specifies Mesh adaptivity criterion and corresponding parameters.
HADAPTL	Specifies local adaptive Mesh refinement control parameters.

Axisymmetry

AXIC	Defines the existence of an axisymmetric conical shell problem.
AXIF	Defines basic parameters and the existence of an axisymmetric fluid analysis.
AXSLOT	Defines the harmonic index and the default values for acoustic analysis entries.

FLSYM	Defines the relationship between the axisymmetric fluid and a structural boundary having symmetric constraints. The purpose is to allow fluid boundary matrices to conform to structural symmetry definitions.
POINTAX	Defines the location of a point on an axisymmetric shell ring at which loads may be applied via the FORCE or MOMENT entries and at which displacements may be requested. These points are not subject to constraints via MPCAX, SPCAX, or OMITAX entries.
RINGAX	Defines a ring for conical shell problem.
SECTAX	Defines a sector of a conical shell.

Coordinate Systems

BAROR	Defines default values for field 3 and fields 6 through 8 of the CBAR entry.
BEAMOR	Defines default values for field 3 and fields 6 through 8 of the CBEAM entry.
CORD1C	Cylindrical coordinate system definition.
CORD1R	Rectangular coordinate system definition.
CORD1S	Spherical coordinate system definition.
CORD3G	Defines a general coordinate system using three rotational angles as functions of coordinate values in the reference coordinate system.

Cyclic Symmetry

CYAX	Lists grid points that lie on the axis of symmetry in cyclic symmetry analysis.
CYJOIN	Defines the boundary points of a segment in cyclic symmetry problems.
NLCYSYM	Defines information to perform nonlinear cyclic symmetry analysis using SOL 600

Fluid Points

ACMODL	Defines modeling parameters for the Fluid-Structure Interface.
FREEPT	Defines the location of points on the surface of a fluid for recovery of surface displacements in a gravity field.
FSLIST	Defines the fluid points (RINGFL entry) that lie on a free surface boundary.
GRID	Defines fluid points in coupled fluid-structural analysis.
GRIDB	Grid point location on RINGFL.
GRIDF	Defines a scalar degree-of-freedom for harmonic analysis of a fluid.
GRIDS	Defines a scalar degree-of-freedom with a two-dimensional location. Used in defining pressure in slotted acoustic cavities.
PRESPT	Defines the location of pressure points in the fluid for recovery of pressure data.

- RINGFL** Defines a circle (fluid point) in an axisymmetric fluid model.
- SLBDY** Defines a list of slot points that lie on an interface between an axisymmetric fluid and a set of evenly spaced radial slots.

Grid Points

- AEGRID** Defines the location of an aerodynamic grid point.
- GRID** Defines the location of a geometric grid point, the directions of its displacement, and its permanent single-point constraints.
- GRIDB** Defines the location of a geometric grid point on a fluid point (RINGFL entry) for an axisymmetric fluid model and/or axisymmetric structure. Also defines the boundary of the fluid.
- GRDSET** Defines default options for fields 3, 7, 8, and 9 of all GRID entries.
- SEQGP** Used to manually order the grid points and scalar points of the problem. This entry is used to redefine the sequence of grid and scalar points to optimize bandwidth.

p-Element and p-Adaptivity Analysis

- FEEDGE** Defines a finite element edge and associates it with a curve.
- FEFACE** Defines geometric information that will be used in elements, surface definition, load definition, and boundary condition definition.
- GMBNDC** Defines a geometric boundary consisting of p-element edges along a curve interface. The boundary may consist of edges of shell, beam, or p-solid elements.
- GMBNDS** Defines a geometric boundary consisting of p-element faces along a surface interface. The boundary may consist of faces of p-solid or p-shell elements.
- GMCORD** Defines a convective/follower coordinate system on an FEEDGE, GMCURV, FEFACE, or GMSURF entry.
- GMCURV** Defines geometric curve that will be used in element geometry, load definition, and boundary condition definition.
- GMINTC** Defines curve interface elements to connect dissimilar meshes.
- GMINTS** Defines an interface element along a surface interface between boundaries of multiple subdomains.
- GMSURF** Defines geometric information that will be used in elements, surface definition, load definition, and boundary condition definition.
- PINTC** Defines properties for curve interface elements (see GMINTC).
- PINTS** Defines the properties for interface elements along surface interfaces between boundaries of multiple subdomains of p-elements.
- POINT** Define edge point for FEEDGE entry.

