User Manual

for the

MYSTRAN General Purpose Finite Element Structural Analysis Computer Program

(Open Source Version)

Dr. Bill Case – Original Author

Subsequent User Manual Authors:

Brian Esp

Zach Lerner

Victor Kemp

Steve Doyle

Bruno Paschoalinoto

www.mystran.com

(September 2025)

Most consistent with MYSTRAN program version 17.0

1 INTRODUCTION

MYSTRAN is a general-purpose finite element analysis computer program for structures that can be modeled as linear (i.e. displacements, forces and stresses proportional to applied load). MYSTRAN is an acronym for "My Structural Analysis", to indicate its usefulness in solving a wide variety of finite element analysis problems on a personal computer (although there is no reason that it could not be used on mainframe computers as well). For anyone familiar with the popular NASTRAN computer program developed by NASA (National Aeronautics and Space Administration) in the 1970's and popularized in several commercial versions since, the input to MYSTRAN will look quite familiar. Indeed, many structural analyses modeled for execution in NASTRAN will execute in MYSTRAN with little, or no, modification. MYSTRAN, however, is *not* NASTRAN. All of the finite element processing to obtain the global stiffness matrix (including the finite element matrix generation routines themselves), the reduction of the stiffness matrix to the solution set, as well as all of the input/output routines are written in independent, modern, Fortran 90/95 code. The major solution algorithms (e.g., triangular decomposition of matrices and forward/backward substitution to obtain solutions of linear equations) as well as the Givens method of eigenvalue extraction, however, were obtained from the popular LAPACK code, Reference 1, available to the general public on the World Wide Web. The code for the Lanczos method of eigenvalue extraction, Reference 2, was obtained from the ARPACK library, also available to the general public on the World Wide Web. The code for the grid point sequencing algorithm (used to ensure a minimum bandwidth for the stiffness matrix) was obtained from the author of Reference 3.

As of Version 11.3, MYSTRAN has available the sparse solver SuperLU (see Reference 13). This solver is currently only used in statics solutions (SOL 1) and is the default method used for matrix decomposition and equation solution (Forward-Backward Solution, or FBS).

There is no inherent limitation to problem size, or number of degrees of freedom, for MYSTRAN. Rather, the users' personal computer memory (RAM and disk) limitations will dictate what size problems can be effectively solved using MYSTRAN on their computer.

Major features of the program are:

- NASTRAN style input. NASTRAN model files will run in MYSTRAN with little or no modification for static and eigenvalue analyses.
- 3D structures with arbitrary geometry.
    - Linear static analysis.
- Eigenvalue analysis via Lanczos, Givens and modified Givens methods. In addition, for the fundamental mode there is also an Inverse Power method.

    - Optional calculation of modal mass and/or modal participation factors (Reference 8)
- Craig-Bampton model generation.
- Interface to the popular FEMAP pre/post processor program.
- Grid points (3 translations and 3 rotations per grid) that define the finite element model mesh:
    - Locations can be defined in rectangular, cylindrical or spherical coordinate systems that can be different for each grid.
    - Global stiffness matrix can be formulated in rectangular, cylindrical or spherical coordinate systems that can be different for each grid.
- Scalar points (SPOINT') that have no defined geometry (one degree of freedom)
- A finite element library consisting of the following elastic and rigid elements.
    - 1D and scalar elements.
        - BAR element with two grids and stiffness for up to six degrees of freedom per grid (axial, two planes of bending, torsion) for beams that have their shear center and elastic axis coincident.
        - BUSH element (spring connecting two grids).
        - ELAS1,2,3,4 elements (scalar spring connecting two degrees of freedom).
        - ROD element (axial load and torsion element connected to two grid points)

- Triangular and quadrilateral plate elements for thick (Mindlin plate theory) *and* thin (Kirchoff plate theory) plates. The plates can include membrane and/or bending stiffness and can be either single or multi-ply composite laminates. The following is a brief overview and Section 3.2 discusses the options in more detail:
    - TRIA3: Flat triangular plate element with separate membrane and bending stiffness, as well as transverse shear flexibility, based on Mindlin thick plate theory (Reference 4). It can be either a single or multi-ply composite.
    - TRIA3K: Flat triangular plate element with separate membrane and bending stiffness based on Kirchoff thin plate theory (Reference 6)
    - QUAD4 with MIN4: Mindlin quadrilateral plate element with separate membrane and bending stiffness, as well as transverse shear flexibility, based on Mindlin thick plate theory (Reference 5). This is essentially a flat element, however small distortion out of plane is accommodated. It can be either a single or multi-ply composite. This is the default for the QUAD4TYP parameter, equivalent to “PARAM, QUAD4TYP, MIN4”.
    - QUAD4 with MITC4+: Quadrilateral plate element with separate membrane and bending stiffness, as well as transverse shear flexibility. (Reference 15). This is the alternative option for the QUAD4TYP param, equivalent to “PARAM, QUAD4TYP, MITC4+”.
    - QUAD4K: Quadrilateral plate element with separate membrane and bending stiffness based on Kirchoff thin plate theory (Reference 7). This is essentially a flat element, however small distortion out of plane is accommodated.
    - QUAD8 MITC8: Quadrilateral plate element with combined membrane and bending stiffness, as well as separate transverse shear flexibility. (Reference 16). It can be curved out of plane but is limited to statics with isotropic material and no body loads.
    - SHEAR element that carries in-plane shear stresses.
- 3D solid elements
    - TETRA 4 and 10 node solid elements. See Reference 10.
    - PENTA 6 and 15 node elements with selective substitution reduction for shear (if desired). See Reference 10.
    - HEXA 8 and 20 node elements with selective substitution reduction for shear (if desired). See Reference 10.
- R-elements:

        - RBE2 rigid element specifying a relationship for one or more degrees of freedom (DOF's) of one or more grids being rigidly dependent on the DOF's of another grid.
        - RBE3 element for distributing loads or mass from one grid to other grids.
        - RSPLINE element for interpolating displacements between elements.
    - User defined elements:
        - CUSERIN element where the user inputs the stiffness and mass matrices and specifies the connection of the element to defined grids and scalar points.
- Single point constraints (SPC's) wherein some degrees of freedom are grounded (e.g. for specifying boundary conditions).
- Other SPC's wherein specified degrees of freedom have a specified motion (enforced displacements).
- Multi point constraints (MPC's), wherein specified degrees of freedom are linearly dependent on other degrees of freedom.
- Loads on the finite element model via:
    - Forces and/or moments applied directly to grid points.
    - Pressure loading on plate element surfaces.
    - Gravity loads on the whole model (in conjunction with mass defined by the user).
    - Equivalent loads due to thermal expansion.
    - Equivalent loads due to enforced displacements.
    - Inertia Loads due to rigid body angular velocity and acceleration about some specified grid (RFORCE).
- Loads on scalar SPOINT's (via SLOAD)
    - Linear isotropic, orthotropic and anisotropic material properties.
- Mass defined via:
    - Density on material entries.
    - Mass per unit length, or per unit area, for finite elements.
    - Concentrated masses at grids (CONM2) with possible offsets and moments of inertia.
    - Scalar masses (CMASS1,2,3,4).
- Multiple subcases to allow for solution for more than one loading condition in one execution.
- Output of:
    - Displacements (six degrees of freedom per grid) for any defined set of grids desired.

    - Applied loads for any defined set of grids.
    - Single point forces of constraint for any defined set of grids.
    - Multi point forces of constraint for any defined set of grids (includes forces of constraint due to MPC's as well as rigid elements).
    - Grid point force balance for any defined set of grids.
    - Element engineering and/or nodal forces for any defined set of elements.
    - Element stresses for any defined set of elements.
    - Element strains for 2D and 3D elements (including ply strains in composite elements).
    - Effective modal mass and/or modal participation factors in eigenvalue analyses.
    - Output transformation matrices (OTM's) in Craig-Bampton analyses for displacement, acceleration, force, and stress quantities.
- Interface to FEMAP post processing program for display of model and results (see Bulk Data entry PARAM with parameter name POST).
- Guyan reduction to statically reduce the stiffness and mass matrices. This is needed if the Givens method of eigenvalue analyses is used to remove degrees of freedom that have no mass (however, LANCZOS is the preferred method of eigenvalue extraction).
- Limited CHKPNT/RESTART feature that allows a previous job to be restarted to obtain new or different outputs (displacements, etc.). The finite element model and solution (SOL in Exec Control) must remain the same.
- General:
    - AUTOSPC (automatic SPC generation based on used control).
    - Stiffness matrix equilibrium checks on request (Bulk Data PARAM entry EQCHECK)
    - Automatic grid point resequencing to reduce matrix bandwidth (Bulk Data PARAM entry GRIDSEQ with value BANDIT – default).

1.1 MYSTRAN Brief History

Dr. Bill Case spent his career at NASA, frequently using the Nastran program in the 1970's and 1908's. Later, he and a colleague created their own version of Nastran, called MYSTRAN, based on modern Fortran. This effort is suspected to have started in the 1990's. The MYSTRAN program was originally a commercial program.

In the mid 2010's, Bill met Dr. Brian Esp and began discussing the MYSTRAN project. In 2019, Bill offered the source code to Brian and requested that he carry on the MYSTRAN project.

The source code was eventually open-sourced and placed on GitHub with a permissive MIT license. By making the program open-source, other members joined the MYSTRAN project. Along with Bill, these members made significant contributions. Bill passed away in 2021, but all of the pieces were in place to allow the team to make progress towards creating a complete Nastran-like solver for linear static and Eigen problems (buckling and natural frequency). The following is a partial list of accomplishments since the program became open source.

- Compling processes (Various)
- Sparse solver SuperLU added (*Bill Case*)
- OP2 support added (*Steven Doyle*)
- Fixed RBE3 (*Bruno Paschoalinoto*)
- Fixed BUSH elements (Various)
- Fixed many memory bugs (*Bruno Paschoalinoto*)
- Upgraded documentation (Various)
- Created F06CSV tool – supports validation effort (*Bruno Paschoalinoto*)
- Shell bucking added (*Victor Kemp*)
- Shell elements: Fixes to existing and added MITC4+ 4-node quad (*Victor Kemp*)
- Bug fixes (Various)
- Support and oversight of all previously listed items (*Zach Lerner*)
- Project Oversight (*Brian Esp*)

In addition, Brian Esp and Zach Lerner provided financial support to ensure the above goals were accomplished.

As of August 2025, with the efforts of the MYSTRAN team, all of the major features have been completed. The major remaining efforts are related to fixing bugs, validating the solver, improving the documentation, and increasing performance. However, there are various other features on the horizon (such as the continued development of the MITC8 8-node quad) and some MYSTRAN specific features.

2 GENERAL DESCRIPTION OF INPUT DATA

A general description of MYSTRAN input data (referred to as a data section) is given in this section. A more detailed description of each of the three parts of the data section will be given in Section 6. Appendix A contains a sample MYSTRAN input and may be of help when reviewing this section.

The MYSTRAN data section consists of three distinct parts:

- The Executive Control section
- The Case Control section
- The Bulk Data section

The Executive Control section is an overall identification of the job and the solution type to be performed (e.g. statics, eigenvalues). It usually consists of a very few entries[1]. It begins with an ID entry and ends with a mandatory CEND entry. All Executive Control section entries are described in Section 6.1.

The Case Control section defines the job title that is printed out with the output, the loading for each of the different subcases, the constraint boundary conditions and the sets that define the grids and elements for displacement, load and stress output. The Case Control section begins with the entry following the Executive Control CEND entry and ends with the mandatory BEGIN BULK entry. The only requirement on the order of entries in the Case Control section is that the order makes sense when there are multiple subcases. The details of each of the Case Control section entries are given in Section 6.2.

The Bulk Data section defines the finite element model in detail. It begins with the entry immediately following the BEGIN BULK entry and ends with the mandatory ENDDATA entry. Grid points form the “mesh” of the finite element model and are defined with their locations (in any of several coordinate systems). The elements that make up the finite element model are defined by the grid points to which they are connected, by their physical properties and by their material

[1] “entry” is used to mean a single line of entry in the data section. It is a holdover from the familiar 80 column punched entries used to enter data into computers long ago. The MYSTRAN data section does consist of lines of entry that can contain data in columns 1 through, possibly, column 80 (each denoted as a physical entry). A logical entry can, in some instances, consist of more than one physical entry.

properties. Loads and boundary conditions are also defined in the Bulk Data section. In the case of eigenvalue analysis, the eigenvalue extraction method is also defined here.

All physical Bulk Data entries are broken down into 10 fields of 8 columns each with field 1 being a mnemonic that defines the type of entry (e.g. GRID for a grid point definition, PBAR for a bar element property definition, etc.). Since 10 fields may not be enough for some of the entries, provision is made to include "continuation" entries. For example, the PBAR Bulk Data entry that defines geometric properties for a bar element has three physical entries necessary to define all of the properties. These three physical entries comprise the one logical PBAR entry. This is explained in detail in the description of Bulk Data entries in Section 6.3. Suffice it to say here that a logical Bulk data entry in MYSTRAN may consist of several physical entries with the initial entry being called the "parent" entry and subsequent continuation entries (if necessary) called "child" entries. Since all logical Bulk Data entries have a mnemonic that defines which type of input it describes, there is no requirement on the order of *logical* entries in the Bulk Data section. Physical entries that make up a given logical entry must, however, be in order and grouped together.

3 FINITE ELEMENT MODEL

The finite element model is specified by defining:

- Grid points that locate the frame to which elements are connected
- Finite elements (connection, property and material definitions)
- Applied loads
- Constraints
- Mass at grid points and or of elements

The following sub-sections discuss each of these.

3.1 Grid points

3.1.1 Grid point and coordinate system definition

Grid points are defined on GRID Bulk Data section entries. The GRID entry gives the grid point number and the coordinates of the grid point in any of several types of coordinate systems. The grid point numbers can be any arbitrary integers containing from 1 to 8 digits as long as the numbers are unique among all grids. The GRID entry can also be used to specify constraint information. A "basic" coordinate system is implicitly defined and is rectangular. Grid coordinates are either defined in the basic system or in other rectangular, cylindrical or spherical coordinate systems whose location can be traced back to the basic system. If coordinate systems other than the implicitly defined basic system are used, their locations are defined using the CORD2R, CORD2C and CORD2S Bulk Data entries (for rectangular, cylindrical and spherical coordinate systems). These entries give the location of three points in some other coordinate system that is previously defined. This is cascaded until the last coordinate system is defined relative to the basic system.

In addition to locating grid points, the GRID entry references another coordinate system, known as the global coordinate system for that grid point. This global coordinate system is the system in which the overall (global) stiffness matrix is generated for each grid and in which constraints are applied and solution for displacements is obtained. Again, the basic system is the default for the global system at any grid but can be overridden on the GRID entry for the grid in question. It is important to realize that when reference is made to the "global" coordinate system, what is really meant is a collection of coordinate systems that may be different for each grid point. Alternatively, the global coordinate system for a grid point is also referred to as its displacement coordinate system.

Each grid point has six degrees of freedom: translations along three orthogonal axes and the orthogonal rotations about these three axes. The six degrees of freedom will be collectively referred to as the displacements of the grid point in question and are denoted as:

$$u_{1_g}, u_{2_g}, u_{3_g}, \theta_{1_g}, \theta_{2_g}, \theta_{3_g} \tag{X.X}$$

where g designates a grid point. In the case of a rectangular displacement coordinate system for a grid point, the three orthogonal translations are positive along axes that are at the grid and parallel to the three coordinate axes directions defined by a CORD2R entry. The three rotations are positive for right hand rule rotation (in radians) about these three axes. For a cylindrical displacement coordinate system for a grid point, the translations are along the radial, tangential and axial directions at the grid and the rotations are again positive for right hand rule rotation about these three axes. For a spherical displacement coordinate system, the three translations are in the radial, meridional and azimuthal directions with the rotations about these axes. Figure 3-1 shows these three coordinate systems.

The GRID entry also has a field that can be used to denote constraints that are for zero displacement for any of the six degrees of freedom for that grid point. These constraints are known as permanent single point constraints (or PSPC's).

3.1.2 Grid point sequencing

It is important to include provision for internally rearranging the order of the grids in order to obtain a global stiffness matrix that has a minimal bandwidth. The CPU time to perform linear equation solutions is directly dependent on the stiffness matrix bandwidth. In addition, several matrices have to be put into "banded" form for the LAPACK algorithms used in MYSTRAN. Thus, bandwidth is extremely important in determining the disk storage requirements for those matrices.

The sequencing method used in any execution of MYSTRAN is controlled via the Bulk Data PARAM GRIDSEQ entry. The user has several options for specifying sequencing that are basically manual or automatic, as explained below.

3.1.2.1 Automatic grid point sequencing

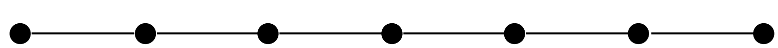
Automatic grid point sequencing to achieve a minimal stiffness matrix bandwidth is accomplished using an algorithm called BANDIT which is described in Reference 3. The code for accomplishing this was obtained from that author and is imbedded in MYSTRAN. BANDIT, when originally written, was a stand-alone program that generated SEQGP Bulk Data entries (see section on the Bulk Data section) which defined the sequence order for each grid. Within MYSTRAN, BANDIT is a subroutine which generates these SEQGP entries and MYSTRAN uses these to define the grid sequencing. BANDIT is the default sequencing method in MYSTRAN and is equivalent to including a Bulk Data PARAM GRIDSEQ entry with BANDIT specified in field 3 of the PARAM entry. When BANDIT sequencing is used, any user supplied SEQGP Bulk Data entries are ignored and a warning message is given.

3.1.2.2 Manual grid point sequencing

In manual grid sequencing, the user supplies the Bulk Data section SEQGP entries which are used to sequence the grids. However, only those grids which are to be re-sequenced from their initial order need to have their sequence number specified on SEQGP entries. In order to facilitate this MYSTRAN starts out with a predefined sequence order that can then be modified with the user supplied SEQGP entries. The predefined sequence order can be one of two possibilities (and is defined on the PARAM GRIDSEQ Bulk Data entry):

- Grid numerical order (PARAM GRIDSEQ GRID)
- Order of the grids as they appear in the Bulk Data section (PARAM GRIDSEQ INPUT)

The following beam model with seven grid points illustrates this:



Assuming that the user has the initial order set with PARAM GRIDSEQ GRID then grid 101 would be sequenced 1st initially. However, for a minimum stiffness matrix bandwidth, it should be sequenced so that it is 4th. Using the SEQGP entry, grid 101 can be re-sequenced to be 4th by giving it a sequence number between where grids 401 and 501 are sequenced. Since the sequence number can be a decimal value then grid 101's sequence number should be a number that is greater than 4 but less than 5 (say 4.1)

3.2 Elements

3.2.1 Element connection, property, and material definition

Elastic elements are defined by their connectivity (the grids to which they attach), by their geometric properties and, in all but the ELAS1 element, by their material properties. The mnemonic in field 1 of all elastic element connection entries begins with a "C" followed by the element name. The mnemonic in field 1 of a bar element connection entry, for example, is CBAR (in columns 1-4). Field 2 of a connection entry gives the element ID, which is an arbitrary integer (although elements must have unique IDs among the set of all elements). Field 3 of the connection entry for all one- and two-dimensional elements gives the ID of an element property Bulk Data entry that is used to specify geometric properties of the element. Following this on the element connection entry, the grid points to which the element connect are specified. With the exception of the scalar spring element, all elements have a local element coordinate system. This local element coordinate system is defined by the order of the grids on the element connection entry and by, for some elements, an orientation vector that is also defined on the element connection entry. This will be discussed in detail in each of the separate element sections below.

Element property entries define the geometric properties of the elements (e.g. cross-sectional areas, moments of inertia of bars, thickness of plates, etc.). The mnemonic in field 1 for all property entries begins with a "P" followed by the element name. The property entry for a bar element, for example, has PBAR in field 1 and has, in field 2, the property ID that was referenced on the connection entry. Field 3 specifies an ID of a material Bulk Data entry. The remaining fields define the geometric properties of the bar element and can take up to three physical entries for the complete description. For example, the PBAR entry has the following properties:

- Cross-sectional area
- Moments of inertia and product of inertia
- Torsional constant
- Mass per unit length
- Up to four locations, on the cross-section, where stresses are to be calculated

- Area factors for shear flexibility

Material properties are specified on the MAT1 Bulk Data entry for linear isotropic materials and on the MAT8 entry for linear orthotropic materials (plate elements only). Field 2 contains the material ID and the remaining fields contain material constants (such as Young's modulus, Poisson's ratio, mass density, thermal expansion coefficients, etc.).

The reason for the connection entries pointing to property entries which, in turn, point to material entries is the following: every element must have a connection entry but many of them may be for elements that have the same physical properties and there may be even fewer material entries needed. Also, in this manner, it is not required that the entries in the Bulk Data section be in any specific order with the exception that, for continuation entries, the child entries must follow the parent entry in order.

3.2.2 Elastic elements

3.2.2.1 Scalar spring (ELAS and BUSH elements)

The ELAS1 scalar spring element connects between two degrees of freedom. The CELAS1 Bulk Data entry defines the connection information, which consists of a pair of grid points and the displacement components at those grid points that the spring is to be connected between. In addition, the CELAS1 entry references a PELAS property entry that will define the spring rate, K, and a stress recovery coefficient, S, such that S times the elongation of the spring gives the stress that is output for the element. No material entry is needed for the CELAS1 element.

Care must be taken when using scalar spring elements that rigid body motion of the model is not constrained. For example, if the spring is connected between two non-coincident grids then rigid body motion of the model may be constrained if the degrees of freedom that the spring is connected to are not along a line between the grids.

Output for a spring element can include any, or all, of the following:

- Element nodal forces:
    - Output in either global or basic coordinates at all grids for selected elements
- Element stress (positive for positive engineering forces):
    - Stress calculated as the spring stress recovery coefficient (specified on the PELAS Bulk Data entry) times the spring elongation.

The BUSH element is a spring connecting two grid points. It can have up to 6 stiffness values (one for each displacement degree of freedom). The element connection can take into consideration that the two grid points are not coincident. It is a better choice for a scalar spring than the ELAS elements if the grids are not coincident. The BUSH can have the following element outputs:

- Element nodal forces:
  - Output in either global or basic coordinates at all grids for selected elements
- Element engineering forces:
- Element stress (positive for positive engineering forces):
  - Stress calculated as the spring stress recovery coefficient (specified on the PELAS Bulk Data entry) times the spring elongation.

3.2.2.2 BUSH element

The BUSH element connects between 2 grid points and can have up to 6 stiffness values defined. It is the same as the BUSH element in some of the NASTRAN software programs. It can have offsets in 3 directions from the line between the 2 grids. See the equations for the element in one of the Appendices.

3.2.2.3 Rod element

The rod is a one-dimensional element that is connected between two grid points (G1 and G2) and which has stiffness for axial and torsional motion. The CROD entry specifies the element connection for the rod and the PROD entry defines the area, torsional constant, torsional stress recovery coefficient and mass per unit length for the rod. The local element coordinate system only requires the definition of one axis; namely along the axis from grid point G1 through grid point G2 as shown in Figure 3-2.

Output for a rod element can include any, or all, of the following:

- Element engineering forces:
  - Axial force (positive is tension)
  - Torsion (positive as shown on Figure 3-2)
- Element nodal forces:
  - Output in either local, global, or basic coordinates at all grids for selected elements
- Element stresses (positive for positive engineering forces):
  - Axial stress and margin of safety
  - Torsional stress and margin of safety

3.2.2.4 Bar element

The bar element is a simple beam that has its shear center coincident with its neutral axis. It is defined using the CBAR connection entry and the PBAR property entry. It can carry bending and shear in two planes, axial force and torque. Shear flexibility can also be included. Figures 3-3 and 3-4 show the element coordinate system and element engineering forces.

The ends of the bar element can be offset from the grids G1 and G2 as indicated on Figure 3-3. This is a rigid offset and can have components in up to three orthogonal directions. The components of the offset vectors are specified on the CBAR entry in the global coordinate systems of grids G1 and G2, respectively.

The v vector in Figure 3-3 is used to determine Plane 1 and Plane 2 of the bar as indicated in the figure. This is necessary so that the moments of inertia (I1, I2, I12) on the PBAR entry can be interpreted correctly. The v vector is specified on the CBAR entry as either three components of a vector measured from end "a" in the global coordinate system of grid G1, or by a grid point, G0, along the v vector (which, together with end "a", defines v). The moment of inertia, I1, on the PBAR entry is the moment of inertia about the element z_e axis. Moment of inertia, I2, on the PBAR entry is about the element y_e axis. Planes 1 and 2 need not be principal planes. If they are not, then the product of inertia, I12, must be specified on the PBAR entry.

The bar can be disconnected from a grid point in any of the six degrees of freedom, resulting in the corresponding force(s) in the bar being zero. This is referred to as a "pin flag" feature for the bar. Either end of the bar can be pin flagged. However, the pin flags specified cannot result in the bar being completely disconnected from the grid mesh in any rigid body degree of freedom. For example, degree of freedom 1 (axial) cannot be pin flagged at both ends. This would result in the bar being disconnected from the grid mesh along its x_e axis.

The following output is available for the bar element:

- Element engineering forces:
    - Axial force
    - Torque
    - Bending moments at both ends in each of the two planes
    - Shear in the two planes
- Element nodal forces
    - Output in either local, global, or basic coordinates at all grids for selected elements
- Element stresses (positive for positive engineering forces):

- Stresses due to bending in the two planes at up to four points defined by the user on the PBAR entry
- Stress due to axial force
- Maximum, and minimum, combined bending and axial stress at each end of the bar
- Margins of safety for tension and compression stresses, flagged when they are less than zero
- Torsional stress (if SCOEFF is input on the Bulk data PBAR entry)

Maximums and minimums are determined from the stress due to axial force and the bending stresses at the four points, at each end, if the user specified those points on the PBAR entry. Otherwise, the maximums and minimums are based on the stress due to axial force.

3.2.2.5 Plate and Shell Elements: Overview

MYSTRAN provides for both triangular and quadrilateral plate elements that include membrane and/or bending stiffness, several of which may be used to model thick plates consistent with Mindlin plate theory. All of the plate element formulations have uniform thickness. The separate connection entries available for this modeling are given below (in all cases the mid-plane of the plate can be offset from the grids – however, this may not available for the MITC4+ and it's not available for the CQUAD8).

Currently, MYSTRAN has two versions of the QUAD4 quadrilateral plate element, referred to as MIN4 and MITC4+. The MIN4 version is described in Reference 5. The MITC4+ version is described in Reference 15. The default QUAD4 is the MIN4 version. Both versions (MIN4 and MITC4+) are differentiated by the Bulk Data File PARAM named QUAD4TYP. A value of QUAD4TYP = MIN4 uses the quad in Reference 5, whereas a value of MITC4+ uses the quad element in Reference 15. Note that the MITC4+ element formulation is newer, but its current implementation in MYSTRAN is less comprehensive than for the MIN4 (discussed in the following section).

Note that in Version 2.06 of MYSTRAN, the MIN4T version of the QUAD4 element (described in Reference 9) was added to correct the deficiency in the MIN4 QUAD4 that could develop stresses in rigid body motion for elements that were not rectangular. The MIN4T QUAD4 element is made up of 4 non-overlapping TRIA3 elements. However, the MIN4T demonstrated poor results for the problem of a plate with a hole (incorrect stress concentration). Therefore, the MIN4T has been depreciated as of 2025-09-01.

- Combination Membrane-Bending Elements:
    - CTRIA3: triangular element for modeling thick plates and shells
    - CTRIA3K: triangular element for modeling thin plates and shells
    - CQUAD4: quadrilateral element for modeling thick plates and shells
    - CQUAD4K: quadrilateral element for modeling thin plates and shells
    - CQUAD8:
- In-plane shear element Elements:
    - CSHEAR: quadrilateral element for modeling thin shear plates

The property entry used for the combination membrane-bending elements is either the PSHELL or PCOMP/PCOMP1 entry. The SHEAR element properties are specified via the PSHELL entry. The PSHELL entry has provision for specifying membrane, bending and transverse shear properties (CTRIA3K, CQUAD4K do not have transverse shear flexibility). As with other property entries, the PSHELL entry has the property ID in field 2 and up to three material IDs (fields 3, 5 and 7); one each for membrane, bending and transverse shear. In addition, the membrane, bending and transverse shear properties themselves are input (fields 4, 6 and 8). A mass per unit area can also be input (field 9). The membrane, bending and transverse shear properties and material IDs are discussed in detail below.

- PSHELL Property Values and Material IDs:
    - Membrane
        - Field 3 specifies MID1, the ID of a material entry for the membrane portion of the plate. If this field is left blank, no membrane stiffness will be computed.
        - Field 4 specifies TM, the membrane thickness. This is required, even if the MID1 field is left blank, since it is used in the computation of bending and transverse shear properties.
    - Bending
        - Field 5 specifies MID2, the ID of a material entry for the bending portion of the plate. If this field is left blank, no bending stiffness or transverse shear flexibility will be computed.
        - Field 6 specifies 12(I/TM**3), a normalized bending property where I is the moment of inertia per unit width of the plate and TM is the membrane thickness discussed above. This normalized bending property has a default value of 1.0. If field 6 is left blank, it signifies a homogeneous plate.
    - Transverse Shear
        - Field 7 specifies MID3, the ID of a material entry for the transverse shear portion of the plate. If this field is left blank, no transverse shear flexibility will be calculated. Only the CTRIA3 and CQUAD4 thick plate elements have the capability for transverse shear flexibility.
        - Field 8 specifies TS/TM, the ratio of shear to membrane thickness. This has a default value of 5/6 = 0.833333, if field 8 is left blank. This is an historic value that is based on the shear stress distribution in a solid cross-section beam. A more realistic value for plates is based on Mindlin plate theory and is $\frac{\pi^2}{12}$ (or 0.822467), which is only a few percent different than the historic value. The default value for all PSHELL property entries can be reset on the

Bulk Data entry PARAM (with name TSTM_DEF in field 2 and the new value in field 3).

The PCOMP or PCOMP1 property entry is for defining the plies (lamina) of composite laminates. Each ply can have a distinct material property that can be isotropic, orthotropic, or anisotropic. The assumption is made that each ply is in a state of plane stress, the bonding material between the plies is perfect, and two-dimensional plate theory can be used for the laminate. This is consistent with Classical Laminate Theory.

The triangular and quadrilateral element coordinate systems are shown in the CTRI3, CQUAD4, and CQUAD8 entries (cards) shown later in the manual. Figures 3.1 and 3.2 show the convention for plate force resultants, which are the basis for calculating element stresses.

The quadrilateral elements can accommodate some out of plane warping, but they are generally intended for use as flat elements. When the quadrilateral element has out of plane distortion, the $x_{\text{element}} - y_{\text{element}}$ plane for the element (as shown in CQUAD4 entry) (anything about CQUAD8?) is the mean plane between the grids.

Output for the plate elements includes:

- Element engineering forces:
    - Membrane force resultants (force/length) as shown on Figure 3.1
    - Bending moment resultants (moment/length) as shown on Figure 3.2
    - Transverse shear force resultants (force/length) for the QUAD4 and TRIA3 as shown on Figure 3.2
- Element nodal forces
    - Output in either global or basic coordinates at all grids for selected elements
- In plane element stresses at fiber distances Z1 and Z2 (on the PSHELL entry, with +/-TM/2 as default) that are derived from the above force and moment resultants
    - Normal stress in the x_{element} direction
    - Normal stress in the y_{element} direction
    - In-plane shear stress
    - Major and minor principal stress and the associated angle
    - Max in-plane shear stress
    - von Mises or max shear stress
    - Transverse shear stresses (for the QUAD4 and TRIA3)

For the QUAD4, the stresses, strains, and element engineering forces can be output at the element center as well as at the corner nodes of the element. The TRIA3 element has constant stress/strain so only one output per element is provided.

3.2.2.6 Plate and Shell Elements: QUAD4 Detailed Discussion

MYSTRAN supports two formulations for the CQUAD4: MIN4 (default) and MITC4+. The formulation that is used is controlled by the QUAD4TYP parameter. The MITC4+ element formulation is newer, but its current implementation in MYSTRAN is less comprehensive than for the MIN4. See the following tables:

Linear Static (SOL 101)

	MIN4	MITC4+
Isotropic	Y	Y
Orthotropic	Y	Y
Anisotropic	Probably	Probably
PCOMP	Y	N
PSHELL MID1	Y	Y
PSHELL MID2	Y	Y
PSHELL MID3	Y	Y
PSHELL MID4	Probably	N
Gravity	Y	Y
Thermal load	Probably	Y
Pressure load	Probably	Y
Offset	Unknown	Unknown
Stress Output	Y	Y
Strain Output	Y	Y
Element Force Output	Y	Y

Buckling (SOL 105)

	MIN4	MITC4+
Isotropic Buckling	Y	Y
Orthotropic Buckling	Y	Probably
Anisotropic Buckling	Probably	Probably
PCOMP, symmetric	Y	N
PCOMP, unsymmetric	N	N
PSHELL, symmetric, MID1≠MID2	Probably	Y
PSHELL, unsymmetric, [B]≠[0]	N	N
PSHELL, unsymmetric, via MID2 manual modification*	Probably	Y

For the PSHELL, MID2 represents the bending stiffness matrix, which can be correlated to a laminate's [D] matrix. For an unsymmetric laminate, [D] may be substituted with [D], to obtain an approximate solution (Reference 14). $D^*=D-BA^{-1}B$.

Natural Frequency/Normal Modes (SOL 103)

The MITC4+ is supported for SOL 103. The MIN4 is probably supported.

3.2.2.7 Plate and Shell Elements: QUAD8 Detailed Discussion

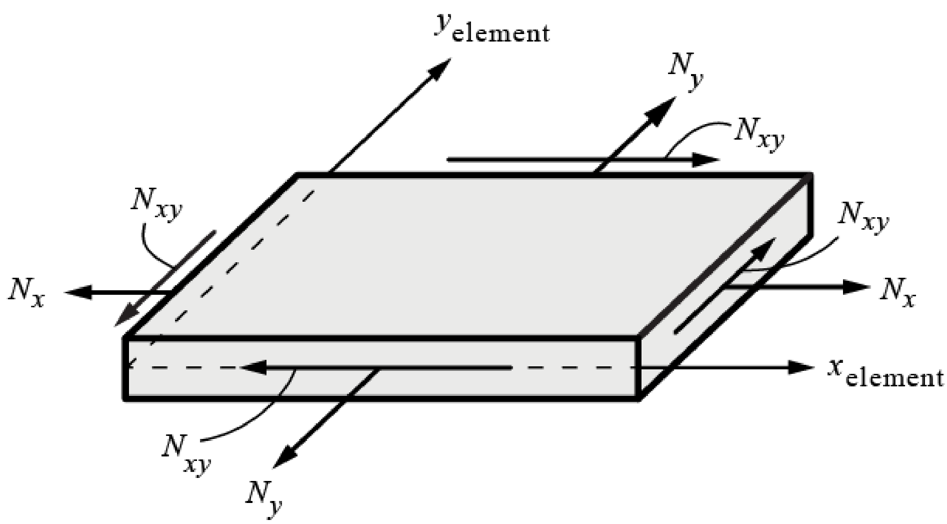
MYSTRAN has limited support for the CQUAD8 (Reference 16). It supports an isotropic material with simple mechanical loading.

Linear Static (SOL 101)

	MITC8+
Isotropic	Y
Orthotropic	N
Anisotropic	N
PCOMP	N
PSHELL MID1	Y
PSHELL MID2	Only if MID1=MID2
PSHELL MID3	Y
PSHELL MID4	N
Gravity	N
Thermal load	N
Pressure load	N
Offset	N
Stress Output	Y
Strain Output	Y
Element Force Output	Y

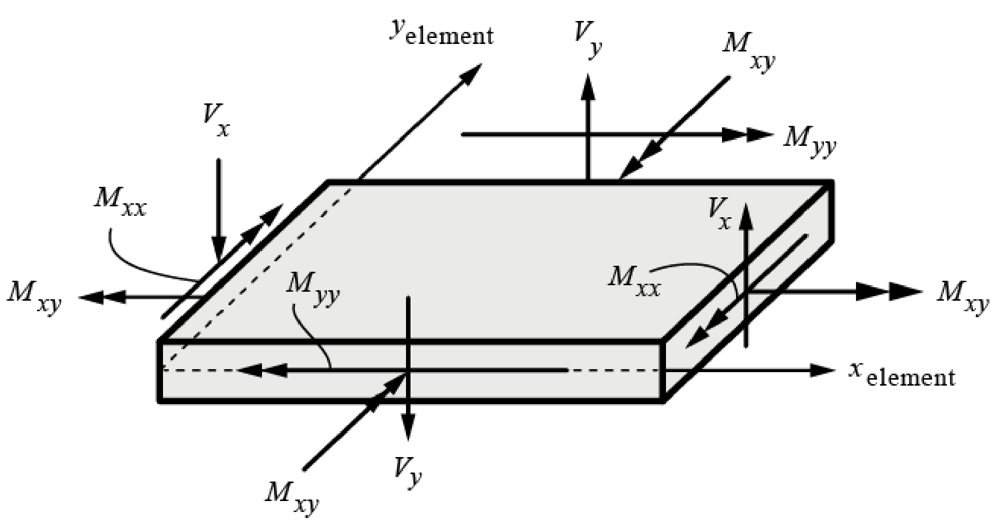
3.2.2.8 Plate and Shell Elements: Running Loads/Moments

The following two figures show the show running loads and running moments (the forces/moments per unit width).



shell membrane force resultants
(running loads = load per width)

Figure 3.1:



shell bending moment and transverse shear force resultants
(running moments = moment per width)

Figure 3.2:

3.2.2.9 3D Solid elements

MYSTRAN has hexahedra, pentahedra and tetrahedra elements for modeling of 3D structures. The CHEXA hex element comes in 8 node and 20 node versions. The CPENTA element comes in 6 node and 15 node versions. The CTETRA is available in 4 node and 10 node versions. Properties for these solid elements are specified on the PSOLID Bulk Data entry, with several choices for integration order and integration scheme. Material properties are specified on the MAT1 entry. Outputs for the solid elements are in the form of stresses at the element center and can include von Mises and max shear results.

3.2.3 Rigid elements

In addition to the elastic elements discussed above, MYSTRAN also has a capability for specifying a rigid relationship among specified degrees of freedom. These elements are suited for situations where a portion of a model is so much stiffer than the remainder that it could cause ill conditioning of the stiffness matrix if it were modeled with elastic elements. When rigid elements are used, selected degrees of freedom are eliminated from the solution set using equations (automatically generated in MYSTRAN) that represent rigid body notion of the "dependent" degrees of freedom based on rigid motion of a selected set of "independent" degrees of freedom. Specification of rigid elements in MYSTRAN is accomplished with Bulk Data entries similar to elastic element connection entries (however, no property ID is needed). Field 1 of the rigid element connection entry, like elastic elements, has a mnemonic describing the rigid element type.

Care must be taken when using rigid elements in thermal distortion analyses. The rigid elements do not expand with temperature and can otherwise constrain a model that the user expects to expand in a stress-free manner.

3.2.3.1 RBE2 rigid element

The RBE2 element specifies that the motion of a set of grid points (all having the same set of dependent degree of freedom numbers) are dependent on the six degrees of freedom at another grid point.

An example of the equations developed by MYSTRAN to eliminate the dependent degrees of freedom is shown in Figure 3-7 (for a simple one-dimensional problem). In this example, degrees of freedom 1, 2 and 6 at grid 103 will be eliminated from the solution set of degrees of freedom using the equations shown. The user does not have to input these equations; only the Bulk Data RBE2 field entries.

3.2.4 RBE3 element

The RBE3 element is not a rigid element but is used to distribute loads and mass from some central grid point to other grids in the model. It is defined by a dependent, central, point at which the load or mass is defined along with grids to which the load or mass are to be distributed along with weighting factors at these distributed grids. The dependent point on the RBE3 should never be connected to other elastic elements in the model to avoid stiffening of the structure by the RBE3 element. Appendix E gives a mathematical derivation of the RBE3 equations which reduce the dependent grid point out of the model equations of motion.

3.2.5 RSPLINE element

The RSPLINE element is generally used to model transitions from a coarse to a fine mesh. In MYSTRAN, the RSPLINE element connects to 2 independent end points. Displacements along and perpendicular to the line between the end points is interpolated using the 6 displacements of the end points as follows:

- Displacements along the line and rotations about the line are linear
- Displacements perpendicular to the line are cubic
- Rotations normal to the line are quadratic

3.3 Applied loads

MYSTRAN provides several methods of specifying applied loads:

- Forces and/or moments applied directly to grids
- Pressure loading on plate elements
- Gravity loads
- Equivalent loads due to thermal expansion
- Equivalent loads due to enforced displacements
- Loads on scalar points (SLOAD)

All of the Bulk Data entries defining these loads have a set ID which is used to control whether they are used in a particular subcase. Thus, the user is free to include load entries in the Bulk Data that may not be used in a particular execution of the program (that might be used in a subsequent run, for example).

3.3.1 Forces and moments directly applied to grids

Bulk Data entries FORCE and MOMENT are used to define forces and/or moments applied directly to a grid point. Both of these entries have, in field 2, a set ID.

Field 3 of both the FORCE and MOMENT entry specifies the grid point where the load is to be applied. Field 5 specifies an overall scale factor and fields 6 – 8 specify the vector components of the load. The load applied in a component direction is the product of the overall scale factor times the vector component in that direction. The vector components are in a coordinate system whose ID is specified in field 4.

FORCE and MOMENT entries to be used in a particular subcase must be requested in Case Control with a LOAD = SID Case Control entry. The SID is either the set ID from the FORCE and/or MOMENT entries or is the set ID of a Bulk Data LOAD entry (see below) that has the FORCE and/or MOMENT set IDs specified.

3.3.2 Pressure loads on plate elements

Pressure loads normal to the surface of plate elements can be specified on PLOAD2 and PLOAD4 Bulk Data entries. As with the grid point load entries discussed above, the PLOAD entries have a set ID in field 2 that must be referenced (directly or indirectly) in Case Control in order to be used for a particular subcase. The pressure value is specified in field 3. The remainder of the entry presents two options for specifying what plate elements are to have this pressure value. One option is to list the element IDs using in fields 4 through 9 of the parent entry and, if necessary, fields 2 through 9 of continuation entries. The other option allows the elements to be specified using a THRU option, in which case any element whose ID is in the range of EID1 (field 4) through EID2 (field 6) will receive the pressure value in field 3.

Pressure loads are requested in Case Control the same as was described for the FORCE and MOMENT entries (either directly or by use of the LOAD Bulk Data entry).