Scalar Points

- EPOINT** Defines extra points for use in dynamic problems.
- SEQGP** Grid and scalar point number resequencing.
- SPOINT** Defines scalar points.

Superelement Analysis

- CSUPER** Defines the grid or scalar point connections for identical or mirror image superelements or superelements from an external source. These are all known as secondary superelements.
- CSUPEXT** Assigns exterior points to a superelement.
- EXTRN** Defines a boundary connection for an external superelement.
- GRID** Defines interior points for a superelement.
- RELEASE** Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.
- SEBNDRY** Defines a list of grid points in a partitioned superelement for the automatic boundary search between a specified superelement or between all other superelements in the model.
- SEBULK** Defines superelement boundary search options and a repeated, mirrored, or collector superelement.
- SECONCT** Explicitly defines grid and scalar point connection procedures for a partitioned superelement.
- SEDLINK** Relates one design variable of a PART SE to one or more other design variables from other PART SEs.
- SEDRSP2** Defines equation responses that are used in the design, either as constraints or as an objective with quantities from multiple PART SEs.
- SEDRSP3** Defines constituents from multiple PART SE for an external response using user-supplied routine(s).
- SEELT** Reassigns superelement boundary elements to an upstream superelement.
- SEEXCLD** Defines grid points that will be excluded during the attachment of a partitioned superelement.
- SELABEL** Defines a label or name to be printed in the superelement output headings.
- SELOC** Defines a partitioned superelement relocation by listing three noncolinear points in the superelement and three corresponding points not belonging to the superelement.
- SEMPLN** Defines a mirror plane for mirroring a partitioned superelement.
- SEQSEP** Used with the CSUPER entry to define the correspondence of the exterior grid points between an identical or mirror-image superelement and its primary superelement.
- SESET** Defines interior grid points for a superelement.
- SETREE** Specifies superelement reduction order.

Module Analysis

EXCLUDE	Specifies Bulk Data entries in the primary Module to be ignored in the secondary (or copied) Module.
MDBCNCT	Defines the Touching and Touched Contact Bodies in Different Modules.
MDBCTB1	Defines a Contact Table for Bodies in Different Modules.
MDBNDRY	Module to Module Boundary Point Definitions.
MDBOLT	Defines the Multi-Point Constraints for a Bolt Between Two Modules.
MDBULK	Module Type Definitions.
MDCONCT	Module Boundary Point Connections.
MDDMIG	Direct Matrix Input at Points Defined in Two or More Modules.
MDEXCLD	Module to Module Excluded Boundary Point Definitions.
MDFAST	A Shell Patch Fastener Connection Between Two Modules.
MDLABEL	Module Output Label.
MDLOC	Module Reposition by Translation and/or Rotation.
MDMIR1	Defines a Module mirror by three non-collinear points on the mirror plane.
MDMIR2	Defines a Module mirror by specifying a pair of coordinate system axes on the mirror plane.
MDMOVE	Defines a Module repositioning sequence. References MDMIRi, MDROTi, and MDTRAN entries.
MDMPC	Multipoint Constraint Between Two or More Modules.
MDMPLN	Module Reposition by Mirroring.
MDRBE3	Rigid Body Element Between Two or More Modules, Form 2.
MDRBE3	Interpolation Constraint Element Between Two or More Modules.
MDRJNT	Rigid Joint Between Two Modules.
MDROT1	Defines a Module rotation by specifying a rotation vector and reference point.
MDROT2	Defines a Module rotation by specifying a coordinate system axis for the rotation vector.
MDRROD	Rigid Pin-Ended Element Connection Between Two Modules.
MDSEAM	A Shell Patch SEAM Connection Between Two Modules.
MDTRAN	Defines a Module translation by specifying a vector.
MDWELD	Weld or Fastener Element Connection Between Two Modules.