3.3.3 Gravity loads

Gravity loads for the model are specified using the GRAV Bulk Data entry. The GRAV entry specifies an acceleration vector that, in conjunction with the mass at the grid points (discussed later), allows MYSTRAN to calculate static forces at all of the grid points due to the specified acceleration using the inertia properties of the model (grid point masses, etc., discussed later). As with other loads, the GRAV entry has a set ID in field 2. Fields 4 through 7 specify the magnitude and vector components of the acceleration in a coordinate system whose ID is given in field 3. The magnitude and/or vector components must be given in units consistent with model mass, discussed in a later section.

Gravity loads are requested in Case Control the same as was described for the FORCE and MOMENT entries (either directly or by use of the LOAD Bulk Data entry).

3.3.4 Equivalent loads due to thermal expansion

The equivalent loads due to thermal expansion are calculated automatically in MYSTRAN based on grid and/or element temperature data supplied by the user on a variety of Bulk Data entries, listed below, all of which have a set ID in field 2 of the entry:

- Grid temperature definition Bulk Data entries:
    - TEMPD specifies a default temperature for all grids
    - TEMP specifies a temperature for grids listed on this entry. These temperatures override any default values on TEMPD entries.
- Element temperature Bulk Data entries:
    - TEMPRB specifies average element temperatures for ROD and BAR elements as well as temperature gradients through the depth for BAR elements
    - TEMPP1 specifies average element temperatures and gradients through the thickness for plate elements

When a temperature load is to be used, all of the elements in the model must have a temperature defined. This may be done either indirectly using a TEMPD or TEMP entry that defines the temperatures of the grids to which the element connects, or directly by specification on a TEMPRB or TEMPP1 element temperature entry. Thermal expansion coefficients and reference temperatures, needed in the calculation of equivalent loads due to thermal expansion, must be specified on material Bulk Data entries.

The user must request temperatures in Case Control with the Case Control entry TEMP = SID where SID is the set ID on the above Bulk Data temperature entries which define the temperatures for the model.

3.3.5 Equivalent loads due to enforced displacements

If the user knows, a priori, the displacement (translation or rotation) of some degrees of freedom, MYSTRAN handles this by what is referred to as “enforced displacements”. The user specifies the known displacement on a Bulk Data SPC entry (in the global directions for the grid) and MYSTRAN uses this as a constraint. The Bulk Data SPC entries’ set ID must be selected in Case Control with the Case Control entry SPC = SID, where SID is the set ID of the Bulk Data SPC entries defining the enforced displacements.

The program calculates loads necessary to enforce this constraint and applies them to the structure in combination with all other loads specified. When forces of constraint are calculated in the program, the forces listed (in the output, if Case Control entry SPCFORCES is included) are those necessary to make the degrees of freedom displace the amounts that were specified as enforced displacements.

3.3.6 Loads due to rigid body rotation about a specified grid (RFORCE)

The finite element model can have loads calculated due to a rigid body angular velocity and/or angular acceleration. The loads are calculated as if the body were rotating when, in actuality, it is fixed. The equivalent loads due to this angular velocity and acceleration are applied to the fixed body. In this fashion, situations such as rotating turbines with centripetal forces can be simulated. This force is calculated via the Bulk data entry RFORCE.

3.3.7 LOAD Bulk Data entry – combining loads

Loads defined via the FORCE, MOMENT, GRAV and PLOAD2 entries that have different set IDs can be combined into one set for use in a subcase using the LOAD Bulk Data entry (not to be confused with the LOAD Case Control entry). The LOAD Bulk Data entry has a set ID in field 2. The following fields (including possible continuation entries) specify which of the individual load sets to use. This is specified as pairs of set IDs (of FORCE, MOMENT, GRAV or PLOAD2 loads) and scale factors for each of the separate loads. In addition, an overall scale factor for the combination of the loads on the LOAD Bulk Data entry is defined in field 3.

3.4 Constraints

3.4.1 Single point constraints

Single point constraints (SPC's) are needed for the following reasons:

- To specify boundary conditions where the model is to be grounded. These constraints will result in those degrees of freedom being zero and will also result in, generally, non-zero forces of constraint at the specified degrees of freedom.
- To remove singularities in the model. The global stiffness matrix is built on the basis of six degrees of freedom (3 translations and 3 rotations) per grid point which, for some models, means that some degrees of freedom may not have any stiffness. For example, a 2D model of a plate for bending and membrane action would have, at most, five degrees of freedom per grid since the plate elements have no stiffness for rotation about the normal to the plate. Thus, this plate model will have a singular global stiffness matrix for the degrees of freedom representing rotation about the normal to the plate. The user has a choice of identifying these explicitly or by having MYSTRAN constrain degrees of freedom that are singular through the use of an AUTOSPC feature (see Bulk Data PARAM entry for parameter AUTOSPC). In either event, these degrees of freedom are constrained to zero prior to solving for the displacements. If there is no stiffness for these degrees of freedom, the forces of constraint for them will be zero
- To specify enforced displacements at degrees of freedom where the user knows, a priori, the nonzero value of those displacements.

For the user defined SPC's the constraints are specified on SPC or SPC1 Bulk Data entries (or as "permanent" single point constraints in field 8 of the GRID Bulk Data entry). Both the SPC and SPC1 entries have a set ID in field 2. In addition, there is a SPCADD Bulk Data entry that can be used to combine requests made by the SPC and/or the SPC1 entries. The constraints specified on the SPC, SPC1 or SPCADD entries must be selected in Case Control with the SPC = SID Case Control entry, where SID is the set ID of either a SPCADD or of one or more SPC and/or SPC1 Bulk Data entries.

The SPC Bulk Data entry must be used for nonzero enforced displacements. Either the SPC or SPC1 entry (two different methods of specifying zero constraints of selected degrees of freedom) can be used for the other types of SPC's.

There can be only one SPC request in Case Control for any one MYSTRAN execution.

3.4.1.1 AUTOSPC Feature

The AUTOSPC feature mentioned above is done automatically in MYSTRAN unless the user includes a Bulk data PARAM AUTOSPC entry with an N in field 3 to request that MYSTRAN do not perform an AUTOSPC calculation. The explanation of the AUTOSPC feature that follows assumes the user is familiar with the displacement set notation defined in Section 3.6.

In order to identify singular degrees of freedom when the G-set singularity processor is run, MYSTRAN uses a comparison of stiffness terms to a small number and constrains the degree of freedom if this criterion is met. The specific procedure is explained below:

- For each grid of the G-set stiffness matrix, the two 3x3 stiffness matrices (one for translation and one for rotation) are obtained for one grid.
- The three eigenvalues and eigenvectors of the two 3x3 matrix are determined.
- The ratio of each of the three eigenvalues to the eigenvalue that is the max among the three is determined. A comparison of the ratio to AUTOSPC_RAT (see PARAM AUTOSPC Bulk Data entry field 4) is made.
- If the ratio is less than the criteria, one degree of freedom will be constrained. The degree of freedom that is constrained is the one whose eigenvector absolute value is largest (using the eigenvector corresponding to the eigenvalue for that ratio).

If the eigenvalues of the 3x3 matrices are exactly zero, then no forces of constraint will result from the AUTOSPC's. There are instances in problems with near singularities in which the eigenvalue ratios are not exactly zero and, in those cases, some small force of constraint will result. These should be generally negligible, but the user should always request output of the forces of constraint, especially when using the AUTOSPC feature. An example of a case where these small ratios can be nonzero is in the case of modeling a curved surface with only plate elements. If the user makes several models and continually refines the mesh, then at some point two contiguous elements will become nearly parallel. At this point there will be negligible stiffness at a common node for rotation about the normal to the plate. When this stiffness gets small enough, MYSTRAN will constrain it if the AUTOSPC feature is turned on.

Through this procedure, the AUTOSPC feature can identify many, but perhaps not all, singular degrees of freedom. In the case where the model has either rigid elements or multi-point constraints

(MPC's) a situation can arise where the G-set stiffness matrix is singular. When the G-set singularity processor is called for each grid, any grid that is specified as independent on an MPC or rigid element is skipped. This is done since these grids may not have any stiffness (they may have no elastic element connected to all six grid components) in the G-set stiffness matrix but may get stiffness when the MPC and rigid element degrees of freedom are eliminated. Thus, they must be ignored until after the reduction from the G-set to the N-set. After this reduction, the N-set stiffness matrix will be scanned (if AUTOSPC_NSET on the PARAM AUTOSPC entry is equal to 1) to see if any rows are null. There may be null rows if some of the independent degrees of freedom on MPC's and rigid elements do not have stiffness at this point. If any rows are null, the degrees of freedom corresponding to these rows are AUTOSPC'd also. AUTOSPC_NSET can also be set to 2 or 3 also. If equal to 2, then MYSTRAN will remove any N-set degrees of freedom whose diagonal stiffness ratio (to max diagonal stiffness) is less than AUTOSPC_RAT. If it is equal to 3, then both actions for AUTOSPC_NSET = 1 and 2 are applied. In general, AUTOSPC_NSET = 1 (default) is recommended.

3.4.2 Multi point constraints

Multi point constraints (MPC's) may be needed for the following reason:

- To specify linear dependence of some degrees of freedom on other degrees of freedom. The equation relating the linear dependence is specified on MPC Bulk Data entries. Rigid elements are really automated multi point constraints that represent rigid motion of an "element" and are a subset of the more general MPC relationship. MPC's are a more general way of specifying linear dependence of some degrees of freedom on other degrees of freedom.

There can be only one MPC request in Case Control for any one MYSTRAN execution.

3.4.3 Boundary degrees of freedom in Craig-Bampton analyses (SUPORT)

This feature is primarily included for Craig-Bampton (CB) model generation. It provides a set of degrees of freedom (DOF's) that are to be boundary DOF's used in calculating modal properties of a substructure. Reference 11 and Appendix D describe the Craig-Bampton method as it is currently implemented in MYSTRAN. The boundary DOF's are identified on Bulk Data SUPORT entries and define the R-set of degrees of freedom (see later discussion on displacement set notation). For CB analyses the modal properties of the substructure are determined with fixed boundaries so that the R-set is constrained to zero for the purposes of calculating modal properties of the substructure. The SUPORT feature is not intended for use in any of the other MYSTRAN solutions (e.g. statics, eigenvalues). If the SUPORT feature is used in any solution method other than Craig-Bampton, the result is the same as if the SUPORT DOF's were identified as constrained to zero motion on SPC or SPC1 Bulk Data entries.

3.5 Mass

Mass for the finite element model can be specified in several ways:

- Mass density for finite elements (specified on property Bulk Data material entries)
- Mass per unit length, or per unit area, for finite elements (specified on element property Bulk Data entries)
- Concentrated masses at grids (using CONM2 Bulk Data entry) with possible offsets and moments of inertia.

Any of the above can be used in combination, or separately, in defining the mass for any finite element (or grid point in the case of CONM2's) in the model.

3.5.1 Mass density on material entries

The MAT1 Bulk data entry used to define material properties, discussed earlier, has a field to specify the mass density of the material. This mass density, together with the volume of each finite element, can be used by MYSTRAN to calculate a mass for each element. For example, plate elements have a surface area defined by the grid locations of the three or four grids that the plate element is connected to. The plate element thickness (membrane thickness on the property entry PSHELL) along with the surface area defines a volume for the element. The mass density on the MAT1 entry times this volume defines the mass for this element. Similarly, a beam element (BAR) has a length defined by the two grids that the element connects to and has a cross-sectional area specified on the PBAR entry. The element volume is calculated from this area and length.

3.5.2 Mass per unit length or area of finite elements

Mass can also be defined using data entered on the element property Bulk Data entries. The PBAR entry, for example, has a provision for specifying mass per unit length of the bar. The plate element property entries have a field in which a mass per unit area can be defined. These can be used in conjunction with the other two methods of defining mass, or can be used independently to completely define the mass for an element.

3.5.3 Concentrated masses at grids

Concentrated masses can be placed directly at grid points using the CONM2 Bulk Data entry. This entry provides the user with the option of specifying a mass value with possible offsets from the grid point and mass moments of inertia, including products of inertia. The offsets and inertia's can be specified in a coordinate system referenced on the CONM2 entry. Use of the CONM2 presents a convenient method for including "rigid masses" at grid points. The CONM2 entry has an "element" ID in field 2, the ID for the grid to which the mass is attached in field 3, the coordinate system in which the mass properties are specified in field 4 and the mass value in field 5. The remainder of the logical entry (which can span two physical entries) is used to specify possible offsets and moments and products of inertia. The offsets are the relative coordinates of the c.g of the mass with respect to the grid and are specified in the coordinate system whose ID is in field 3. The inertia values are the moments and products of inertia of the mass about its own c.g., also with respect to the coordinate system specified in field 3. Moments of inertia about any of the three axes of this coordinate system can be specified. There are, possibly, six products of inertia but only the three independent ones need be specified. The offsets and inertia values are optional.

A 6 x 6 symmetric mass matrix, *M,* (at the c.g. of the mass) is created by MYSTRAN as given by:

$$M = \begin{bmatrix} m & 0 & 0 & 0 & md_3 & -md_2 \\ & m & 0 & -md_3 & 0 & md_1 \\ 0 & & m & md_2 & -md_1 & 0 \\ & & & I_{11} & -I_{12} & -I_{13} \\ & SYM & & & I_{22} & I_{23} \\ & & & & & I_{33} \end{bmatrix} \qquad 0\text{-}1$$

In the above, m denotes the mass value on the CONM2 entry and d_1, d_2 and d_3 denote the offsets of *m* from the grid and Iij are the six independent moments and products of inertia. The 1,2 and 3 subscripts refer to the 3 axes of the coordinate system whose ID is in field 4 of the CONM2 entry.

3.5.4 Model total mass

MYSTRAN can calculate the rigid body mass properties (total mass, overall c.g. and moments of inertia) of the finite element model if the user desires. The calculation is done in the basic coordinate system and can be done relative to any user specified grid point. The Bulk Data entry PARAM with a parameter name of GRDPNT in field 2 is used to request output of the rigid body mass properties of the model. If field 3 of this PARAM entry contains a grid point ID, the calculation will give the mass properties relative to that grid point. If field 3 is blank (or zero), the calculation will be done relative to the origin of the basic coordinate system.

3.5.5 Mass units

All units of mass input in the Bulk data must be consistent. However, the user can input these in terms of mass or weight. If weight units are used, the finite element mass matrix must be converted back to mass units prior to performing eigenvalue analyses. This is accomplished using the Bulk Data PARAM entry with a parameter name of WTMASS in field 2. The value of the WTMASS parameter is used to <u>multiply</u> the mass matrix prior to eigenvalue analyses. Thus, if the user has input weight units instead of mass units a WTMASS value of 1.0/gravity (e.g. 1.0/386 if gravity is 386 in/sec^2) must be used. The units of the output for the rigid body mass properties of the whole model (discussed above) are the same as the input units (mass or weight).

If the user has specified a gravity loading (see section on Applied Loads) the units of the acceleration on the GRAV entry must also be consistent with the units of mass. For example, if mass units are used then the GRAV entry should specify the gravity loading in acceleration units. However, if weight units are used the gravity loading should be specified in terms of g's.

3.6 Displacement set notation

As was mentioned in an earlier section, MYSTRAN originally constructs stiffness and mass matrices for the model based on all grid points having six degrees of freedom. These matrices are referred to as the G-set matrices such that if there are n grid points, the original stiffness and mass matrices will have 6n rows and columns (i.e., the G-set consists of 6n degrees of freedom). The stiffness matrix for these G-set degrees of freedom must, therefore, be singular since no constraints of any kind will have been imposed on it; either through specification of boundary constraints or through rigid elements (which cause constraints as well). In order to reduce this matrix to the independent degrees of freedom, MYSTRAN partitions and reduces the G-set to the independent degrees of freedom, denoted as the L-set. This section describes the various sets as MYSTRAN reduces from the G-set to the L-set.

The G set is initially constructed in a degree of freedom (DOF) order that is discussed in the section on Grid point sequencing. The G-set is then partitioned into two sets; one of which consists of all degrees of freedom denoted as dependent on rigid elements or multi-point constraints (M-set) plus all others (denoted as the N-set). In displacement set notation, then:

$$U_G = \begin{Bmatrix} U_N \\ U_M \end{Bmatrix} \tag{0-2}$$

The M-set degrees of freedom are eliminated using the multi point constraint equations as well as equations developed in MYSTRAN based on the rigid element geometry and the dependent degrees of freedom in the N-set. Following this reduction, the stiffness and mass matrices are in terms of the N-set degrees of freedom. This N-set is further partitioned into two sets; those that are constrained via single point constraints (denoted as the S-set) plus all other degrees of freedom from the N-set (denoted as the F-set). The displacement set notation for this is:

$$U_N = \begin{Bmatrix} U_F \\ U_S \end{Bmatrix} \tag{0-3}$$

The S-set degrees of freedom are eliminated using the single point constraints (both zero constraints and enforced displacements). Following this reduction, the stiffness and mass matrices are in terms of the F-set degrees of freedom. At this point, the F-set may well be an independent set of degrees of freedom. However, MYSTRAN allows for a further reduction of the F-set based on Guyan reduction (static condensation). A Guyan reduction is necessary, for real eigenvalue analysis by the Givens method, if there are any zeros on the diagonal of the mass matrix. Zero diagonal terms would occur, for example, if the mass matrix had mass terms only for the translation degrees of freedom and not for the rotation degrees of freedom. Other situations could also result in zero diagonal terms in the mass matrix. The degrees of freedom to be eliminated by static condensation are denoted as the O-set. The O-set is defined using the Bulk Data entry OMIT or OMIT1 (or alternately via the ASET or ASET1 entry). In general, there is no reason to specify an O-set for static analysis. At any rate, the F-set is partitioned into these 0-set degrees of freedom plus all remaining degrees of freedom in the F-set (denoted as the A-set). The displacement set notation for this is:

$$U_F = \begin{Bmatrix} U_A \\ U_O \end{Bmatrix} \tag{0-4}$$

The O-set degrees of freedom are eliminated via Guyan reduction (static condensation). Following this reduction, the stiffness and mass matrices are in terms of the A-set degrees of freedom. In the static and eigenvalue analysis solutions, the A-set is the final, independent, set of degrees of freedom. However, for Craig-Bampton (CB) model generation the A-set is comprised of the L and R-sets. The displacement set notation for this is:

$$U_A = \begin{Bmatrix} U_L \\ U_R \end{Bmatrix} \quad \text{0-5}$$

The R-set are the degrees of freedom at the boundary of the substructure where it connects to other substructures. The R-set is defined by the user via the SUPORT Bulk Data entry. In CB analysis, the R-set are constrained to zero for the purposes of calculating the fixed interface modal properties of the substructure and the R-set is used in determining the boundary stiffness and mass. As shown in Reference 11, these matrices provide the overall properties of the substructure in terms of modal and boundary degrees of freedom which are typically a much smaller subset of the physical degrees of freedom in the R and L-sets combined.

Following elimination of the R-set degrees of freedom, MYSTRAN is set to solve for the displacements of the L-set.

If there is no R-set defined by the user, then the L-set is equivalent to the A-set. If there is no O-set defined by the user, then the A-set is equivalent to the F-set. If there is no S-set, the F-set is equivalent to the N-set (although the stiffness matrix for this would be singular since no boundary constraints would exist). If there is no M-set then the N-set is equivalent to the G-set.

The mutually exclusive sets are the M-set, the S-set, the O-set and the R-set and the L-set. The G-set consists of all of these.

Appendix B has a complete mathematical discussion on the details of how the G-set is reduced to the A-set.

When the degree of freedom (DOF) tables are printed out (if requested by the user through the PARAM PRTSET and PARAM PRTDOF Bulk Data entries), the S-set is broken down into the several sub-sets. Below is a summary of all of the columns of the DOF table:

- G: All DOF’s in the model
- M: All DOF's multi-point constrained
- N: G – M (or F + S)
- SA: DOF’s SPC’d when AUTOSPC = Y
- SB: DOF’s SPC’d to zero via Bulk Data SPC, SPC1 Bulk Data entries (requested in Case Control)

- SE: DOF’s SPC’d to nonzero values (enforced displacements) (requested in Case Control)
- SG: DOF’s SPC’d to zero values that are identified in field 8 of the Bulk data GRID entry
- SZ: SA + SB + SG (all zero value SPC’s)
- S: All DOF’s single-point constrained (S = SA + SB + SG + SE)
- F: N – S (or A + O)
- O: All DOF’s statically omitted
- A: F – O (or L + R)
- R: All DOF's defined via Bulk Data SUPORT entries
- L: A – R

4 MYSTRAN SOLUTION TYPES

MYSTRAN currently has 5 solution types:

- SOL = 101 for static analysis
- SOL = 103 for natural frequency (normal modes) – Eigen solution
- SOL = 104 for differential stiffness
- SOL = 105 for Eigen solution buckling
- SOL = 31 for Craig-Bampton (CB) model generation.

The CB model generation is a new analysis type and is discussed in more detail.

4.1 Static Analysis

SOL 101 or, alternately, SOL STATICS is for static solution of a model with constant loads. It is the same as statics for NASTRAN and uses all of the features described above for model description, load definition, etc. Output for displacements, applied loads, constraint forces, grid point force balance, element forces and stresses are available. In addition, output of matrices and debug information is available.

4.2 Natural Frequency (Normal Modes)

SOL 3 or, alternately, SOL MODES, or SOL MODAL or SOL NORMAL MODES is for eigenvalue analyses of a model. It is the same as the eigenvalue analysis type of solution in NASTRAN. All of the model features in statics (with a few exceptions such as loads and enforced displacements) are available. Besides the eigenvalues themselves, output for displacements, constraint forces, element forces and stresses are available. Also, output of modal participation factors and modal effective mass is available. In addition, output of matrices and debug information is available.

4.3 Differential Stiffness

SOL 4 or, alternately, SOL DIFFEREN is for static analysis with the same differential stiffness that would also be used in linear static buckling analysis.

4.4 Buckling

SOL 105 or, alternately, SOL BUCKLING is for linear static buckling. A differential stiffness matrix is calculated and added to the normal linear elastic stiffness matrix. This solution requires two subcases: an initial static load of some value (generally a unit load) simulating the buckling load followed by a subcase with an eigenvalue extraction method. The eigenvalue found is a multiplier of the load applied in the first subcase in order to get the buckling load.

Shell buckling is partially implemented (See Section 3.2).

4.5 Craig-Bampton model generation

SOL 31 or SOL GEN CB MODEL is for Craig-Bampton (CB) model generation and is a new feature in MYSTRAN that is not a direct solution type available in NASTRAN. It involves reduction of a large model, originally in terms of physical degrees of freedom (DOF's) at all grid locations, to one in which the DOF's are a smaller subset using modal DOF's for fixed base modes to describe the vibration characteristics of the model and physical DOF's for the boundaries between substructures. Appendix D gives a detailed description of CB analyses including references to the original work by those that pioneered the technique and also includes an example problem. Using NASTRAN to get CB models is a more cumbersome technique than the direct one in MYSTRAN in that it employs a rather complicated (and in some areas arcane) DMAP (or Direct Matrix Abstraction Programming) program.

Sometimes called dynamic substructure analysis, CB analysis is often used in cases where a very large model is broken into smaller pieces each of which is generally a defined substructure. An example would be a spacecraft with several scientific instrument and appendages. Each of these individual pieces may come from different analytical groups and may be needed in a combined analysis. Each of the groups developing models of their substructure would deliver an analytical

CB model of their hardware and the systems contractor would assemble these for a combined structural dynamic analysis.

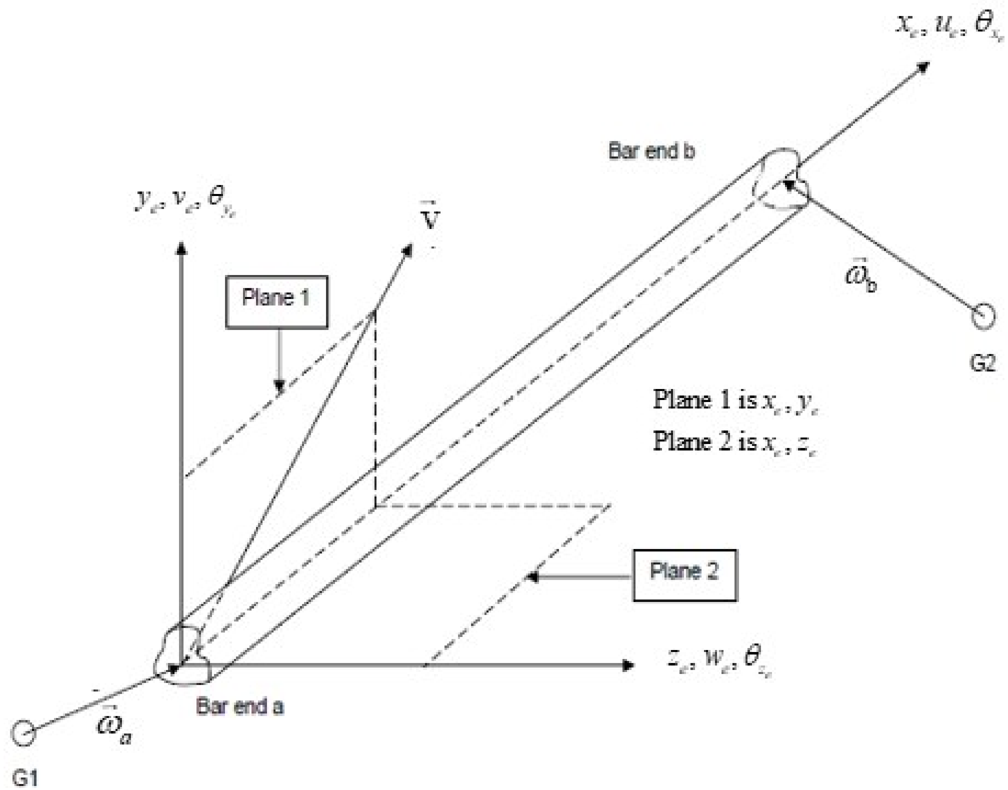
The input to a SOL 31 CB model generation analysis for a single substructure is the same as that for a standard eigenvalue analysis with a few additions. The biggest difference is in defining the boundary DOF's for the substructure where it connects to other substructures. The boundaries are defined using Bulk Data SUPORT entries which key MYSTRAN to put these DOF's into the R-set. The fixed base modes of the substructure are those for which the R-set is constrained to zero. However, the model delivered to the system contractor for integration cannot be grounded at these DOF's since they will be active in the combined analysis. Thus, the CB solution takes into account that these boundary DOF's are free in the matrices that define the CB model even though they were temporarily grounded to obtain the fixed mode properties of the substructure. It should be mentioned that the boundary DOF's defined via the SUPORT Bulk Data entry must be the only DOF's constrained to zero motion except for those removed to avoid singularities.

The output from the CB analysis of a single substructure is quite different than those from a normal eigenvalue analysis except that the fixed base modal frequencies and mode shapes can be output and are the same as those that would result from a SOL 3 eigenvalue analysis with the R-set constrained to zero motion. The rest of the available outputs are generally for Output Transformation Matrices (OTM's) and other CB model matrices needed by the systems contractor in performing the combined analysis. Appendix D discusses all of the available OTM's from a SOL 31 CB model generation analysis. However, the following is a general idea of how to obtain CB model data from MYSTRAN:

- For any of the matrices listed in Table 9.5 of Appendix B (including Net C.G. loads and Interface Force LTM) use the OUTPUT4 entry in Executive Control. Theses are written to disk files with the names *filename*.ext where ext (file extension) is OPi with i=1,2,3,4,5,6,7 as defined by the user in the OUTPUT4 command.
- For displacement, acceleration, element force, element stress, MPC forces, use normal Case Control requests (including defining sets of grids/elements for output). These OTM's are output in the normal F06 output file and also onto disk files with the extension OP8 (for grid related OTM's) and extension OP9 (for element related OTM's. Text files (extensions OT8 and OT9) have explanations of the rows of the OTM's written to the OP8 and OP9 files.

In addition to creating CB models, MYSTRAN can synthesize CB models, along with an optional finite element model, into a systems model for eigenvalue analyses. This feature is demonstrated in: ???

Figure 4-1: Bar Element Geometry and Coordinate System



$x_e =$ Neutral axis of the bar (positive direction is from end a to end b

$\vec{v} =$ Vector specified on CBAR or BAROR entry used in defining Plane 1

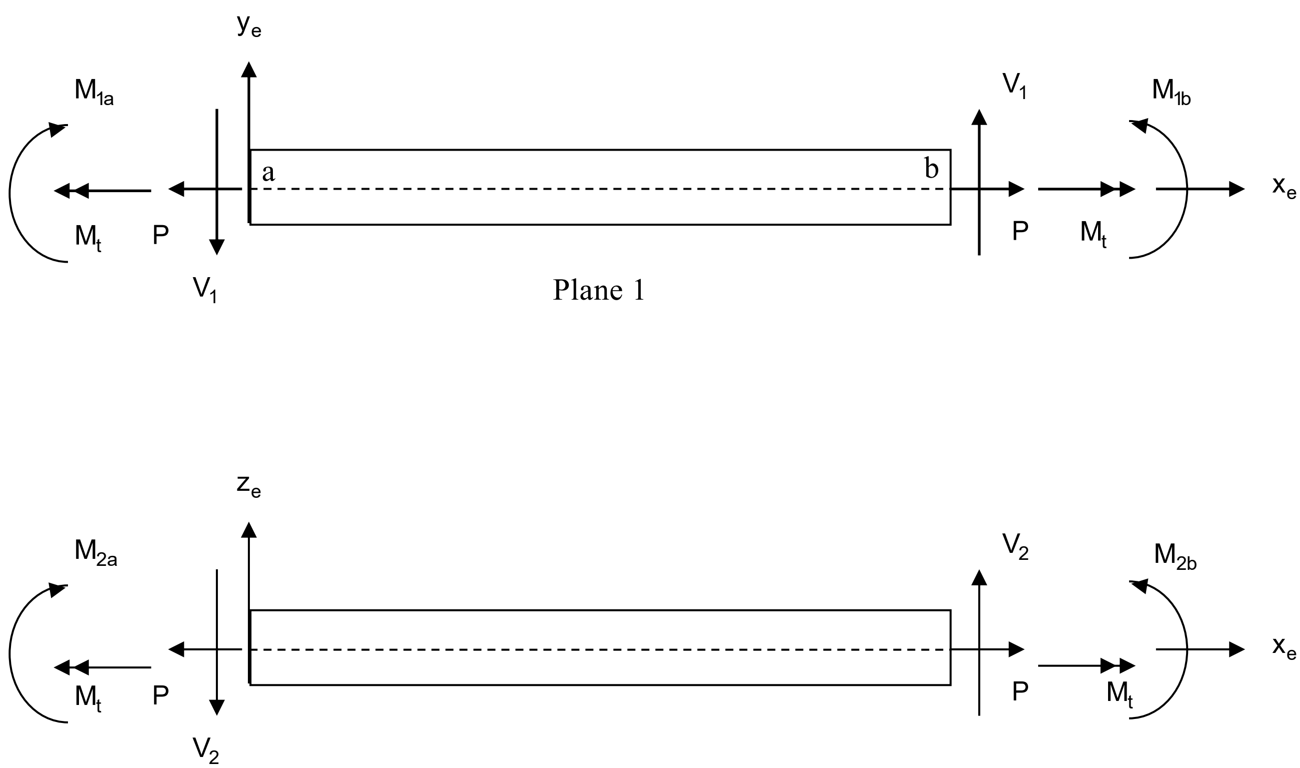
$z_e =$ Axis in the plane defined by x_e and the vector cross product $x_e \otimes \vec{v}$

$y_e =$ Axis in the direction of the vector cross product $z_e \otimes x_e$

$\vec{\omega}_a =$ Vector from grid G1 on the CBAR entry to end a of the Bar (the offset at end a)

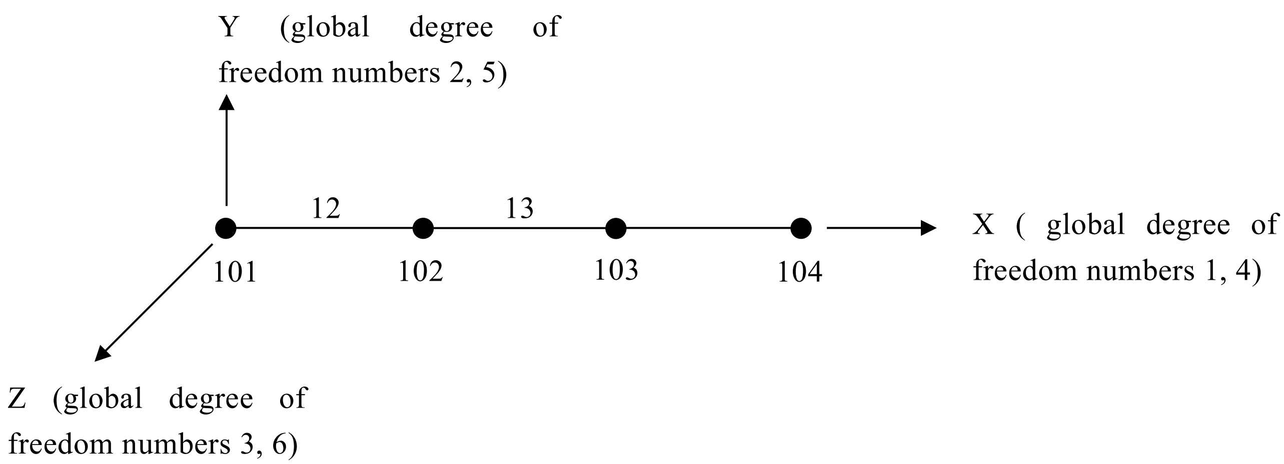
$\vec{\omega}_b =$ Vector from grid G2 on the CBAR entry to end b of the Bar (the offset at end b)

Figure 4-2: Bar Element Forces



P = Axial Load
M_t = Torque
V_1 = Shear in Plane 1
V_2 = Shear in Plane 2
M_{1a} = Bending Moment in Plane 1 at end a
M_{1b} = Bending Moment in Plane 1 at end b
M_{2a} = Bending Moment in Plane 2 at end a
M_{2b} = Bending Moment in Plane 2 at end b

Figure 4-3: Example of MYSTRAN Development of Equations for a Rigid Element



Grid ID's are: 101 - 106

Element ID's are: 12 - 14 (12 and 14 elastic and 13 rigid)

Global displacement system is the X, Y, Z basic system shown

Define:

u_i = displ of grid i in the X direction, θ_{x_i} = rotation of grid i about X axis

v_i = displ of grid i in the Y direction, θ_{y_i} = rotation of grid i about Y axis

w_i = displ of grid i in the Z direction, θ_{z_i} = rotation of grid i about Z axis

X_i = X coordinate of grid i

Assume that rigid element 13 is rigid only in the X - Y plane.

Take grid 103, degrees of freedom 1,2,6 as dependent. Use grid 102 as independent.

The linear equations that specify the dependence of grid 103 on grid 102 in the X - Y plane are:

$$u_{103} = u_{102}$$
$$v_{103} = v_{102} + (X_{103} - X_{102})\theta_{z_{102}}$$
$$\theta_{z_{103}} = \theta_{z_{102}}$$

5 DETAILED DESCRIPTION OF INPUT DATA

The input entries for the Executive Control, Case Control and Bulk Data Sections are described in detail in the next three sections. In all of the sections, an entry with a $ sign in column 1 is considered as a comment and is ignored. In addition, any blank entry is ignored. All other entries must be in upper case. Appendix A contains a sample problem input/output.

5.1 File Management

As mentioned earlier, the input data file consists of 3 sections: Executive Control, Case Control and Bulk Data. In order to make the most efficient use of resources, each of these can contain requests to include some defined file to be part (or all) of that portion of the input data file. This is accomplished through the use of an INCLUDE entry whose format is:

INCLUDE '*filename*'

Where *filename* is the name of a file to include at the location where the INCLUDE entry exists. The INCLUDE entries can be used in any or all of the 3 sections of the input data file. In addition, multiple INCLUDE entries in any section are permitted. The quotes around *filename* are recommended but not required.

5.2 Executive Control

The Executive Control Section consists of only a few entries. Most are free field; that is they can begin in any column and the parts of an entry may be separated by any amount of columns within the confines of the 80 column physical entry. In addition, the fields of an entry may be delimited by tabs, as well as a white space. Some of the entries are required and some are not required but are recognized. Other entries are ignored with a warning message printed in the output. Any requirements on the order of the entries in the Executive Control Section are noted.

With the CHKPNT/RESTART feature, users may restart a previously run job to get additional outputs. In a restart the Bulk Data must remain the same except for a few PARAM and DEBUG entries. Case Control requests for additional displacements, element forces, stresses, etc. will be processed.

Table 5.1 Executive Control Entries required and/or recognized by MYSTRAN

Entry	Required (Y/N)	Format	Description
ID	N	Free Field	If input, it is generally the first entry in the Exec Control Section.
IN4	N		Defines a file containing element stiffness, mass and other data for a CUSERIN element.
APP	N	Free Field	An entry of APP DISP is common if this entry is included.
CHKPNT	Y/N	Free Field	Required if the user expects to restart the current job, at a later date, to obtain additional outputs.
DEBUG	N	Fields of 8 chars like Bulk Data	These are the same as the Bulk Data DEBUG entries and are allowed here since some DEBUG values need to be used prior to reading the Bulk Data.
OUTPUT4	N	Free Field	Requests for CB matrices to be written to unformatted files in the same format as NASTRAN uses. An example is shown below along with the allowable matrices that can be output.
PARTN	N	Free Field	Requests to partition a previously defined OUTPUT4 matrix.
RESTART	Y/N	Free Field	Required only if the current job is a restart of an earlier job in which the CHKPNT entry was present. The file name (w/ ext) of the CHKPNT'd original run must follow the command RESTART.
SOL	Y	Free Field	SOL entry must have a value that designates what kind of problem this is: (1) SOL 1 or SOL STATICS designates the job as a statics problem. (2) SOL 3 or SOL = MODAL or SOL MODES or SOL NORMAL MODES for eigenvalues. (3) SOL 31 or SOL GEN CB MODEL for Craig-Bampton (CB) model generation. (4) SOL 5[2] or SOL BUCKLING for linear static buckling (5) SOL 4 or SOL DIFFEREN for static analysis with the same differential stiffness that is used in static buckling analyses.
TIME	N	Free Field	TIME n, where n is the job estimated time in minutes.
CEND	Y	Free Field	The CEND entry has no other input required. It must be the last entry in the Exec Control Section.

[2] As of 1/1/2019 only the BAR element is coded for buckling (SOL 5) or differential stiffness (SOL 4)

5.2.1 IN4 Exec Control command

The Exec Control command IN4 specifies binary files (NASTRAN INPUTT4 format) which contain the element matrices needed for CUSERIN Bulk Data element definition. The IN4 command has the following format:

IN4 i *filename*

Where i is the ID of the file and is what must appear in field 3 of the Bulk Data PUSERIN property entry for the CUSERIN element. *filename* is the name of the file that contains the matrices specified on the PUSERIN entry for the element. *filename* must contain the full path unless the file is in the current path where the program is being executed. An example is: **IN4 100 cb1_example1.OP1**

5.2.2 OUTPUT4 and PARTN Exec Control commands

MYSTRAN allows output of selected matrices to binary files in the OUTPUT4 format that is the same as that currently used by NASTRAN. The form of the OUTPUT4 command is:

OUTPUT4 MAT1,MAT2,MAT3,MAT4,MAT5//ITAPE/IUNIT $

From 1 to 5 matrices can be output per OUTPUT4 command. All 4 commas must be present even if fewer than 5 matrices are requested. The // followed by ITAPE value (must be 0 to -3 but is currently not used) must also be present. The final / followed by a file unit number (can be 21-27) is also required. A trailing $ can exist but is not required. If present, it signifies the end of data read for the OUTPUT4 command.

These OUTPUT4 matrices can be partitioned, in some cases, using an Exec Control PARTN command. The resulting partitioned matrix will be the one output to the OUTPUT4 binary file. The partitioning vectors that define which columns and rows to partition from the original OUTPUT4 matrix are defined on Bulk Data PARVEC and PARVEC1 entries. These Bulk Data partitioning vector entries give the grid and component pairs of the columns and rows to partition. As such, the partitioning can only be done on OUTPUT4 matrices that have columns and/or rows that are part of a normal displacement set (the G-set, M-set, etc.). See section 3.6, “Displacement set notation”, for a definition of all of the displacement sets. The general form for the PARTN command for MYSTRAN is:

PARTN MAT, CP, RP / $

where MAT is an OUTPUT4 matrix previously requested for OUTPUT4 output and CP and RP are column and row partitioning vectors defined in the Bulk data using PARVEC and/or PARVEC1 Bulk Data entries.

If the input file for a MYSTRAN run is *filename*.DAT, the binary OUTPUT4 file names are *filename*.OPi where i=1,7 (corresponding to units 21-27 used as values for UNIT in the OUTPUT4 command). The format in which these files are written is the same as that for the NASTRAN OUTPUT4 matrices.

The table on the following page shows the matrices that are currently eligible for OUTPUT4 output. Note that there is a correspondence between MYSTRAN and NASTRAN matrix names. The OUTPUT4 commands can use either name as desired by the user. All matrix names must be no more than16 characters long. An example of the use of the Exec Control commands OUTPUT4 and PARTN is given following the table.