Loads

Dynamic Loads

ACSRCE	Defines acoustic source as a function of power vs. frequency.
DAREA	Defines scale (area) factors for static and dynamic loads. In dynamic analysis, DAREA is used in conjunction with RLOADi and TLOADi entries.
DELAY	Defines the time delay term τ in the equations of the dynamic loading function.
DLOAD	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.
DPHASE	Defines the phase lead term θ in the equation of the dynamic loading function.
LOADCYH	Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.
LOADCYN	Defines a physical static or dynamic load for use in cyclic symmetry analysis.
LSEQ	Defines a sequence of static load sets.
NOLIN1	Nonlinear transient load definition.
NLRGAP	Defines a nonlinear transient radial (circular) gap.
RBE3U	Defines methods to distribute applied loads to a surface. (SOL 600)
RLOAD1	Frequency dependent excitation definition.
TABLED1	Tabular functions for generating dynamic loads.
TLOAD1	Time dependent excitation definition.

Heat Transfer Loads

CONV	Specifies a free convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).
CONVM	Specifies a forced convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).
PCONV	Specifies the free convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.
PCONVM	Specifies the forced convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.
QBDY1	Defines a uniform heat flux into CHBDYj elements.
QBDY2	Defines grid point heat flux into CHBDYj elements.
QBDY3	Defines a uniform heat flux load for a boundary surface.
QHBDY	Defines a uniform heat flux into a set of grid points.
QVECT	Defines thermal vector flux from a distant source into a face of one or more CHBDYi boundary condition surface elements.
QVOL	Defines a rate of volumetric heat addition in a conduction element.
RADBC	Specifies an CHBDYi element face for application of radiation boundary conditions.

RADBND	Specifies Planck's second radiation constant and the wavelength breakpoints used for radiation exchange problems.
RADCAV	Identifies the characteristics of each radiant enclosure.
RADLST	Identifies the individual CHBDYi surface elements that comprise the entire radiation enclosure.
RADMTX	Provides the $F_{ji} = A_j f_{ji}$ exchange factors for all the faces of a radiation enclosure specified in the corresponding RADLST entry.
RADSET	Specifies which radiation cavities are to be included for radiation enclosure analysis.
SLOAD	Defines concentrated static loads on scalar or grid points.
SLOADN1	Describes TOP/BOT/MID Scalar Load for Heat Shell Element in SOL 400
TEMP	Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPB3	Defines a temperature field for the three-node beam element (CBEAM3 entry).
TEMPBC	Defines the temperature boundary conditions for heat transfer analysis. Applies to steady-state and transient conditions.
TEMPD	Specifies default initial temperature at grid points.
TEMPN1	TOP/BOT/MID Grid Point Temperature Field for Heat Shell Element in SOL 400
VIEW	Defines radiation cavity and shadowing for radiation view factor calculations.
VIEW3D	Defines parameters to control and/or request the Gaussian Integration method of view factor calculation for a specified cavity.

p-element Loads

GMBC	Defines enforced displacements for GRID, FEEDGE, GMCURV, FEFACE, and GMSURF entries.
GMCONV	Defines convection boundary conditions.
GMLOAD	Defines the forces and moments to be applied to a FEEDGE, GMCURV, FEFACE, or GMSURF entry.
GMQVOL	Defines volumetric heat loads.
TEMPF	Defines the thermal loading to be applied to a group of elements.

Static Loads

ACCEL	Defines static acceleration loads, which may vary over a region of the structural model. The load variation is based upon the tabular input defined on this Bulk Data entry.
ACCEL1	Defines static acceleration loads at individual GRID points.
CLOAD	Defines a static load as a linear combination of previously calculated superelement loads defined by the LSEQ entry in nonlinear static analysis (SOL 106 or 153).