Table 5.2: Matrices that can be written to OUTPUT4 files (and the correspondence between MYSTRAN matrix names, NASTRAN names and CB Equation Variables)

	MYSTRAN Matrix Name (OUTPUT4 matrices)	NASTRAN DMAP Name	CB equation variable in Appendix D (where applicable)	Matrix size[1]	Partition rows and/or cols
1	CG_LTM		$[LTM11_{6r} \quad LTM12_{6N} \quad 0]$	6x(2R+N)	
2	DLR	DM	D_{LR}	LxR	rows and cols
3	EIGEN_VAL	LAMA	Ω_{NN}^2	NxN	
4	EIGEN_VEC	PHIG	Φ_{GN} , (Φ_{LN} with rows expanded to G-set)	GxN	rows
5	GEN_MASS	MI	m_{NN}	Nx1 vector of diag. terms	
6	IF_LTM		$[LTM21_{RR} \quad LTM22_{RN} \quad LTM23_{RR}]$	Rx(2R+N)	rows
7	KAA	KAA	K_{AA}	AxA	rows and cols
8	KGG	KGG	K_{GG}	GxG	rows and cols
9	KLL	KLL	K_{LL}	LxL	rows and cols
10	KRL	KLR(t)	K_{LR}	LxR	rows and cols
11	KRR	KRR	K_{RR}	RxR	rows and cols
12	KRRcb	KBB	$k_{RR} = K_{RR} + K_{LR}^T D_{LR}$	RxR	rows and cols
13	KXX	KRRGN	K_{XX}	(R+N)x(R+N)	
14	LTM	LTM	CG_LTM and IF_LTM merged	(6+R)x(2R+N)	
15	MCG	RBMCG	m_{cg}	6x6	
16	MEFFMASS		Modal effective mass	Nx6	
17	MPFACTOR		Modal participation factors	Nx6 or NxR	
18	MAA		M_{AA}	AxA	rows and cols
19	MGG		M_{GG}	GxG	rows and cols
20	MLL	MLL	M_{LL}	LxL	rows and cols
21	MRL	MRL	M_{RL}	RxL	rows and cols

22	MRN		$m_{RN} = m_{NR}^T$	RxN	rows
23	MRR	MRR	M_{RR}	RxR	rows and cols
24	MRRcb	MBB	$m_{RR} = M_{RR} + M_{LR}^T D_{LR} + (M_{LR}^T D_{LR})^T + D_{LR}^T M_{LL} D_{LR}$	RxR	rows and cols
25	MXX	MRRGN	$M_{XX} = \begin{bmatrix} m_{RR} & m_{NR}^T \\ m_{NR} & m_{NN} \end{bmatrix}$	(R+N)x(R+N)	
26	PA		(A-set static reduced loads - only used in statics)		Rows
27	PG		(G-set static loads - only used in statics)		Rows
28	PL		(L-set static reduced loads - only used in statics)		rows
29	PHIXG	PHIXG	Ψ_{AX} , (Ψ_{AX} with rows expanded to G-set)	Gx(R+N)	rows
30	PHIZG		The G-set displacement transformation matrix is written out in the F06 file under "C B D I S P L A C E M E N T O T M"	Gx(2R+N)	rows
31	RBM0		Rigid body mass matrix relative to the basic origin	6x6	
32	TR6_0	RBR	T_{R6}: rigid body displacement matrix for R-set relative to the model basic coordinate system	Rx6	rows
33	TR6_CG	RBRCG	T_{R6}: rigid body displacement matrix for R-set relative to the model CG	Rx6	rows

Note: (t) indicates matrix transposition

Example of OUTPUT4 request in Exec Control

Format:

OUTPUT4 MAT1, MAT2, MAT3, MAT4, MAT5 // ITAPE / IUNIT $

Example:

OUTPUT4 PHIZG, KRRcb,,, // -1 / 22 $

a) The OUTPUT4 entry is free-field (except that there can be no blank characters in any of the names, including OUTPUT4).

b) MATi can be any of the matrix names in the OUTPUT4 table above. There can be 1 to 5 matrices in any OUTPUT4 request but all 4 commas must be present. If there is a name for the matrix in the column "NASTRAN DMAP Name", that name can be used in place of the MYSTRAN Matrix Name for OUTPUT4 purposes

c) ITAPE (using NASTRAN notation) should be: -3 $\leq ITAPE \leq 0$ (but is currently not used in MYSTRAN),

d) IUNIT must be: $21 \leq IUNIT \leq 28$. Any number of the OUTPUT4 matrices can be sent to one IUNIT and more than one IUNIT can be used in one Exec Control section,

e) The / characters must be present,

f) Anything after the $ character (if present) is ignored.

Example of PARTN request in Exec Control

Format:

PARTN MAT, CP, RP/ $

CP is the column partitioning vector and RP is the row partitioning vector

Example:

OUTPUT4 PHIZG,, RVEC1 / $

a) The PARTN entry is free-field (except that there can be no blank characters in any of the names, including PARTN).

b) MAT is the name of the matrix to partition (with restrictions noted in Table 6-1 regarding whether rows and or column of this matrix are available for partitioning).

c) RP (RVEC1 in the example) is the row partition vector which must be specified using either the PARVEC or PARVEC1 Bulk Data entry.

d) The PARTN entry must have 2 and only 2 commas. Note that in the example above that CP is not specified (since PHIZG is only available for row partitioning) but the 2nd comma is present.

e) The PARTN entry for MAT must follow (but not necessarily immediately) the mandatory OUTPUT4 request for it.

6 CASE CONTROL

The Case Control Section performs several functions outlined below. The entries for each of the major purposes are enumerated below. A detailed explanation of each is contained in the following section. A BEGIN BULK entry is considered as the last, and mandatory, entry in the Case Control Section. In addition, the fields of an entry may be delimited by tabs, as well as a white space.

- The following entries specify the titles that will be printed in the output file, none of which are required:

TITLE	Specifies a line of text to be printed in the output file
SUBTITLE	Specifies a 2nd line of text to be printed in the output file
LABEL	Specifies a 3rd line of text to be printed in the output file

- The following entries select items from the Bulk data to be used in the current job (loads, constraints, temperature sets, eigenvalue extraction ID):

ENFORCED	Specifies a file containing all grid displacements (all translations and rotations for all grids). With this command, users can run cases in which all displacements are known (as for example from test data) and can request any outputs based on these displacements.
LOAD	Selects FORCE, MOMENT, GRAV, PLOAD2, PLOAD4, RFORCE and LOAD sets from the Bulk Data Section that define loads for a statics solution.
METH	Selects an eigenvalue extraction set from the Bulk Data for a eigenvalue solution.
SPC	Selects SPC, SPC1 from the Bulk Data Section that define single point constraints (including enforced displacements) for the current job.
MPC	Selects MPC entries from the Bulk Data Section that define multi-point constraints for the current job.
TEMP	Selects TEMP, TEMPD and TEMPP1 sets from the Bulk Data Section that define temperature loads for a statics solution.

- The following entries define output requests:

ACCEL	Requests output of accelerations.
DISPL	Requests output of displacements.
ECHO	Requests form of the input file echoed to the output file.
ELDATA	Requests element matrix generation output to the BUG file[3].
ELFORCE	Requests output of element engineering and/or node forces.
GPFORCE	Requests output of grid point force balance showing all of the forces acting on a grid point and checking equilibrium of those forces.
MEFFMASS	Requests output of modal effective masses in eigenvalue analyses.
MPCFORCE	Requests output of multi point forces of constraint (due to MPC's as well as rigid elements).
MPFACTOR	Requests output of modal participation factors in eigenvalue analyses.
OLOAD	Requests output of applied loads.
SET	Specifies sets that define grid points and elements for which output is desired.
SPCFORCE	Requests output of single point forces of constraint.
STRESS	Requests output of element stresses.
STRAIN	Requests output of element strains for shell and solid elements

- The following entry defines subcases for which solutions will be calculated in static analyses (SOL 1):

---

[3] The various files (output and scratch) generated by MYSTRAN are described in a later section. BUG is the extension of one of those files.

SUBCASE — A entry that indicates that the following entries (until another SUBCASE entry is encountered) define the conditions for one solution in the current job. A separate subcase must be used for each loading condition for which a solution is desired.

6.1 Detailed Description of Case Control Entries

The following pages give the details for each of the Case Control Section entries listed above. The format of each is free field with the following conventions:

- Upper case letters must be entered as shown.
- Lower case letters indicate that a substitution must be made.
- Parentheses shown must be entered.
- Braces { } indicate that a choice, from the items listed, must be made.
- Brackets [ ] indicate that the terms enclosed may be omitted, if desired. Braces within brackets indicate that if terms within the brackets are input a choice must be made of the portion within the braces.
- Underlined values are the default values.

In addition, some of the entries have an acceptable abbreviation of the entry name. For example, the entry requesting displacement output can be DISPLACEMENT or at least the first four letters of the name. This is noted in the detailed description with brackets. Thus, DISP[LACEMENT] indicates the acceptable forms of this Case Control entry.

6.2 Output File Types

Output files are either in ASCII or binary. The following output file types are possible:

- F06 – (ASCII). This is a common Nastran file type (.f06), but does not have a standard format. The format may be different for different commercial versions of Nastran and may even be different between different versions for the same commercial version. The MYSTRAN format is similar to other F06 formats, but it is not exactly the same as any of them. A custom tool, called F06CSV, can parse a F06 file (both for MYSTRAN and some commercial Nastran F06 files). This tool can create a customized output of the data in a format that is convenient for processing the data: https://github.com/ClassicalFEA/nastools
- OP2 (binary). This is common Nastran file type (.op2) that is read by some post-processors. A MYSTRAN OP2 is expected to be able to be read by Patran and Femap, but not by Apex. It can be also be read by the open-source programs pyNastran and M3D. The OP2 is fully supported in MYSTRAN with the following exceptions:
    - CTRIA3K and CQUAD4K
    - Corner stresses/strains for quad elements
- PUNCH (ASCII). As of 2025-09-01, the state of the MYSTRAN punch file is unknown. It may be the output from a Craig-Bampton solution. However, it is not equivalent to the typical Nastran punch file that has outputs for SOL 101, SOL 103, and SOL 105.
- ANSWER File (ASCII). Having an extension of .ans, this is a MYSTRAN specific file. The file is similar to the F06, but is more compact. Because of this, it has sometimes been used to interrogate results. It can be called via "PARAM, PRTANS, YES".
- NEU (ASCII). The Neutral file (.neu) is a legacy file option. The version used in MYSTRAN may be version 7.1. It can be called via "PARAM, PRTNEU, YES".
- BUG
- F04
- Other

For examples of how to request the different files, see the DISPLACEMENT and STRAIN entries. Also, see the following parameters, which can be used as overrides. Most notably are the PRTALL and FILES params (equivalent). If present, and set to YES, ("PARAM, PRTALL, YES" or "PARAM, FILES, YES"), then the F06, OP2, ANS, and NEU will always be created.

- PRTALL and FILES (equivalent)

- PRTF06
- PRTOP2
- PRTANS
- PRTNEU

6.3 BEGIN BULK

Description:

Indicates the end of the Case Control section.

Format:

BEGIN BULK

6.4 ACCELERATION

Description:

Requests output of grid point accelerations in the global coordinate system for selected grids. For Craig-Bampton model generation, the output is of the columns of the acceleration transfer matrix (ATM).

Format:

ACCE[LERATION][(<u>PRINT</u>, PLOT)] = $\left\{ \begin{matrix} \text{ALL} \\ n \\ \text{NONE} \end{matrix} \right\}$

Examples:

See the DISPLACEMENT card (Section 6.5)

Options:

See the DISPLACEMENT card (Section 6.5)

Remarks:

See the DISPLACEMENT card (Section 6.5)

6.5 DISPLACEMENT

Description:

Requests output of grid point displacements in the global coordinate system for selected grids. For an eigenvalue analyses, the output is the eigenvectors.

Format:

$$\text{DISP[LACEMENT][(}\underline{\text{PRINT}}\text{, PLOT)] = } \begin{Bmatrix} \text{ALL} \\ n \\ \text{NONE} \end{Bmatrix}$$

Examples:

DISPLACEMENT = ALL	requests all relevant output to the F06 file (ASCII). Default (no use of options).
DISP = 45	requests relevant output of Case Control entry. SET 45 goes to the F06 file.
DISP(PRINT, PLOT) = ALL	requests a F06 (ASCII) and OP2 file (binary) for all relevant output.
DISP(PLOT) = ALL	requests an OP2 file (binary) for all relevant output. A F06 file will not be created.

Options:

Option	Meaning
ALL	All relevant output is requested.
n	ID of a SET Case Control entry previously defined. Relevant output defined by SET n will be requested. Integer > 0, no default value.
NONE	No output.
PRINT	The output will go to the *filename*.F06 file (ASCII). This file can be parsed with a utility created by the MYSTRAN project (See Remark 8).
PLOT	The output will go to the *filename*.OP2 file (Binary).

Remarks:

1. Optional entries are surrounded by brackets.
2. If no output file is requested, the only output file will be the F06 (the default output is PRINT, which creates the F06).
3. The PUNCH option may be used in the same manner as PRINT and PLOT, which will create a .PCH file (ASCII). However, this punch file appears to be related to a Craig-Bampton solution and NOT a typical Nastran punch file. It is NOT active for DISP (although it is presented in this entry for reference), but it may be active for ACCELERATION and other entries for a C-B solution. The MYSTRAN punch file requires further investigation.
4. The “PARAM, POST” entry has no effect on whether or not the OP2 is created and has no effect on the output requests. This is unlike some other Nastran versions.
5. NONE is used to override an overall output request made above the SUBCASE level.
6. The first four letters are the shorthand version of the full name. For example, both DISPLACMENT and DISP are acceptable.
7. A Neutral file (.NEU) may be requested by placing “PARAM, PRTNEU, YES” in the bulk data section of the deck.
8. An Answer file (.ANS) file may be requested by placing “PARAM, PRTANS, YES” in the bulk data section of the deck.
9. The parameters PRTALL, FILES, PRTF06, PRTOP2 may be used to override the output requests.
10. The MYSTRAN project developed a utility that parses a F06 file. This tool can create a customized output of the data in a format that is convenient for processing the data: https://github.com/ClassicalFEA/nastools

6.6 ECHO

Description:

Requests that the input data file be echoed in the output file.

Format:

$$\text{ECHO} = \left\{ \begin{array}{c} \text{NONE} \\ \underline{\text{UNSORT}} \end{array} \right\}$$

Examples:

ECHO = NONE

Options:

Option	Meaning
NONE	No echo of the input data file will be in the output file.
UNSORT	The echo of the data file in the output will be in the same entry order that the input data file is in.

6.7 ELDATA

Description:

Requests output of element data from the element matrix generation subroutines for selected elements. The data is written to files separate from the standard output file. Description of the data items that can be output is given in the table below. The output files that the data is written to are described in the MYSTRAN Installation and Run Manual. This card needs a review/update.

Format:

$$\text{ELDA[TA]}\left\{(m\left[\begin{Bmatrix}\text{,}\underline{\text{PRINT}}\\ \text{,FIJFIL}\\ \text{,BOTH}\end{Bmatrix}\right])\right\}=\begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

Examples:

ELDATA(4,BOTH) = 2	print to .BUG output file,and write to unformatted file,elem data item 4 for SET 2 elems.
ELDATA(3) = 9	print to .BUG file the output of elem data item 3 for elems included in SET 9.
ELDATA(2,FIJFIL) = ALL	write elem data item 2 for all elems to unformatted file.

Options:

Option	Meaning
m	Defines which element data items are to be output (see table below)
ALL	Data items m for all elements will be output.
n	ID of a SET Case Control entry previously defined. Element data for item m defined by SET n will be output. Integer > 0, no default value.
NONE	No element data items will be output.

Remarks:

1. NONE is used to override an overall output request made above the SUBCASE level.
2. See table below for a description of the data items that can be output.

Element Data Items Output for ELDATA Case Control Entry

m	Data Item(s) Output	Printed to Text File With Extension	Written To Unformatted File With Extension
0	Actual and internal grid points and their basic coordinates	BUG	
1	Array of element property data. Array of element material data. Bar element v vector in basic coordinates. Bar pin flag data. Bar offsets. TE coord transform matrix (transforms a vector from basic to local elem coords). Actual and internal grid points and local element coordinates.	BUG	
2	Element thermal and pressure loads in local element coordinates.	BUG	F21
3	Element mass matrix in local element coordinates.	BUG	F22
4	Element stiffness matrix in local element coordinates.	BUG	F23
5	Element stress and strain recovery matrices in local element coordinates.	BUG	F24
6	Element grid point displacements and loads. The coordinate system will be the one defined by Bulk data PARAM ELFORCEN.	BUG	F25
7	Data on isoparametric element shape functions and Jacobian matrices	BUG	
8	Isoparametric element shape functions	BUG	
9	Check isoparametric element strain-displ matrices for rigid body motion and constant strain. **NOTE: as of 03/07/2020 the check on strain-displacement matrices using Case Control ELDATA(9) suspended until an error in that calculation is found. This can be overridden with Bulk Data entry: DEBUG, 202, 1**	BUG	

Notes:

1. The filename will be the same as the input data file but with the extension given in the table.
2. See Appendix B for a description of some of these matrices that can be output.

6.8 ENFORCED

Description:

Requests a run in which the displacements (all 3 translations and rotations) are specified in a file whose name is given as part of this command. The situation in which this might be useful is one in which all grid displacements are known from test data and the user would like to get other outputs (e.g. stresses) due to these displacements.

Format:

ENFORCED = filename

Examples:

ENFORCE = Case1-displacements-rotations.txt

Remarks:

1. filename is a text file with NGRID+1 records (where NGRID are the number of grids in the model).

   a) Record 1 is a comment line.

   b) Records 2 through NGRID+1 have the following in CSV format for each grid:

   grid ID, T1, T2, T3, R1, R2, R3

2. An example of the ENFORCED file for 2 grids is:

   Displacements and rotations for model A with 3 grids (101, 102)

   101, 1.23456D-02, 2.34567D-02, 3.45678D-03, 0.00000D+00, 4.56789D-04, 3.67890D-05

   102, 6.54321D-02, 7.65432D-03, 8.76543D-03, 9.87654D-05, 5.43210D-06, 0.00000D-05

3. All grids must have all 6 components specified in the file (i.e. all DOF's must be in the S-set).

4. Any Case Control requests for SPC's or MPC's will result in an error.

5. Any Bulk Data ASET or OMIT entries will result in an error.

6.9 FORCE

Description:

Requests output of nodal or engineering forces for selected elements. This card needs review/update (specifically wrt ENGR vs NODE)

Format:

$$\text{FORC[E][(}\underline{\text{PRINT}}\text{, PLOT)]} = \begin{Bmatrix} \text{ALL} \\ n \\ \text{NONE} \end{Bmatrix}$$

Examples:

FORCE = ALL	requests output of element engineering forces for all elements
FORC(NODE) = 125	requests output of element nodal forces for elements included in SET 125

Options:

Option	Meaning
ALL	Element forces for all elements in the model will be output.
n	ID of a SET Case Control entry previously defined. Element forces for the elements defined by SET n will be output. Integer > 0, no default value.
NONE	No accelerations will be output.
PRINT	The element forces will go to the *filename*.F06 file
PLOT	The output will go to the *filename*.OP2 file
PUNCH	The output will go to the *filename*.PCH file

Remarks:

1. NONE is used to override an overall output request made above the SUBCASE level.

2. The forces can be output in local element, basic, or global coordinates. See Bulk Data PARAM ELFORCEN entry.

6.3.1.6 ELFORCE

Description:

Requests output of nodal or engineering forces for selected elements.

Format:

$$\text{ELFO[RCE]} \begin{bmatrix} \text{PLOT} \\ \text{PRINT} \\ \text{PUNCH} \end{bmatrix} \begin{bmatrix} \underline{\text{ENGR}} \\ \text{(NODE)} \\ \text{(BOTH)} \end{bmatrix} = \begin{Bmatrix} \text{ALL} \\ n \\ \text{NONE} \end{Bmatrix}$$

Examples:

ELFORCE = ALL (requests output of element engineering forces for all elements)
ELFO(NODE) = 125 (requests output of element nodal forces for elements included in SET 125)

Options:

Option	Meaning
ALL	Element forces for all elements in the model will be output.
n	ID of a SET Case Control entry previously defined. Element forces for the elements defined by SET n will be output. Integer > 0, no default value.
NONE	No element forces will be output.

6.10 GPFORCES

Description:

Requests output of grid point force balance in the global coordinate system for selected grids.

Format:

$$\text{GPFO[RCES][(}\underline{\text{PRINT}}\text{, PLOT)] = } \left\{ \begin{matrix} \text{ALL} \\ n \\ \text{NONE} \end{matrix} \right\}$$

Examples:

See the DISPLACEMENT card (Section 6.5)

Options:

See the DISPLACEMENT card (Section 6.5)

Remarks:

See the DISPLACEMENT card (Section 6.5)

6.11 LABEL

Description:

Specifies a third text line to be printed in the output file.

Format:

LABE[L] = [optional text material up, and including, column 80]

Remarks:

1. This line of text will be printed in the output file and can be different for each subcase.

6.12 LOAD

Description:

Indicates what applied loads (identified in the Bulk Data) are to be used for a solution.

Format:

LOAD = *n*

Examples:

LOAD = 98	Requests load set 98 be used

Options:

Option	Meaning
n	Set ID of a load (must be the ID of at least one of the following Bulk data entries: LOAD, FORCE, GRAV, MOMENT, PLOAD2). Integer > 0, no default value.

Remarks:

1. If the Case Control LOAD entry identifies a Bulk Data LOAD entry (load combining entry), then n must not appear as a set ID on any of the Bulk Data FORCE, GRAV, MOMENT or PLOAD2 entries that are in the input data file.

2. The Case Control LOAD entry must be present if a static loading is desired in a solution.

6.13 MEFFMASS

Description:

Requests calculation and output of modal effective masses in an eigenvalue solution.

Format:

MEFFMASS

Remarks:

3. This entry may appear in the Case Control section for eigenvalue extraction solutions.
4. See Bulk Data PARAM MEFMLOC for the reference point to use in calculating effective masses in Craig-Bampton (SOL 31) analyses.

6.14 METHOD

Description:

Indicates what eigenvalue extraction method (identified in the Bulk Data on an EIGR or EIGRL entry) is to be used for an eigenvalue solution.

Format:

METH[OD] = *n*

Examples:

METHOD = 18 requests that eigenvalue extraction method 18 be used

Options:

Option	Meaning
n	Set ID of a Bulk data EIGR entry. Integer > 0, no default value.