DEFORM	Defines enforced axial deformation for one-dimensional elements for use in statics problems.
FORCE	Defines a static concentrated force at a grid point by specifying a vector.
FORCE1	Defines a static concentrated force at a grid point by specification of a magnitude and two grid points that determine the direction.
FORCE2	Defines a static concentrated force at a grid point by specification of a magnitude and four grid points that determine the direction.
FORCEAX	Defines a concentrated force on a conical shell ring.
FORCEi	Defines concentrated load at grid point.
GRAV	Defines acceleration vectors for gravity or other acceleration loading.
GRIDA	Defines an associative GRID point to be used in the COMBINE step of PAA.
LOAD	Defines a static load as a linear combination of load sets defined via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADB3, PLOADX1, SLOAD, RFORCE, and GRAV, ACCEL and ACCEL1 entries.
LOADCLID	Defines a loading combination in PAA using the Load ID from SUBCASEs of the Parts
LOADCNAM	Used only in PAA to define a loading combination using the LOADNAMEs used in Case Control for Parts
LOADCSUB	Defines a loading combination in PAA using the SUBCASE IDs from the Parts
LOADCYH	Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.
LOADCYN	Defines a physical static or dynamic load for use in cyclic symmetry analysis.
LOADCYT	Specifies loads as a function of azimuth angle by references to tables that define scale factors of loads versus azimuth angles. This entry is used only when STYPE = "AXI" on the CYSYM entry.
LOADOF	Specifies table IDs for individual degrees of freedom for the static loads with tables described using LOADT entries for SOL 600 only.
LOADT	Specifies static loads that will use a table to describe their variation with pseudo-time.
LSEQ	Defines a sequence of static load sets.
MOMAX	Defines a static concentrated moment load on a ring of a conical shell.
MOMENT	Defines a static concentrated moment at a grid point by specifying a scale factor and a vector that determines the direction.
MOMENTi	Defines moment at grid point.
PLOAD	Defines a uniform static pressure load on a triangular or quadrilateral surface comprised of surface elements and/or the faces of solid elements.
PLOAD1	Defines concentrated, uniformly distributed, or linearly distributed applied loads to the CBAR or CBEAM elements at user-chosen points along the axis. For the CBEND element, only distributed loads over an entire length may be defined.

PLOAD2	Defines a uniform static pressure load applied to CQUAD4, CSHEAR, or CTRIA3 two-dimensional elements.
PLOAD4	Defines a pressure load on a face of a CHEXA, CPENTA, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element.
PLOADB3	Defines a distributed load to a CBEAM3 element over entire length of the beam axis.
PLOADX1	Defines surface traction to be used with the CQUADX, CTRIAX, and CTRIAX6 axisymmetric element.
PRESAX	Defines the static pressure loading on a conical shell element.
RBE3U	Defines methods to distribute applied loads to a surface. (SOL 600)
RFORCE	Defines a static loading condition due to an angular velocity and/or acceleration.
SLOAD	Defines concentrated static loads on scalar or grid points.
SPCD	Defines an enforced displacement value for static analysis and an enforced motion value (displacement, velocity or acceleration) in dynamic analysis.
SPCR	Defines an enforced relative displacement value for a load step in SOL 400.
TEMP	Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPD	Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.
TEMPP1	Defines temperature field for surface elements.
TEMPRB	Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPAX	Defines temperature sets for conical shell problems.

Materials

Anisotropic

MAT2	Defines the material properties for linear anisotropic materials for two-dimensional elements.
MAT3	Defines the material properties for linear orthotropic materials used by the CTRIAX6 element entry. It also is allowed with orthotropic materials on the PSHLN2 and PLCOMP entries.
MAT5	Defines the thermal material properties for anisotropic materials.
MAT8	Defines the material property for an orthotropic material for isoparametric shell elements.

MAT9	Defines the material properties for linear, temperature-independent, anisotropic materials for solid isoparametric elements (see PSOLID entry description).
MATORT	Define elastic 3D Orthotropic Material (SOL 400, 600. Also all linear solutions between SOL101 and 112 in conjunction with PCOMPLS, or CPYRAM).
Fatigue	
MATFTG	Defines cyclic material properties (S-N + ε -N) for use in fatigue analysis.
Fluid	
AXIF	Includes default values for mass density and bulk modulus.
AXSLOT	Includes default values for mass density and bulk modulus.
CFLUIDi	Includes mass density and bulk modulus.
FSLIST	Includes mass density at free surface.
MAT10	Defines material properties for fluid elements in coupled fluid-structural analysis.
MFLUID	Defines the properties of an incompressible fluid volume for the purpose of generating a virtual mass matrix.
SLBDY	Includes mass density at interface between fluid and radial slots.
Isotropic	
MAT1	Defines the material properties for linear isotropic materials.
MAT4	Defines the constant or temperature-dependent thermal material properties for conductivity, heat capacity, density, dynamic viscosity, heat generation, reference enthalpy, and latent heat associated with a single-phase change.
MATHP	Specifies material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers).
RADM	Defines the radiation properties of a boundary element for heat transfer analysis.
Stress Dependent	
CREEP	Defines creep characteristics based on experimental data or known empirical creep law.
MATS1	Specifies stress-dependent material properties for use in applications involving nonlinear materials.
NLMOPTS	Defines nonlinear material option control for SOL 400.
TABLES1	Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).