Remarks:

1. This entry must appear in the Case Control section for all eigenvalue extraction solutions.

6.15 MPC

Description:

Indicates what multipoint constraints (identified in the Bulk Data) are to be used for a solution.

Format:

MPC = *n*

Examples:

MPC = 47 requests multi point constraint set 47, defined in Bulk Data, be used

Options:

Option	Meaning
n	Set ID of an MPC and/or MPCADD Bulk data entry. Integer > 0, no default value.

Remarks:

1. There can be only one Case Control MPC entry per solution. It should appear in the Case Control section above any SUBCASE definitions.

6.16 MPCFORCES

Description:

Requests output of multi point constraint forces in the global coordinate system for selected grids. Multi point constraint forces consist of forces due to directly defined MPC's and also due to rigid elements (which are automated, internally in MYSTRAN, as MPC's).

Format:

$$\text{MPCF[ORCES][(}\underline{\text{PRINT}}\text{, PLOT)] = } \begin{Bmatrix} \text{ALL} \\ n \\ \text{NONE} \end{Bmatrix}$$

Examples:

See the DISPLACEMENT card (Section 6.5)

Options:

See the DISPLACEMENT card (Section 6.5)

Remarks:

See the DISPLACEMENT card (Section 6.5)

6.17 MPFACTOR

Description:

Requests calculation and output of modal participation factors in an eigenvalue solution.

Format:

MPFACTOR

Remarks:

1. This entry may appear in the Case Control section for eigenvalue extraction solutions.

6.18 OLOAD

Description:

Requests output of applied loads in the global coordinate system for selected grids.

Format:

OLOA[D][(<u>PRINT</u>, PLOT)] = $\begin{Bmatrix} \text{ALL} \\ n \\ \text{NONE} \end{Bmatrix}$

Examples:

See the DISPLACEMENT card (Section 6.5)

Options:

See the DISPLACEMENT card (Section 6.5)

Remarks:

See the DISPLACEMENT card (Section 6.5)

6.19 SET

Description:

Defines sets of grid points or elements for which output is desired.

Format:

SET *n* = {i_1[, i_2, i_3, i_4 THRU i_5, EXCEPT i_6, i_7, i_8 THRU i_9]}

Examples:

SET 39

SET 57 = 101 THRU 298

SET 12 = 301, 305, 491 THRU 672 EXCEPT 501

Options:

Option	Meaning
n	Set ID number. Integer > 0, no default.
i_1, i_2, i_3, etc.	Individual grid point or element numbers.
i_4 THRU i_5	Inclusive group of grid or element numbers.
EXCEPT	Grid or element numbers following EXCEPT (but before next THRU) will be excluded from the previous THRU group.

Remarks:

1. Any number of SETs can be defined as long as the ID numbers are unique integers. The SET logical entry can consist of multiple physical entries, each of 80 columns max. If a SET definition requires more than one physical entry each entry (except the last) must end with a ",".

2. Ranges in THRU statements must be increasing (that is, i_4 must be less than i_5 in the above example). It is acceptable that some grid or element numbers in the THRU range do not exist. However, all grids or elements that are in the THRU range will be included in the SET.

3. Whether the set indicates grids or elements is dependent on the context in which the SET is used. If DISP = 39 output is requested, then the integers in SET 39 will be interpreted as grid point numbers. If ELFORCE = 39 output is requested, then the integers in SET 39 will be interpreted as element numbers.

6.20 SPC

Description:

Indicates what single point constraints (identified in the Bulk Data) are to be used for a solution.

Format:

SPC = *n*

Examples:

SPC = 74   requests single point constraint SET 74 be used

Options:

Option	Meaning
n	Set ID of at least one SPC, SPC1 and/or SPCADD Bulk data entries. Integer > 0, no default value.

Remarks:

1. There can be only one Case Control SPC entry per solution. It should appear in the Case Control section above any SUBCASE definitions.

6.21 SPCFORCES

Description:

Requests output of single point constraint (SPC) forces in the global coordinate system for selected grids.

Format:

SPCF[ORCES][(PRINT, PLOT)] = $\begin{Bmatrix} \text{ALL} \\ n \\ \text{NONE} \end{Bmatrix}$

Examples:

See the DISPLACEMENT card (Section 6.5)

Options:

See the DISPLACEMENT card (Section 6.5)

Remarks:

See the DISPLACEMENT card (Section 6.5)

6.22 STRAIN

Description:

Requests output of strains for selected elements.

Format:

$$\text{STRA[IN][(}\underline{\text{CENTER}}\text{, CORNER, }\underline{\text{PRINT}}\text{, PLOT)] = } \begin{Bmatrix} \text{ALL} \\ n \\ \text{NONE} \end{Bmatrix}$$

Examples:

STRAIN = ALL	Requests strain output for all elements to the default F06 file (ASCII). For shell and solid elements, the default CENTER location is requested.
STRAIN = 45	requests strain output of Case Control entry. SET 45 goes to the F06 file.
STRAIN(PRINT, PLOT) = ALL	requests a F06 (ASCII) and OP2 (binary) file for all strains at the CENTER location.
STRAIN(CORNER, PRINT, PLOT) = ALL	Requests strain output to the F06 (ASCII) and OP2 file (binary) files. The center AND corner strains for shells and solids will be output. Exception: No corner strains for shell elements for the OP2 file (only F06).
STRAIN(PLOT) = ALL	requests an OP2 file (binary) for the strains. A F06 file will not be created.

Options:

Option	Meaning
ALL	Strains for all elements in the model will be output.
n	ID of a SET Case Control entry previously defined. Strains for the elements defined by SET n will be output. Integer > 0, no default value.
NONE	No strains will be output.

PRINT	The output will go to the *filename*.F06 file (ASCII). This file can be parsed with the F06CSV program.
PLOT	The output will go to the *filename*.OP2 file (Binary)
VONMISES	Requests von Mises strain (default). What elements?
MAXS or SHEAR	Requests maximum shear strain for shell elements and octahedral strain for solid elements.
CENTER	Requests strains at the center of shell elements (default). For solid elements, the only option is the center as of 4/21/2024, but this will be adjusted to CORNER later.
CORNER	Requests strains at the element corners for the QUAD4 and QUAD4K elements, in addition to strains at the element center. Not supported for solids as of 4/21/2024, but there is an action to support this.

Remarks:

1. Optional entries are surrounded by brackets.
2. If no output file is requested, the only output file will be the F06 (the default output is PRINT, which creates the F06).
3. The "PARAM, POST" entry has no effect on whether or not the OP2 is created and has no effect on the output requests. This is unlike some other Nastran versions.
4. NONE is used to override an overall output request made above the SUBCASE level.
5. The first four letters are the shorthand version of the full name. For example, both DISPLACMENT and DISP are acceptable.
6. A Neutral file (.NEU) may be requested by placing "PARAM, PRTNEU, YES" in the bulk data section of the deck.
7. An Answer file (.ANS) file may be requested by placing "PARAM, PRTANS, YES" in the bulk data section of the deck.
8. The parameters PRTALL, FILES, PRTF06, PRTOP2 may be used to override the output requests.
9. The MYSTRAN project developed a utility that parses a F06 file. This tool can create a customized output of the data in a format that is convenient for processing the data: https://github.com/ClassicalFEA/nastools
10. ELSTRAIN is an alternate form of this Case Control command.
11. The options VONMISES, MASS (or SHEAR), CENTER and CORNER will apply for all subcases. This does not appear to be supported as of 4/21/2024.

6.23 STRESS

Description:

Requests output of stresses for selected elements.

Format:

STRE[SS][(<u>CENTER</u>, CORNER, <u>PRINT</u>, PLOT)] = $\begin{Bmatrix} \text{ALL} \\ n \\ \text{NONE} \end{Bmatrix}$

Examples:

See the STRAIN card (Section 6.22)

Options:

See the STRAIN card (Section 6.22)

Remarks:

See the STRAIN card (Section 6.22)

6.24 SUBCASE

Description:

Beginning of the portion of the Case Control section that defines the options to be used in one subcase. Multiple subcases must be used when solution with separate static loads in one run is desired.

Format:

SUBC[ASE] = *n*

Examples:

SUBCASE = 361

Options:

Option	Meaning
n	Set ID of a subcase. Integer > 0, no default value.

Remarks:

1. There can be multiple subcases and there is no restriction on the integer numbers used for subcase IDs.
2. All Case Control entries following a SUBCASE entry (up to the next SUBCASE Case Control entry) identify the conditions for solution (loads and output) for this subcase. Case Control entries "above" the SUBCASE level will be used for all subcases unless specifically overridden in the subcase definition.

6.25 SUBTITLE

Description:

Specifies a second text line to be printed in the output file.

Format:

SUBT[ITLE] = [optional text material up to, and including, column 80]

Remarks:

1. This line of text will be printed in the output file and can be different for each subcase.

6.26 TEMPERATURE

Description:

Indicates temperature distributions (identified in the Bulk Data) that are to be used for a statics solution.

Format:

TEMP[ERATURE] = *n*

Examples:

TEMP = 174	requests temperature SET 174 be used
TEMPERATURE = 13	requests temperature SET 13 be used

Options:

Option	Meaning
n	Set ID of Bulk Data TEMP, TEMPD, TEMPRB and/or TEMPP1 cards. Integer > 0, no default value.

Remarks:

1. Thermal loads can be used in combination with other static loads in any subcase but must be selected in Case Control with the TEMPERATURE = *n* card.

6.27 TITLE

Description:

Specifies a text line to be printed in the output file.

Format:

TITLE = [optional text material up to, and including, column 80]

Remarks:

This line of text will be printed in the output file and can be different for each subcase.

6.28 VECTOR

Description:

Requests eigenvector output. What else goes here?

6.29 Bulk Data

The major function of the Bulk Data Section is to define the finite element model and the loading and constraints. In the case of loading and constraints, the Bulk Data entries have a set ID which must be chosen in Case Control for the particular load or constraint to be applied.

The entries for each of the major purposes are enumerated below. A detailed explanation of each is contained in the following section. An ENDDATA entry is considered as the last, and mandatory, entry in the Bulk data Section.

- Geometry/scalar point definition

  GRID — Defines grid point ID and location, coordinate systems for the grid location and for the global coordinate system, and permanent single point constraints.

  GRDSET — Defines default values for coordinate systems and permanent SPC's for GRID entries whose corresponding fields are blank.

  SPOINT — Defines a scalar point to which elastic and mass elements may be attached.

- Grid point sequencing

  SEQGP — Used to define the internal sequence order for grid points so as to obtain a banded stiffness matrix. If not input, then the grid order is set to, either: grid numerical order (default) or grid input order (using PARAM SEQUENCE)

- Coordinate system definition (i = 1 or 2)

  CORDiR — Defines a rectangular coordinate system.

  CORDiC — Defines a cylindrical coordinate system.

  CORDiS — Defines a spherical coordinate system.

- Element connection definition

Scalar and bushing elastic elements

CBUSH — Spring element with geometry definition

CELAS1 Defines a spring element ID, property ID and the grid/degrees of freedom to which the spring element is connected.

CELAS2 Defines a spring element ID, stiffness and the grid/degrees of freedom to which the spring element is connected.

CELAS3 Defines a spring element ID, property ID and the scalar points to which the spring element is connected.

CELAS4 Defines a spring element ID, stiffness and the scalar points to which the spring element is connected.

1D elastic elements

CBAR Defines a bar (axial load, bending, torsion) element ID, property ID and the grid connections and v vector (which, together with the bar axis, defines the orientation of the bar cross-section in the model).

BAROR Defines default values of property ID and v vector for the CBAR entry.

CROD, Defines a rod (axial load and torsion) element ID, property ID and the grid connections. The bar element can be used to describe 1D element extension, as well.

CONROD Alternate form of CROD

2D elastic elements

CQUAD4K Defines a thin quadrilateral plate (membrane, bending, twist) element ID, property ID and the grid points to which the quad element is connected.

CQUAD4 Defines a thick quadrilateral plate (membrane, bending, twist) element ID, property ID and the grid points to which the quad element is connected.

CTRIA3K Defines a thin triangular plate (membrane, bending, twist) element ID, property ID and the grid points to which the triangular element is connected.

CTRIA3 Defines a thick triangular plate (membrane, bending, twist) element ID, property ID and the grid points to which the triangular element is connected.

CSHEAR Defines a thin quadrilateral element that carries only in-plane shear

3D elastic elements

CHEXA Defines a hexahedron element with either 8 or 20 nodes.

CPENTA Defines a pentahedron element with either 6 or 15 nodes.

CTETRA Defines a tetrahedron element with either 4 or 10 nodes.

R- elements

The R-elements (currently RBE2 and RBE3) are used to generate internal multi-point constraint equations (MPC's) that define a dependence of some degrees of freedom of the model with respect to the other degrees of freedom in the model.

RBE2 Defines a rigid portion of the finite element model by specifying an element ID plus a number of dependent grid points that will behave in a rigid fashion relative to the six components of motion at a specified independent grid point. The degrees of freedom for the dependent grids are also specified. In its most simplistic form, the RBE2 can be used to define, for instance, a rigid 1-D bar or a rigid 2-D element.

RBE3 Defines one dependent grid point (and the dependent degrees of freedom at that grid point) and one or more grids (and their degrees of freedom) that the dependent degrees of freedom depend on. The most common use of this element is to distribute loads or mass specified at the dependent grid to ones at the independent grid. This is very different than the RBE3 which is a rigid element. In general, the dependent grid on the RBE3 should not be connected via elastic or rigid elements to the rest of the structure except via the RBE3 element on which it is defined. There is also a provision for specifying weighting factors at the independent grids (which in many cases are just 1.0).

RSPLINE Constraint element that defines interpolations of displacements between its 2 ends. Displacements and rotations about a line between the 2 ends are interpolated linearly. Displacements perpendicular to the line are interpolated cubically. Rotations perpendicular to the line are interpolated quadrically.

Scalar mass elements

CMASS1 Defines a mass element ID, property ID and the grid/degrees of freedom to which the mass element is connected.

CMASS2 Defines a mass element ID, stiffness and the grid/degrees of freedom to which the mass element is connected.

CMASS3 Defines a mass element ID, property ID and the scalar points to which the mass element is connected.

CMASS4 Defines a mass element ID, stiffness and the scalar points to which the mass element is connected.

User defined elements

CUSERIN Elements whose elastic properties will be defined via stiffness and mass matrices on disk files. The CUSERIN entry defines the degrees of freedom that the element is connected to. These elements are used in substructure analyses (primarily Craig-Bampton dynamic analyses).

- Element property definition

Scalar elastic element

PELAS Defines a spring element property ID and the stiffness, damping and stress recovery values for a ELAS1 scalar spring element

PBUSH Defines the elastic properties of a CBUSH element

1D elastic elements

PBAR, PBARL Defines a bar property ID and material ID and the bar properties, including: cross-sectional area, area moments, and cross-products, of inertia, torsional constant, mass per unit length, stress recovery locations on the cross-section and area factors for shear flexibility.

PROD Defines a rod property ID and material ID and the rod properties, including: cross-sectional area, torsional constant, torsion stress recovery coefficient and mass per unit length

2D elastic elements

PSHEAR	Defines the elastic properties of a CSHEAR element
PSHELL	Defines a 2D plate element property ID and material IDs and the plate properties, including: thickness, bending moment of inertia ratio, shear thickness ratio, fiber distances for stress calculation, mass per unit length.
PCOMP, 1	Defines the properties of a 2D composite plate element with n plies.

3D elastic elements

PSOLID	Defines a 3D solid element property ID and material ID and integration parameters.

User elements

PUSERIN	Defines information needed to locate the matrices (specified on disk files) for CUSERIN elements.

- Element material definition

MAT1	Defines a material ID and the material properties, including: Young's modulus, shear modulus, Poisson's ratio, material mass density, thermal expansion coefficient, reference temperature, and a damping coefficient.
MAT2	Defines a 2D anisotropic material.
MAT8	Defines an orthotropic material.
MAT9	Defines an anisotropic material.
PMASS	Defines scalar mass for elements defined on CMASS2,4 entries.

- Grid point mass

CONM2	Defines a concentrated mass at a grid point, including: mass ID, grid where mass is located, the mass value, the offsets from the grid to the mass center of gravity (c.g.), the six independent moments and products of inertia of the mass about its c.g., and the coordinate system in which the offsets and moments of inertia are specified.

- Applied loads

FORCE	Defines a concentrated force at a grid point, including: load ID, grid ID at which the force acts, coordinate system in which the force is specified, and the magnitude and direction of the force.
MOMENT	Defines a concentrated moment at a grid point, including: load ID, grid ID at which the moment acts, coordinate system in which the moment is specified, and the magnitude and direction of the moment.
GRAV	Defines an acceleration vector for the finite element model, including: load ID, coordinate system in which the acceleration vector is specified, and magnitude and direction of the acceleration vector. MYSTRAN creates a static load that is applied to a model to simulate a gravity type of loading but with rigid body motion restrained.
PLOAD2	Defines a pressure load for 2D elements, including: load ID, pressure magnitude, and element IDs for the elements that are to have the pressure load.
PLOAD4	Defines a pressure load for 2D elements, including: load ID, pressure magnitudes at up to 4 grids, and element IDs for the elements that are to have the pressure load.
LOAD	Defines a static load for the finite element model that is a linear combination of loads that are defined on FORCE, MOMENT, GRAV and PLOAD2 entries, including: ID of this load combination, a scale factor to be applied to all loads being combined, and load set IDs and magnitudes of the various load sets being combined.

RFORCE	Defines an angular velocity and optional angular acceleration of the finite element model about some defined grid point and in some defined coordinate system.
SLOAD	Defines a.

- Thermal loads (all are used by MYSTRAN to calculate loads on the model)

TEMPD	Defines an overall constant temperature for the finite element model including: temperature set ID and the temperature value.
TEMP	Defines a temperature for a grid point including: temperature set ID, the grid ID, and the temperature value
TEMPRB	Defines a temperature field for the bar element including: temperature set ID, the average temperature of the cross-section at the two bar ends, the two temperature gradients through the bar cross-section at each of the two ends.
TEMPP1	Defines a temperature field for 2D elements including: temperature set ID, the average temperature of the element at its mid-plane, the temperature gradient through the element.

- Single point constraints (SPC)

SPC	Defines a constraint for a single degree of freedom including: SPC set ID, the grid and degree of freedom component number, and the constraint value. If the constraint value is nonzero (that is, an enforced displacement), MYSTRAN calculates equivalent grid forces and applies them to the model.
SPC1	Defines degrees of freedom where displacement is zero. The definition Includes: the SPC set ID, the degree of freedom component number and the grids that are to be constrained.
SPCADD	Defines an SPC as a union of SPC's defined via SPC and/or SPC1 Bulk data entries.

- Multi point constraints (MPC)

MPC	Defines a dependence of one degree of freedom on one or more other degrees of freedom.
MPCADD	Defines an MPC as a union of MPC's defined via MPC Bulk data entries.

- Boundary degrees of freedom for Craig-Bampton (CB) analyses

SUPORT	Defines degrees of freedom at the boundary of a CB model.

- Analysis degrees of freedom (only needed when Guyan reduction is employed)

ASET	Defines degrees of freedom that are to be included in the A-set by specifying pairs of component/grid IDs
ASET1	Defines degrees of freedom that are to be included in the A-set by specifying a component number and a list of grid IDs
OMIT	Defines degrees of freedom that are to be included in the O-set by specifying pairs of component/grid IDs
OMIT1	Defines degrees of freedom that are to be included in the O-set by specifying a component number and a list of grid IDs

- Eigenvalue extraction

EIGR	Defines the data needed during eigenvalue extraction by the Givens (GIV), modified Givens (MGIV) or Inverse Power (INV) method, including: eigenvalue extraction set ID, extraction method, frequency range to search, number of estimated and desired eigenvalues, the eigenvector orthogonality criteria, and method of eigenvector renormalization.
EIGRL	Defines the data needed during eigenvalue extraction by the Lanczos method, including: eigenvalue extraction set ID, desired number of eigenvalues, and method of eigenvector renormalization.

- Partitioning vectors (used in conjunction with the OUTPUT4 and PARTN Exec Control entries)

PARVEC — The format for this entry is similar to the Bulk Data SPC entry and gives the grid/component pairs of the degrees of freedom (in any of the allowable displacement sets[4]) that define the rows or columns to be partitioned from the OUTPUT4 matrix.

PARVEC1 — The format for this entry is similar to the Bulk Data SPC1 entry and gives the same information as for the PARVEC entry, only in a different format

- Degree of freedom set definition (requests output in a row format of a displacement set)

USET — The format for this entry is similar to the Bulk Data SPC entry and requests a tabular output of selected grid/component pairs, in internal sort, that are members of a named displacement set (e.g. the A-set).

USET1 — The format for this entry is similar to the Bulk Data SPC1 entry and gives the same information as for the USET entry, only in a different format.

- PARVEC The format for this entry is the same as that for the Bulk Data SPC entry PARAM Field 2 identifies the parameter name and subsequent fields define the Parameters (used to control solution options during execution)

PARAM — Field 2 identifies the parameter name and subsequent fields define the parts of the parameter either as character, integer or real data.

- Debug (used to control debug options during execution)

DEBUG — The word DEBUG must be in field 1. The DEBUG number (I) goes in field 2 and the value of DEBUG(I) goes in field 3.