Temperature Dependent

MATTi	Table references for temperature-dependent MATi materials.
RADMT	Specifies table references for temperature dependent RADM entry radiation boundary properties.
TABLEM1	Tabular functions for generating temperature-dependent material properties.
TABLEST	Table references for temperature dependent MATS1 materials.
TEMP	Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPAX	Defines temperature sets for conical shell problems.
TEMPD	Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.
TEMPP1	Defines temperature field for surface elements.
TEMPRB	Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery.

Miscellaneous

Brake Squeal (SOL 400)

BSQUEAL	Specifies data for brake squeal analysis using Implicit Nonlinear (SOL 400).
---------	--

Comments

\$	Used to insert comments into the input file. Comment statements may appear anywhere within the input file.
----	--

Delete

/	Control Input Stream
---	----------------------

Direct Matrix Input

CONM1	Defines a 6x6 mass matrix at a geometric grid point.
DMI	Defines matrix data blocks.
DMIG	Defines direct input matrices related to grid, extra, and/or scalar points.
DMIG,UACCEL	Defines rigid body accelerations in the basic coordinate system.
DIMIAX	Defines axisymmetric (fluid or structure) related direct input matrix terms.
TF	Defines a dynamic transfer function.

Direct Matrix Input for Aeroelasticity

- DMIJ** Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for CAERO1, CAERO3, CAERO4 and CAERO5 and for the slender body elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries.
- DMIJI** Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for the interference elements of CAERO2.
- DMIK** Defines direct input matrices related to physical (displacement) degrees-of-freedom (ks-set) of aerodynamic grid points.

End of Input

- ENDDATA** Designates the end of the Bulk Data Section.

Include File

- INCLUDE** Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.
- VCCT** Virtual Crack Closure Technique - SOL 400, SOL 600
- LORENZI** Contour Integral Approach for stress intensity calculation - SOL 600

Materials (SOLs 400/600)

- COHESIV** Defines data for cohesive materials. (SOL 600)
- MATDIGI** e-XstreamDigimat material interface (SOL 400)
- MATEP** Elasto-plastic material properties.
- MATF** Specifies material failure model. Also maybe used in all linear solutions between SOL101 and 112 in conjunction with PCOMPLS.)
- MATG** Gasket material properties.
- MATHE** Hyperelastic material properties.
- MATHED** Damage model properties for hyperelastic materials. (SOL 600)
- MATNLE** The MATNLEX entries specify advanced forms of nonlinear elastic materials. (SOL 600)
- MATORT** Elastic 3D orthotropic material properties. Also maybe used in all linear solutions between SOL101 and 112 in conjunction with PCOMPLS, or CPYRAM.)
- MATTEP** Thermoelastic-Plastic material properties.
- MATTF** Material Failure Model Temperature Variation
- MATTG** Temperature variation of interlaminar materials.
- MATTHE** Thermo hyperelastic material.
- MATTORT** Thermoelastic orthotropic material

MATS3	Specifies NLELAST option for advanced orthotropic, nonlinear elastic materials at axisymmetric conditions. SOL 400 only.
MATS8	Specifies NLELAST option for advanced orthotropic, nonlinear elastic material for plane stress and shell elements SOL 400 only.
MATSMA	Material properties for shape memory alloys.
MATSORT	Specifies NLELAST option for advanced 3D orthotropic, nonlinear elastic materials. SOL 400)
MATTVE	Thermo-visco-elastic material properties
MATVE	Viscoelastic material properties
MATVP	Viscoplastic or creep material properties
MCOHE	Cohesive materials (SOL 400)
MDELAM	Defines materials for which delamination may occur.

Non-Structural Mass Distribution Selection

NSM	Non Structural Mass entry by property or element ID, value.
NSM1	Non Structural Mass entry by property or element ID, value.
NSMADD	Non Structural Mass as sum of listed sets.
NSML	Lumped non structural mass entry by property or element ID, value.
NSML1	Lumped non structural mass entry by property or element ID, value.

OpenFSI (SOL 400)

FSICTRL	Defines the analysis type for OpenFSI fluid structure analysis simulations.
WETELME	Defines a wetted element used in OpenFSI fluid structure interaction simulations.
WETELMG	Defines a wetted element used in OpenFSI fluid structure interaction simulations.
WETLOAD	Defines the load set and associated wetted surface for OpenFSI fluid structure interaction simulations.
WETSURF	Defines a wetted surface used in OpenFSI fluid structure interaction simulations.

Output Control

BOUTPUT	Defines slave nodes at which output is requested.
CBARAO	Defines a series of points along the axis of a bar element (CBAR entry) for stress and force recovery output.
ECHOOFF	Marks the point or points in the input file to deactivate printed echo of the Bulk Data.
ECHOON	Marks the point or points in the input file to activate printed echo of the Bulk Data.
FREEPT	Surface point location for data recovery in hydroelastic problems.
MARPRN	Defines “print” options for SOL 600.

MLAYOUT	Selects layered composite shell output to be placed in Marc's t16 and/or t19 files and (if requested) to be transferred from Marc to the Nastran Database. (SOL 600)
PLOTEL	Defines a one-dimensional dummy element for use in plotting.
POINTAX	Defines the location of a point on an axisymmetric shell ring at which loads may be applied via the FORCE or MOMENT entries and at which displacements may be requested.
PRESPT	Defines the location of pressure points in the fluid for recovery of pressure data in hydroelastic problems.
SET1	Defines a set of grid points.
TSTEP	Specifies time step intervals for data recovery in transient response.
TSTEPNL	Specifies time step intervals for data recovery in nonlinear transient response.

Monitor Points

AECOMP	Defines a component for use in aeroelastic monitor point definition or external splines.
AECOMPL	Defines a component for use in aeroelastic monitor point definition or external splines as a union of other components.
MONPNT1	Defines an integrated load monitor point at a point (x,y,z) in a user defined coordinate system.
MONPNT2	Element Monitor Output Results Item.
MONPNT3	Sums select Grid Point Forces to a user chosen monitor point.
MONCNC	Stripwise aerodynamic lift and pitching moment coefficients.
M	
MONSUM	Linear combination of monitor point components.
MONSUM1	Linear combination of two or more monitor points.
MONSUMT	Linear combination of two or more monitor points with moment transfer.

p-element Output Control

OUTPUT	Output control for p-adaptive analysis.
OUTRCV	Defines options for the output of displacements, stresses, and strains of p-elements.

Parameters

CAMPBLL	Specifies the parameters for Campbell diagram generation.
PARAM	Specifies values for parameters used in solution sequences or user-written DMAP programs.
MDLPRM	Specifies parameters which affect the solution of the structural model.

Solution Control

ITER	Defines options for the iterative solver in SOLs 101, 106, 108, 111 and 153.
RVDOF	Degrees-of-freedom specification for residual vector computations.
RVDOF1	Degrees-of-freedom specification for residual vector computations (alternate form).

Tabular Input

DTI	Defines table data blocks.
DTI,ESTDATA	Provides override data for time and space estimation for superelement processing operations.
DTI,INDTA	Specifies or overrides default item codes for the sorting and filtering of element stresses, strains, and forces.
DTI,SETREE	Defines a superelement tree that determines the superelement processing order.
DTI,SPECSEL	Correlates spectra lines specified on TABLED1 entries with damping values.
DTI,SPSEL	Correlates output requests with frequency and damping ranges.
DTI,UNITS	Defines units necessary for conversion during the analysis for the Nastran/ADAMS interface and Nastran fatigue analysis.
TABDMP1	Defines modal damping as a tabular function of natural frequency.
TABLED1	Tabular functions for generating dynamic loads.
TABLEL1	Specifies a table of amplitude vs pseudo-time for static loads specified in LDTABL entries SOL 600 only.
TABLEM1	Tabular functions for generating temperature-dependent material properties.
TABLES1	Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).
TABL3D	Specifies a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc. (SOLs 400/600)
TABLE3D	Specify a function of three variables for the GMBC, GMLOAD, and TEMPF entries only.
TABLFTG	Defines cyclic load variation for use in fatigue analysis.
TABLRPC	Tabular functions for generating dynamic loads by reading the tabular data from an external channel data file.
TABRND1	Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.