- Plot elements (only for compatibility with NASTRAN input data files)

[4] see section 3.6 for a definition of displacement sets

PLOTEL

A Bulk Data physical entry contains 80 columns of data in up to 10 fields of 8 columns each. As discussed in an earlier section, some Bulk data entries require more than the 10 fields in order to specify all of its data. Thus, a logical entry exists to describe all of the data required for one Bulk data entry. This logical entry can consist of more than one physical entry with the initial entry of 10 fields being called the "parent" and subsequent continuation entries called "child" entries. Whenever a logical entry requires continuation entries, or is capable of having continuation entries, this is noted.

Each of the Bulk Data entries is described with:

- Name of the entry and a brief sentence describing its function.
- Format of the entry with names of the data items that go in each of the (up to) 10 fields.
- Numerical example(s).
- Description of each fields' contents, data type (i.e. character, integer, real) and default values.
- Remarks regarding the entry.

An example of the format section for the PBAR Bulk Data entry is shown below with some explanation of the format. The data can be entered in the traditional way as shown with 10 fields of 8 columns each. Alternatively, the 10 fields can be separated by either commas (referred to as comma separated values, or CSV) or tabs (TSV)

Format (small field entry with 8 columns for each of the 10 fields):

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	MPL		+CONT1
+CONT1	Y1	Z1	Y2	Z2	Y3	Z3	Y4	Z4	+CONT2
+CONT2	K1	K2	I12						

The format section for the PBAR has four rows of text. Note the following:

- Row 1 of the format section (for all Bulk Data entry descriptions) is only to show the field number of the Bulk Data entry and is not part of the input for the Bulk Data entry. Each of the 10 fields is 8 columns wide.

- Row 2 is the "parent" entry for the entry illustrated here (PBAR) and is always required.
    - The entry in field 1 is the name of the Bulk Data entry and must be entered exactly as shown, starting in column 1 of field 1.
    - Fields 2-9 in general (2-8 in the PBAR above), show names of the data items (in row 1) for the Bulk Data entry (e.g. PID is the property ID for this PBAR). The data names are to be replaced by actual data that can be placed anywhere in the field. The data for a specific field might call for a character or integer or real value and this requirement is noted for each field. The entry in field 10 is only required if there is a continuation entry. If no continuation entry will be used, field 10 could contain comments.

- If continuation entries are required or optional for the parent entry, they will be shown in rows 3 and on as in the example above.
    - The entry in field 1 of a continuation must be the same as that in field 10 of the previous continuation (or parent, in the case of the first continuation).
    - The entry in fields 2-9, like those on the parent are to contain data that can be placed anywhere in the field.
    - The entry in field 10 is only required if there is to be another continuation entry to follow.
    - Continuation entries must contain a "+" sign in column 1 of field 10 of one entry and field 1 of the following entry and be the same otherwise. They do not have to be as shown in the example above (e.g. +CONT1 in field 10 of the parent and in field 1 of the first continuation entry)

- Shaded fields (like field 9 of the parent entry, above, and fields 5-9 of the second continuation entry), must be left blank.

- Data can be character, integer or real but must be of the type specified and with the following conventions:
    - Character data can be alphanumeric but must begin with an alpha character. No quotation marks are to be included. Character data that can go in fields 2-9 are always spelled out as to what the options are and must be entered exactly as shown (except that they may be placed anywhere in the field).
    - Integer data must contain no decimal point or imbedded blanks.

- Real data must contain a decimal point and no imbedded blanks. Some examples of valid real entries are:
    - 1.234567
    - 2.57E-4 or 2.57-4 (i.e. 2.57×10^{-4})
    - Each of the Bulk Data entries are described in detail on the following pages

There is also a large field Bulk data entry capability where data fields 2 through 9 of a Bulk Data entry can be 16 characters long, instead of just 8 characters. This is done in order to allow more precision in the input for real data fields. Recall that each small field physical entry has 10 fields of 8 characters each. In the large field entry, there are 2 physical entries required to specify all of the data from a small field entry. The following shows the correspondence between small and large field entries:

Small field PBAR parent entry (1 physical entry for the 10 fields of data):

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	MPL		+CONT1

Format (large field entry with 16 columns for each of fields 2 through 9):

Large field PBAR parent entry (2 physical entries needed to specify the 10 fields of data)

1	2	3	4	5	link
PBAR*	PID	MID	A	I1	*

link	6	7	8	9	10
*	I2	J	MPL		

Note that an * is used after PBAR to indicate that this is a large field entry. In addition, in order to link the 2 halves of the physical entry, an * is placed in column 73 of the 1st part of the entry and in column 1 of the 2nd part of the entry. Fields 1 and 10, as well as the last field of the 1st part and the 1st field of the 2nd part, are 8 columns each. Fields 2 through 9 are 16 columns each.

Large field entries MUST come in pairs, even for continuation entries where the 2nd of the large field entry contains no data. For example, the large field entry for the PBAR, if all data is to be entered, would be:

PBAR*	PID	MID	A	I1	*P1
*P1	I2	I12	J	MPL	*P2
*P2	Y1	Z1	Y2	Z2	*P3
*P3	Y3	Z3	Y4	Z4	*P4
*P4	K1	K2	I12	CT	*P5
*P5					

Note the last entry, which would be fields 6-9 of the small field 2nd continuation for the PBAR, is empty but must be included ***or the entry before it will be ignored***.

7 DETAILED DESCRIPTION OF BULK DATA ENTRIES

The following sections describe the input required for each of the different Bulk Data entries.

7.1 ASET

Description:

Define degrees of freedom to go into the analysis set (A-set).

Format:

1	2	3	4	5	6	7	8	9	10
ASET	G1	C1	G2	C2	G3	C3	G4	C4	

Example:

ASET	19	1	28	2345	37	124	46	134	

Data Description:

Field	Contents	Type	Default
Gi	ID numbers of grids	Integer > 0	None
Ci	Displacement component numbers	Integers 1-6	None

Remarks:

1. The degrees of freedom defined by grids Gi, components Ci will be placed in the mutually exclusive A-set. These degrees of freedom cannot have been defined to be in any other mutually exclusive set (i.e. M, S or O-sets).
2. If there are no ASET (or ASET1) and no OMIT (or OMIT1) entries, all degrees of freedom not in the M or S-set will be placed in the A-set.
3. If ASET (or ASET1) entries are present in the input data file, then all degrees of freedom not specified on these entries and also not in the M or S-sets will be placed in the O-set.
4. If both ASET (or ASET1) and OMIT (or OMIT1) are present, then all degrees of freedom not in the M and S-sets must be explicitly defined on these ASET (or ASET1) and OMIT (or OMIT1) entries.

5. Up to four pairs of Gi, Ci can be specified on one ASET entry. For more pairs, use additional ASET entries (i.e. there is no continuation entry for ASET).

7.2 ASET1

Description:

Define degrees of freedom to go into the analysis set (A-set).

Format No. 1:

1	2	3	4	5	6	7	8	9	10
ASET1	C	G1	G2	G4	G4	G5	G6	G7	
	G8	G9	(etc)						

Format No. 2:

1	2	3	4	5	6	7	8	9	10
ASET1	C	G1	THRU	G2					

Example:

ASET1	135	17934	THRU	19012					

Data Description:

Field	Contents	Type	Default
Gi	ID numbers of grids. G2 > G1	Integer > 0	None
C	Displacement component numbers	Integers 1-6	None

Remarks:

1. In Format No. 2, any grid whose ID is in the range G1 through G2 will have component C defined in the A-set.

2. The degrees of freedom defined by grids GI, components Ci will be placed in the mutually exclusive A-set. These degrees of freedom cannot have been defined to be in any other mutually exclusive set (i.e.. M, S or O-sets).

3. If there are no ASET (or ASET1) and no OMIT (or OMIT1) entries, all degrees of freedom not in the M or S-set will be placed in the A-set.

4. If ASET (or ASET1) entries are present in the input data file, then all degrees of freedom not specified on these entries and also not in the M or S-sets will be placed in the O-set.

5. If both ASET (or ASET1) and OMIT (or OMIT1) are present, then all degrees of freedom not in the M and S sets must be explicitly defined on these ASET (or ASET1) and OMIT (or OMIT1) entries.

6. Up to four pairs of Gi, Ci can be specified on one ASET entry. For more pairs, use additional ASET entries (i.e. there is no continuation entry for ASET).

7.3 BAROR

Description:

Define default values for the CBAR entry.

Format No.1:

1	2	3	4	5	6	7	8	9	10
BAROR		PID			V1	V2	V3		

Format No.2:

1	2	3	4	5	6	7	8	9	10
BAROR		PID			G0				

Examples:

BAROR		57			1.3	3.5	0.7		

BAROR		57			1563				

Data Description:

Field	Contents	Type	Default
PID	ID number of a PBAR Bulk data entry	Integer > 0 or blank	None
G0	ID of a grid used to define the orientation v vector	Integer > 0 or blank	None
Vi	The three components of the orientation v vector specified in the global coordinate system for grid G1 on the CBAR entry.	Real or blank	None

Remarks:

1. Only one BAROR entry is allowed in the input data file. Any data entered on a BAROR entry will be used unless overridden on a CBAR entry. If format 1 is used, all three components of the *v* vector must be entered.

2. The orientation v vector can be specified using either a grid point (G0) or the components Vi. Either one of these, in conjunction with the grid G1 on the CBAR entry, defines the orientation vector.

3. See CBAR entry for remarks concerning the *v* vector.

7.4 CBAR

Description:

1D bar element for axial load, bending, and torsion.

Format No. 1:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	V1	V2	V3		
	P1	P2	W1A	W2A	W3A	W2A	W2B	W3B	

Format No. 2:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	G1	G2	G0				
	P1	P2	W1A	W2A	W3A	W2A	W2B	W3B	

Examples:

CBAR	98	43	1234	56	78				
	456	13	0.0	0.2	0.3	0.1	0.05	0.10	

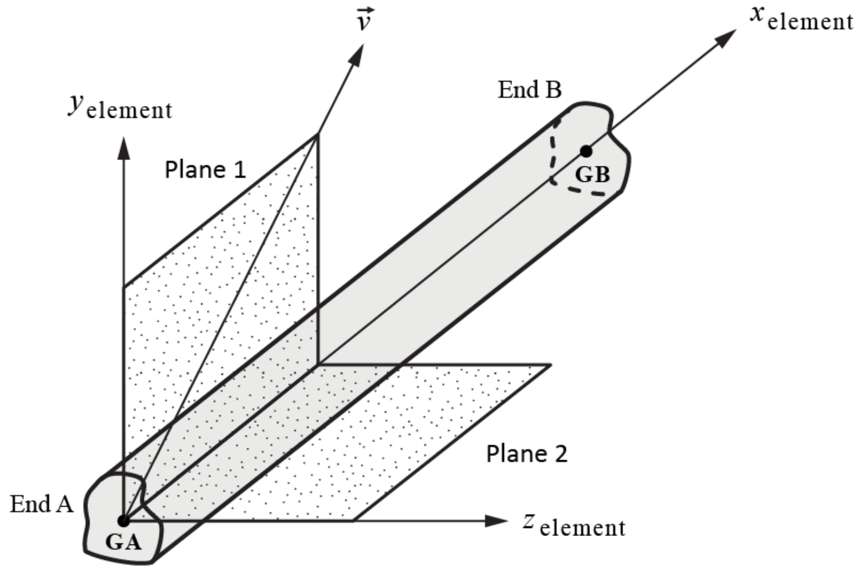
CBAR	98	43	1234	56	0.5	1.5	3.2		

Data Description:

Field	Contents	Type	Default
EID	Element ID number	Integer > 0	None
PID	ID number of a PBAR Bulk data entry	Integer > 0	EID
GA, GB	ID numbers of the grids to which the element is attached	Integer > 0	None
G0	ID of a grid used to define the orientation *v* vector	Integer > 0	None
Vi	Components of the orientation *v* vector	Real	None
PA, PB	Pin flags for bar ends A and B, respectively	Integers 1-6	None
W1j	Components of the bar offset from grid G1	Real	None
W2j	Components of the bar offset from grid G2	Real	None

Remarks:

1. No other element in the model may have the same element ID.
2. The *v* vector is a vector from either: (a) grid GA in the direction of the vector defined by V1, V2, V3 or (b) from grid GA to grid G0. These components are measured in the global coordinate system of grid G1 (see GRID entry for definition of the global coordinate system for a grid). If format 1 is used, all three components of the v vector must be entered.
3. The local *x*-axis of the element is a vector from GA through GB.
4. The *x*-axis and the *v* vector define a plane. On the PBAR entry, I1 is the bending moment of inertia in this plane.



$x_{element}$ = neutral axis of the bar

$\vec{v}$ = vector specified on CBAR or BAROR entry used to define Plane 1

CBAR without offsets, Format No. 1

7.5 CBUSH

Description:

Bush element

Format No. 1:

1	2	3	4	5	6	7	8	9	10
CBUSH	EID	PID	G1	G2	G0			CID	
	S	OCID	S1	S2	S3				

Format No. 2:

1	2	3	4	5	6	7	8	9	10
CBUSH	EID	PID	GA	GB	V1	V2	V3	CID	
	S	OCID	S1	S2	S3				

Examples:

CBUSH	98	43	1234	56	78				
	456	13	0.0	0.2	0.3				

CBUSH	98	43	1234	56	0.5	1.5	3.2		

Data Description:

Field	Contents	Type	Default
EID	Element ID number	Integer > 0	None
PID	ID number of a PBUSH Bulk data entry	Integer > 0	EID

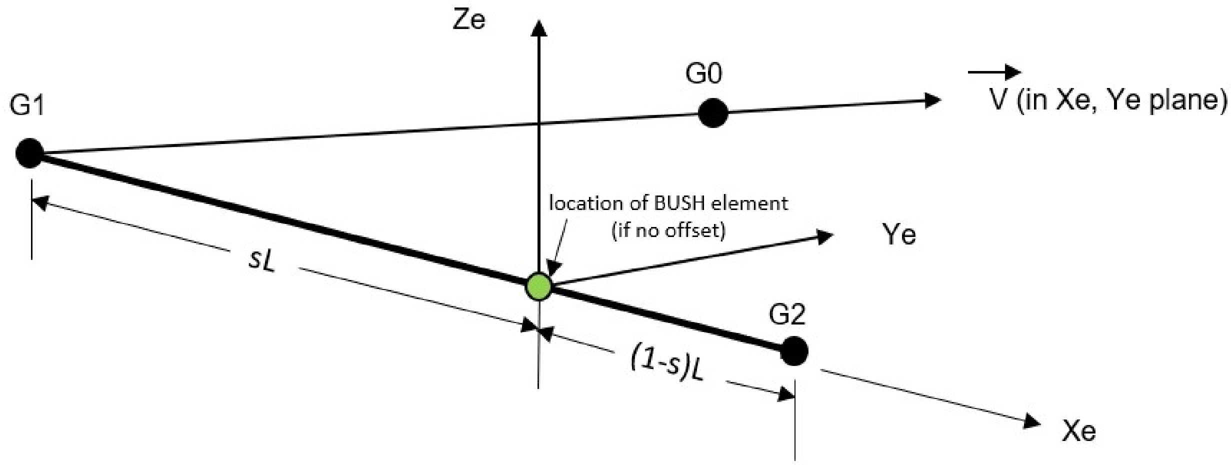
G1, G2	ID numbers of the grids to which the element is attached	Integer > 0	None
G0	ID of a grid used to define the orientation v vector	Integer > 0	None
Vi	Components of the orientation v vector	Real	None
CID	Element coordinate system identification (0 is basic system) If blank, the element system is defined by G0 or Vi	Integer >= 0 or blank	None
S	Location of spring	0.< Real < 1.	0.5
OCID	ID of coordinate system used in defining the offsets. OCID = -1 indicates that the offsets are specified in the element coordinate system	Integer >= -1	-1
Si	Components of spring offset	Real	0.

Remarks:

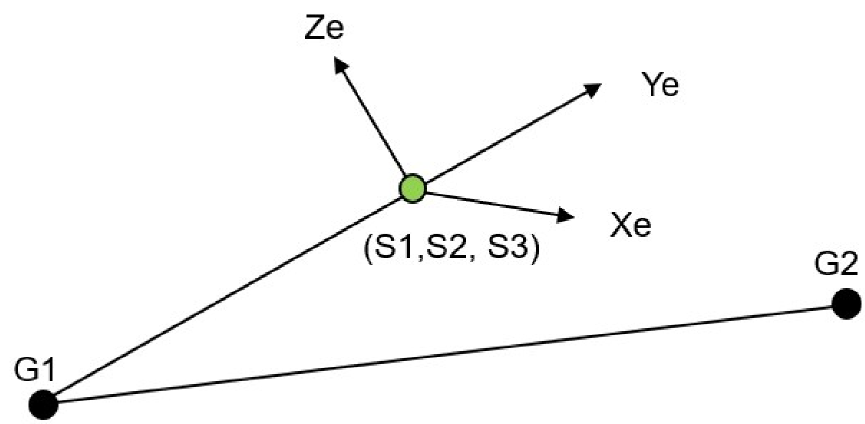
1. No other element in the model may have the same element ID.
2. If CID >= 0 the element x axis is along the x axis of coordinate system CID, etc.
3. A V vector must be specified. That is, fields 6-9 cannot all be blank.
4. GB cannot be blank.
5. The following pertains to OCID:

   (a) OCID = -1 (or blank) means S is used and Si are ignored.

   (b) OCID >= 0 means S is ignored and Si are used.



BUSH element



Offsets Si

7.6 CELAS1

Description:

Scalar spring element connected to 2 grid points (GRID's) with reference to a PELAS entry to define the real values for the element.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS1	EID	PID	G1	C1	G2	C2			

Example:

CELAS1	789	32	3731	5	67	5			

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
PID	ID number of a PELAS Bulk data entry	Integer > 0	EID
Gi	ID numbers of the grids to which the element is attached	Integer > 0	None
Ci	Component number (1-6) of the degree of freedom, at Gi, to which the spring element is connected	Integer 1-6	None

Remarks:

1. No other element in the model may have the same element ID.
2. The degrees of freedom specified by Gi/Ci must be global degrees of freedom.
3. Care must be exercised that rigid body motion of the model is not restrained when using scalar springs. For example, connecting a scalar spring between two translational degrees of freedom that are not colinear may restrain rigid body motion and give erroneous results.

7.7 CELAS2

Description:

Scalar spring element connected to 2 grid points (GRID's) with the element stiffness defined.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS2	EID	K	G1	C1	G2	C2			

Example:

CELAS2	789	1.234+06	3731	5	67	5			

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
K	Stiffness value	Real	0.
Gi	ID numbers of the grids to which the element is attached	Integer > 0	None
Ci	Component number (1-6) of the degree of freedom, at Gi, to which the spring element is connected	Integer 1-6	None

Remarks:

1. No other element in the model may have the same element ID.
2. The degrees of freedom specified by Gi/Ci must be global degrees of freedom.
3. Care must be exercised that rigid body motion of the model is not restrained when using scalar springs. For example, connecting a scalar spring between two translational degrees of freedom that are not colinear may restrain rigid body motion and give erroneous results.

7.8 CELAS3

Description:

Scalar spring element connected to 2 scalar points (SPOINT's) with reference to a PELAS entry to define the real values for the element.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS3	EID	PID	S1	S2					

Example:

CELAS3	789	32	3731	5					

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
PID	ID number of a PELAS Bulk data entry	Integer > 0	EID
Si	ID numbers of the SPOINT's to which the element is attached	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID.
2. The degrees of freedom specified by Si must be global degrees of freedom.
3. Care must be exercised that rigid body motion of the model is not restrained when using scalar springs. For example, connecting a scalar spring between two translational degrees of freedom that are not colinear may restrain rigid body motion and give erroneous results.

7.9 CELAS4

Description:

Scalar spring element connected to 2 scalar points (SPOINT’s) with the element stiffness defined.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS4	EID	K	S1	S2					

Example:

CELAS4	789	32	3731	5					

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
K	Stiffness value	Real	0.
Si	ID numbers of the SPOINT’s to which the element is attached	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID.
2. The degrees of freedom specified by Si must be global degrees of freedom.
3. Care must be exercised that rigid body motion of the model is not restrained when using scalar springs. For example, connecting a scalar spring between two translational degrees of freedom that are not colinear may restrain rigid body motion and give erroneous results.

7.10 CHEXA

Description:

3D solid tetrahedron element.

Format:

1	2	3	4	5	6	7	8	9	10
CHEXA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20			

Example:

CHEXA	98	43	101	123	254	12	621	8945	
	43	998							

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
PID	ID number of a PSOLID Bulk data entry	Integer > 0	None
G1-G20	ID numbers of the grids to which the element is attached. Specify G1-G8 for a 4 node HEXA and all 20 for a 20 node HEXA	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID.
2. The first continuation entry is required. The second is only needed for the 20-node element.

7.11 CMASS1

Description:

Scalar mass element connected to 2 grid points (GRID's) with reference to a PMASS entry to define the real values for the element.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS1	EID	PID	G1	C1					

Example:

CMASS1	789	32	3731	5					

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
PID	ID number of a PMASS Bulk data entry	Integer > 0	EID
G1	ID number of the grid to which the element is attached	Integer > 0	None
C	Component number (1-6) of the degree of freedom, at G1, to which the mass element is connected	Integer 1-6	None

Remarks:

1. No other element in the model may have the same element ID.
2. The degrees of freedom specified by Gi/Ci must be global degrees of freedom.
3. For MYSTRAN, the mass can only be connected to 1 grid (not 2 as is allowed in NASTRAN).