3D Contact Region

BCBMRAD	Allows the equivalent radius in beam-to-beam contact to be different for each beam cross section. (SOLs 400/600)
BCBOX	Defines a 3D contact region.
BCHANGE	Changes definitions of contact bodies.
BCMATL	Defines a 3D contact region by element material.
BCMOVE	Defines movement of bodies in contact.
BCPARA	Defines contact parameters.
BCPROP	Defines a 3D contact region by element properties.
BSURF	Defines a contact body or surface by element IDs.
GMNURB	3D contact region made up of NURBS.

Old Contact Format

BCTABLE	Defines a contact table.
BCBODY	Defines a flexible rigid contact body in 2D or 3D.

New Contact Format

BCTABL1	Defines a contact table.
BCONNECT	Defines the Touching and Touched Contact Bodies
BCONPRG	Geometric Contact Parameters of Touching Bodies
BCONPRP	Physical Contact Parameters of Touching Bodies in SOL 101 and 400
BCBODY1	Flexible or Rigid Contact Body in 2D and 3D
BCBDPRP	Contact Body Parameters in SOLs 101 and 400
BCRIGID	Defines a Rigid Contact Body in SOLs 101 and 400
BCRGSRF	Rigid Contact Surface List in SOLs 101 and 400
BCPATCH	Defines a Rigid Contact Body Made up of Quadrilateral Patches in SOLs 101 and 400
BCBZIER	Defines a Rigid Contact Body Made up of Bezier Surfaces in SOLs 101 and 400
BCNURB2	Defines a 2D Rigid Contact Body Made up of NURBS in SOLs 101 and 400
BCNURBS	Defines a Rigid Contact Body Made up of NURBS in SOLs 101 and 400
BCTRIM	Defines the Geometry of a Trimming Curve

User-Defined Subroutines and Services

BCONUDS	Allows the user to provide contact routines for use with enhanced MSC Nastran contact analysis. (SOL 600 and SOL 400)
GENUDS	User Data for Notify User Defined Service or Subroutine

MATUDS	Allows the user to provide material routines for use with enhanced MSC Nastran material models. (SOL 600 and SOL 400)
MATTUSR	Specifies table variation of user defined generic materials in SOL 600 and SOL 400 only.
MATUSR	Specifies user-defined, generic material properties for hypoelastic material models in SOL 600 and user defined material models in SOL 400 only.
UDSESV	Define the number and names of user state variables for material user subroutines (SOL 400)

Nastran Implicit Nonlinear (SOL 600)

3D Contact Region (SOL 600)

BCBMRAD	Allows the equivalent radius in beam-to-beam contact to be different for each beam cross section. (SOL 600)
SANGLE	Defines automatic analytical contact threshold angle for multiple subcases.
UNGLUE	Defines grids that should be eliminated from glued contact for SOL 600 and SOL 400.
WEAR	Specifies values for modeling mechanical wear in deformable contact bodies for SOL 600 only.

Analysis Termination Options (SOL 600)

TERMIN	Control to terminate a SOL 600 analysis under certain conditions
--------	--

Bolts (SOL 600)

MBOLT	Defines a bolt for use in countries outside the USA.
MBOLTUS	Defines a bolt for use in the USA and all other countries.

Brake Squeal (SOL 600)

BRKSQ	Specifies data for brake squeal calculations using SOL 600.
-------	---

Composite Integration Options (SOL 600)

MIXTURE	Defines constituents of “composite” material on original and potentially damaged state.
PCOMPF	Defines the integration procedure for through the thickness integration of composite shells.

Creep Analysis (SOL 600)

- MACREEP** Controls a transient creep analysis.
- MPCREEP** Specifies input values for Marc's creep parameter when creep analysis is performed using SOL 600.
- MTCREEP** Controls a transient thermal creep analysis. This entry or the MACREEP entry is required if ITYPE is not zero on the MPCREEP entry.

Element Birth and Death (SOL 600)

- ACTIVAT** This entry allows the user to re-activate certain elements that were previously deactivated in a previous subcase.
- DEACTEL** This entry allows the user to deactivate elements that have failed or are no longer necessary in a particular subcase.

Elements (SOL 600)

- ALIASM** Allows selected elements which normally use a default formulation to be aliased to a different formulation.
- CSSHL** Defines a connection for a Solid Shell with 6 or 8 grid points.
- CSSHLLH** Defines conversion of CHEXA elements to Solid Shell elements.
- CSSHLM** Defines conversion of CHEXA or CPENTA elements described by material ID to Solid Shell elements.
- CSSHLP** Defines conversion of CPENTA elements to Solid Shell elements.
- MISLAND** Defines an island of connected elements that will be completely removed if the number of elements within the island becomes smaller than a specified value.

Element Properties (SOL 600)

- NTHICK** Defines nodal thickness values for beams, plates, and/or shells.
- PSSHL** Defines the properties for Solid Shell (CSSHL) elements.

Fatigue, Fracture and Crack Propagation (SOL 600)

- LORENZI** This option gives an estimation of the J-Integral for a crack configuration using the domain integration method.
- VCCT** Virtual crack closure technique

General Tables (SOL 600)

- MTABRV** Defines a list of tables to reverse positive and negative values and/or add points at the lower and upper end of tables.

TABD1MD	Defines how TABLED1 entries are internally modified in SOL 600.
TABL3D	Specifies a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc.

Heat Transfer (SOL 600)

MPHEAT	Maps to Marc's HEAT parameter for SOL 600 heat transfer analysis.
MTHERM	Iteration control for automatic thermal loading for structural analysis following a heat transfer analysis.
NLHEATC	Defines numerical analysis parameters for SOL 600 heat transfer analysis

Inertia Relief (SOL 600)

SUPORT6	Inertia relief used in Nastran Implicit Nonlinear (SOL 600 only).
-------------------------	---

Initial Conditions (SOL 600)

IPSTRN	Defines initial plastic strain values.
ISTRESS	Defines initial stress values.

Input/Output (SOL 600)

MARCON	Inserts a text string in Marc.
MARCOOUT	Selects data recovery output.

Matrix Input/Output (SOL 600)

DMIGOUT	Defines DMIG matrices to be output from the Marc Portion of SOL 600.
DMIGROT	Defines large rotation and other characteristics of a matrix entered using DMIG..
MDMIAUX	Specifies the DOMAINSOLVER command to be used in conjunction with secondary spawned jobs when MDMIOUT is used. SOL 600 only.
MDMIOUT	Defines full or reduced stiffness and mass matrices to be output from the Marc portion of SOL 600.
MESUPER	Defines external superelement DMIG input for SOL 600 residual analyses
MNF600	Defines auxiliary data for MSC Adams MNF files.

Solid Composites (SOL 600)

MSTACK	Defines the direction in which 3D solid composites are stacked.
------------------------	---

Solution Control (SOL 600)

MPROCS	Provides additional control for parallel processing.
NLAUTO	Parameters for automatic load/time stepping.
NLHEATC	Defines Numerical Analysis Parameters for Heat Transfer Analysis.
NLSTRAT	Strategy parameters for nonlinear structural analysis.
PARAMARC	Parallel domain decomposition.
RESTART	Restart data.

Structural Analysis Following a Heat Transfer Analysis (SOL 600)

MINSTAT	This option is used to enter initial (stress free) temperatures calculated from a previous heat transfer analysis and saved on a t16 or t19 file.
-------------------------	---

t16 Output Control (SOL 600)

MT16SEL	Limits elements and/or grid results to selected elements or grids for t16 and t19 file results.
MT16SPL	Determines how to split a Marc t16 file into one or more smaller t16 files.

User Subroutine Control (SOL 600)

USRSUB6	Defines user subroutines used in SOL 600 only.
-------------------------	--

Explicit Nonlinear (SOL 700)**Air Bags (SOL 700)**

GBAG	Defines the pressure within an enclosed volume.
----------------------	---