7.12 CMASS2

Description:

Scalar mass element connected to 2 grid points (GRID's) with the element stiffness defined.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS2	EID	K	G1	C1					

Example:

CMASS2	789	1.234+06	3731	5					

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
K	Stiffness value	Real	0.
Gi	ID numbers of the grids to which the element is attached	Integer > 0	None
Ci	Component number (1-6) of the degree of freedom, at Gi, to which the mass element is connected	Integer 1-6	None

Remarks:

1. No other element in the model may have the same element ID.
2. The degrees of freedom specified by Gi/Ci must be global degrees of freedom.
3. For MYSTRAN, the mass can only be connected to 1 grid (not 2 as is allowed in NASTRAN).

7.13 CMASS3

Description:

Scalar mass element connected to 2 scalar points (SPOINT's) with reference to a PMASS entry to define the real values for the element.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS3	EID	PID	S1						

Example:

CMASS3	789	32	3731	5					

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
PID	ID number of a PMASS Bulk data entry	Integer > 0	EID
Si	ID numbers of the SPOINT's to which the element is attached	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID.
2. The degrees of freedom specified by Si must be global degrees of freedom.
3. For MYSTRAN, the mass can only be connected to 1 scalar point (not 2 as is allowed in NASTRAN).

7.14 CMASS4

Description:

Scalar mass element connected to 2 scalar points (SPOINT's) with the element stiffness defined.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS4	EID	K	S1						

Example:

CMASS4	789	32	3731	5					

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
K	Stiffness value	Real	0.
Si	ID numbers of the SPOINT's to which the element is attached	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID.
2. The degrees of freedom specified by Si must be global degrees of freedom.
3. For MYSTRAN, the mass can only be connected to 1 scalar point (not 2 as is allowed in NASTRAN).

7.15 CONM2

Description:

Concentrated mass at a grid point.

Format:

1	2	3	4	5	6	7	8	9	10
CONM2	EID	G	CID	M	X1	X2	X3		
	I11	I21	I22	I31	I32	I33			

Example:

CONM2	98	354	29	0.5	0.3	1.2	0.65		
	123.	-45.	321.	12.	-43.	567.			

Data Description:

Field	Contents	Type	Default
EID	Element identification (ID) number	Integer > 0	None
G	ID number of the grid to which the mass is attached	Integer > 0	None
CID	ID number of a coordinate system defined on a CORD2C, CORD2R or CORD2S Bulk Data entry	Integer > 0	0
M	Mass value	Real	0.
Xi	Offset distances from grid G to the center of gravity of M in coordinate system CID	Real	0.
Iij	The 6 independent moments of inertia of M about its center of gravity measured in coordinate system CID.	Real	0.

Remarks:

1. EID must be unique among all CONM2 entries.

2. The continuation entry is optional.

3. The moments of inertia I11, I22 and I33 (if entered) must be > 0.

4. A blank entry for CID implies the basic coordinate system.

7.16 CONROD

Description:

1D elastic rod element for axial load and torsion with properties defined in the entry (no reference to a property entry). Note that the CROD entry references a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CONROD	EID	G1	G2	MID	A	J	C	NSM	

Example:

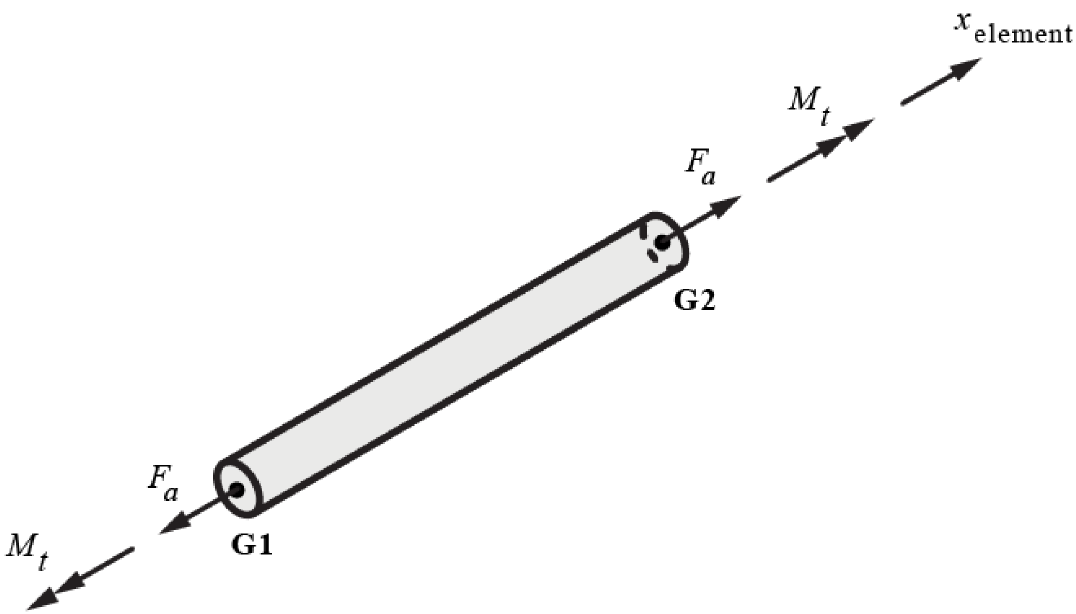
CONROD	98	43	1234	56					

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
G1, G2	ID numbers of the grids to which the element is attached	Integer > 0	EID
MID	Material ID number	Integer > 0	None
A	Bar cross-sectional area	Real	0.
J	Torsional constant	Real	0.
C	Torsional stress recovery coefficient	Real	0.
MPL	Mass per unit length	Real	0.

Remarks:

1. No other element in the model may have the same element ID.
2. The local x_{element} axis of the element is a vector from G1 through G2.



F_a = axial load

M_t = torque

x_{element} = rod axis (positive from grid G1 through G2)

7.17 CORD1C

Description:

Cylindrical coordinate system definition defined via 3 grids. TWO separate coordinate systems may be defined on one CORD1C entry. Note that CORD2C use 3 points to define to the coordinate system.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1C	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example:

CORD1C									

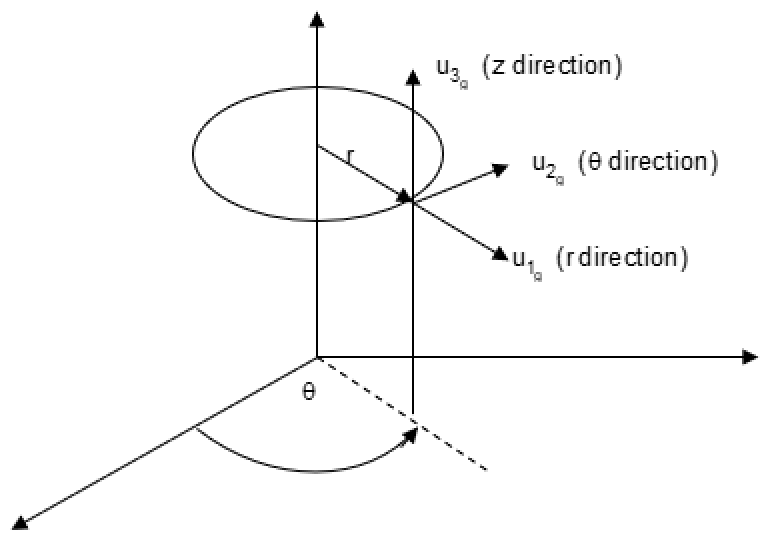
Data Description:

Field	Contents	Type	Default
CID	Coordinate system ID number	Integer > 0	None
G1A, G1B	ID's of grid points at the origin of systems A, B respectively	Integer > 0	None
G2A, G2B	ID's of grid points along the z axis of systems A, B respectively	Integer > 0	None
G3A, G3B	ID's of grid points in the x-z plane of systems A, B respectively	Integer > 0	None

Remarks:

1. See the following figure for the cylindrical coordinate system notation and the "defining" rectangular system.
2. CIDA, CIDB must be unique over all coordinate systems defined in the model.

3. One or two coordinate systems may be defined on a single CORD1C entry.
4. The grid points on this entry must be defined in a system that does not involve the system being defined.
5. The location of a grid point using this coordinate system is defined by the r, θ, z coordinates of a cylindrical coordinate system.



7.18 CORD1R

Description:

Rectangular coordinate system definition defined via 3 grids. TWO separate coordinate systems may be defined on one CORD1R entry. Note that CORD2R use 3 points to define to the coordinate system.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1R	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example:

CORD1R									

Data Description:

Field	Contents	Type	Default
CID	Coordinate system ID number	Integer > 0	None
G1A, G1B	ID's of grid points at the origin of systems A, B respectively	Integer > 0	None
G2A, G2B	ID's of grid points along the *z*-axis of systems A, B respectively	Integer > 0	None
G3A, G3B	ID's of grid points in the *x*-*z* plane of systems A, B respectively	Integer > 0	None

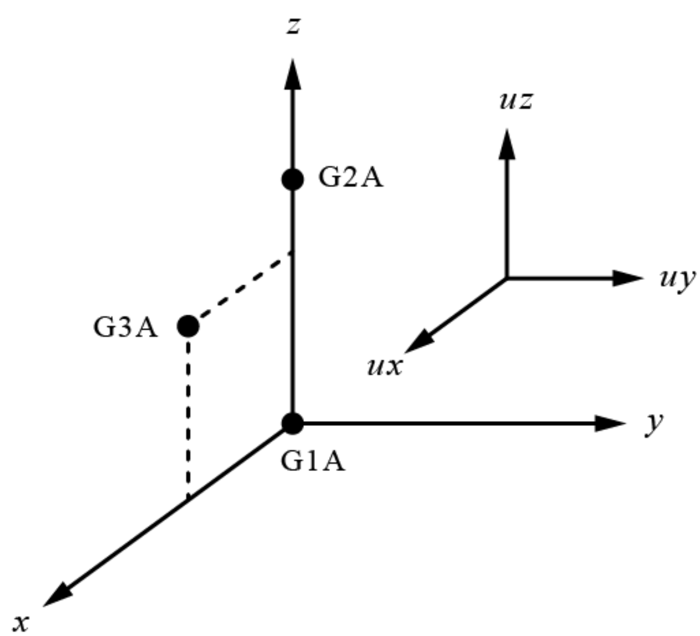
Remarks:

1. CIDA, CIDB must be unique over all coordinate systems defined in the model.
2. One or two coordinate systems may be defined on a single CORD1R entry.

3. The grid points on this entry must be defined in a system that does not involve the system being defined.

4. The location of a grid point using this coordinate system is defined by the x, y, z coordinates of a rectangular coordinate system.

The following figure needs to be improved for clarity.



For coordinate system CIDA, which uses G1A, G2A, G3A
(Note that CIDB uses G1B, G2B, G3B)

7.19 CORD1S

Description:

Spherical coordinate system definition defined via 3 grids. TWO separate coordinate systems may be defined on one CORD1S entry. Note that CORD2S use 3 points to define to the coordinate system.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1S	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example:

CORD1S									

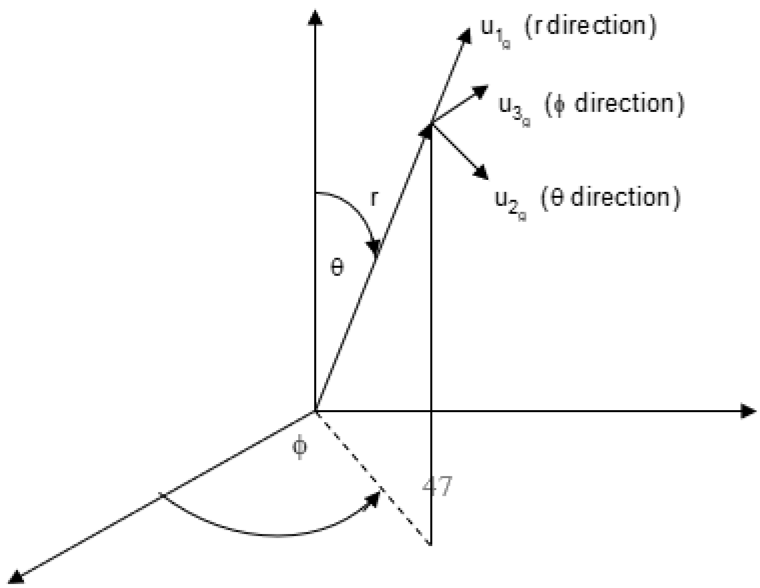
Data Description:

Field	Contents	Type	Default
CID	Coordinate system ID number	Integer > 0	None
G1A, G1B	ID's of grid points at the origin of systems A, B respectively	Integer > 0	None
G2A, G2B	ID's of grid points along the *z*-axis of systems A, B respectively	Integer > 0	None
G3A, G3B	ID's of grid points in the *x*-*z* plane of systems A, B respectively	Integer > 0	None

Remarks:

1. CIDA, CIDB must be unique over all coordinate systems defined in the model.
2. One or two coordinate systems may be defined on a single CORD1S entry.
3. The grid points on this entry must be defined in a system that does not involve the system being defined.

4. The location of a grid point using this coordinate system is defined by the *r*, *θ*, *φ* coordinates of a spherical coordinate system.



7.20 CORD2C

Description:

Cylindrical coordinate system definition defined via 3 points (9 values). Note that CORD1R use 3 grids to define to the coordinate system.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2C	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

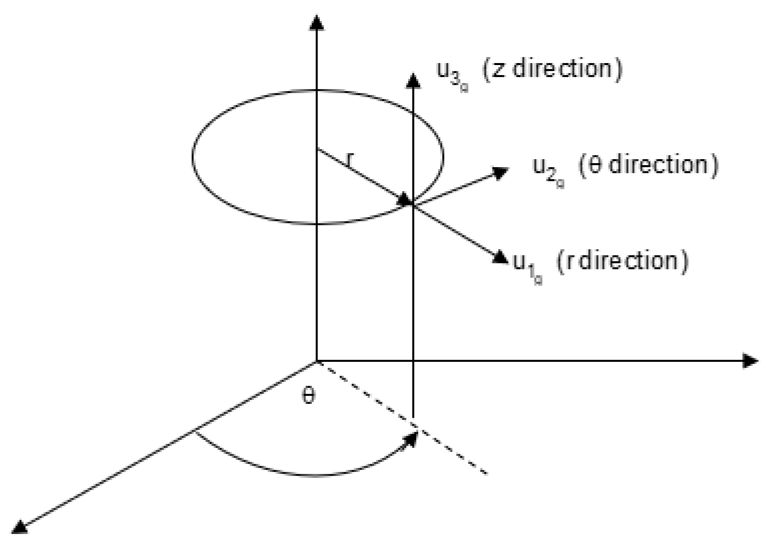
CORD2C	26	41	4.6	1.9	13.89	5.76	11.3	2.7	
	4.9	26.2	3.4						

Data Description:

Field	Contents	Type	Default
CID	Coordinate system ID number	Integer > 0	None
RID	ID number of the reference coordinate system in which the points Ai, Bi, Ci are specified	Integer >= 0 or blank	0
Ai	Coordinates of the origin of CID (specified in RID coordinate system)	Real	None
Bi	Coordinates of a point on the z axis of the defining rectangular system of CID (specified in RID coordinate system)	Real	None
Ci	Coordinates of a point in the x-z plane of the defining rectangular system of CID (specified in RID coordinate system)	Real	None

Remarks:

1. See the following figure for the cylindrical coordinate system notation and the “defining” rectangular system.
2. CID must be unique over all coordinate systems defined in the model.
3. The continuation entry is required.
4. RID = 0 or blank means that the reference coordinate system is the basic coordinate system.
5. CID must be able to be traced, through a chain of coordinate references, back to the basic system. For example, in the example above CID 26 is defined using system 46. Coordinate system 46 can be defined using some other coordinate system, and so on, until the final RID is 0 (basic).
6. The basic system need not be defined explicitly. Its axes are implied from the model (grid point coordinates on GRID entries and coordinate system definitions of all other systems).



7.21 CORD2R

Description:

Rectangular coordinate system definition defined via 3 points (9 values). Note that CORD1R use 3 grids to define to the coordinate system.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2R	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

CORD2R	26	41	4.6	1.9	13.89	5.76	11.3	2.7	
	4.9	26.2	3.4						

Data Description:

Field	Contents	Type	Default
CID	Coordinate system ID number	Integer > 0	None
RID	ID number of the reference coordinate system in which the points Ai, Bi, Ci are specified	Integer >= 0 or blank	0
Ai	Coordinates of the origin of CID (specified in RID coordinate system)	Real	None
Bi	Coordinates of a point on the *z*-axis of the defining rectangular system of CID (specified in RID coordinate system)	Real	None
Ci	Coordinates of a point in the *x*-*z* plane of the defining rectangular system of CID (specified in RID coordinate system)	Real	None

Remarks:

1. CID must be unique over all coordinate systems defined in the model.
2. The continuation entry is required.
3. RID = 0 or blank means that the reference coordinate system is the basic coordinate system.
4. CID must be able to be traced, through a chain of coordinate references, back to the basic system. For example, in the example above CID 26 is defined using system 46. Coordinate system 46 can be defined using some other coordinate system, and so on, until the final RID is 0 (basic).
5. The basic system need not be defined explicitly. Its axes are implied from the model (grid point coordinates on GRID entries and coordinate system definitions of all other systems).

7.22 CORD2S

Description:

Spherical coordinate system definition defined via 3 points (9 values). Note that CORD1S use 3 grids to define to the coordinate system.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2S	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

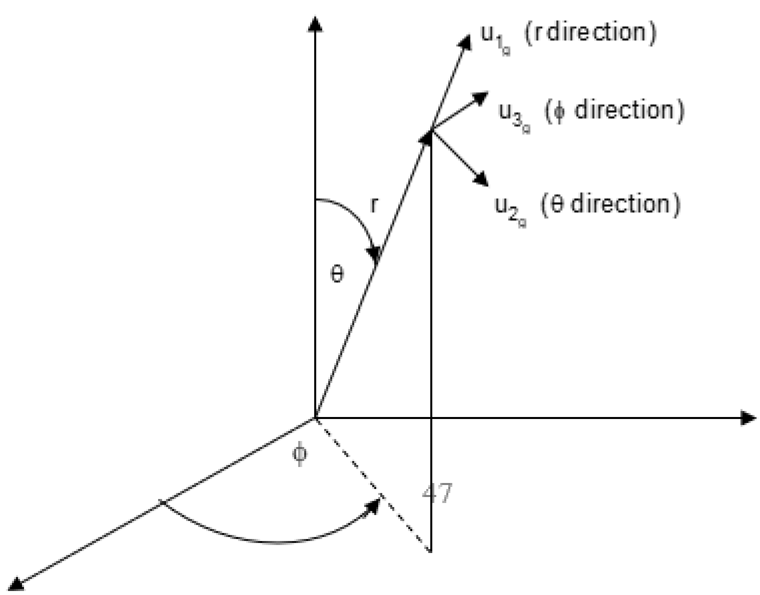
CORD2S	26	41	4.6	1.9	13.89	5.76	11.3	2.7	
	4.9	26.2	3.4						

Data Description:

Field	Contents	Type	Default
CID	Coordinate system ID number	Integer > 0	None
RID	ID number of the reference coordinate system in which the points Ai, Bi, Ci are specified	Integer >= 0 or blank	0
Ai	Coordinates of the origin of CID (specified in RID coordinate system)	Real	None
Bi	Coordinates of a point on the z axis of the defining rectangular system of CID (specified in RID coordinate system)	Real	None
Ci	Coordinates of a point in the x-z plane of the defining rectangular system of CID (specified in RID coordinate system)	Real	None

Remarks:

1. See the following figure for the spherical coordinate system notation and the "defining" rectangular system.
2. CID must be unique over all coordinate systems defined in the model.
3. The continuation entry is required.
4. RID = 0 or blank means that the reference coordinate system is the basic coordinate system.
5. CID must be able to be traced, through a chain of coordinate references, back to the basic system. For example, in the example above CID 26 is defined using system 46. Coordinate system 46 can be defined using some other coordinate system, and so on, until the final RID is 0 (basic).
6. The basic system need not be defined explicitly. Its axes are implied from the model (grid point coordinates on GRID entries and coordinate system definitions of all other systems).



7.23 CPENTA

Description:

3D solid pentahedron element.

Format:

1	2	3	4	5	6	7	8	9	10
CPENTA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15								

Example:

CPENTA	98	43	101	123	254	12	1002	98	

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
PID	ID number of a PSOLID Bulk data entry	Integer > 0	None
G1-G15	ID numbers of the grids to which the element is attached. Specify G1-G6 for a 6 node PENTA and all 15 for a 15 node PENTA	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID.
2. Continuation entries are only needed for the 15-node element.

7.24 CQUAD4

Description:

Thick quadrilateral plate element. This element can have membrane, bending, and transverse shear stiffness.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	

Example:

CQUAD4	68	123	935	67	1357	2			

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
PID	ID number of a PSHELL Bulk data entry	Integer > 0	EID
Gi	ID numbers of the grids to which the element is attached	Integer > 0	None
THETA	Material property orientation angle in degrees measured from axis connecting grids 1 and 2	Real	0.
MCID	Material coordinate system identification number. The *x*-axis of the material coordinate system is the projection of the 1st axis (*x* or *r*) of the MCID coordinate system onto the surface of the shell element at the element center.		
ZOFFS	Offset of the grid plane to element reference plane	Real	0.

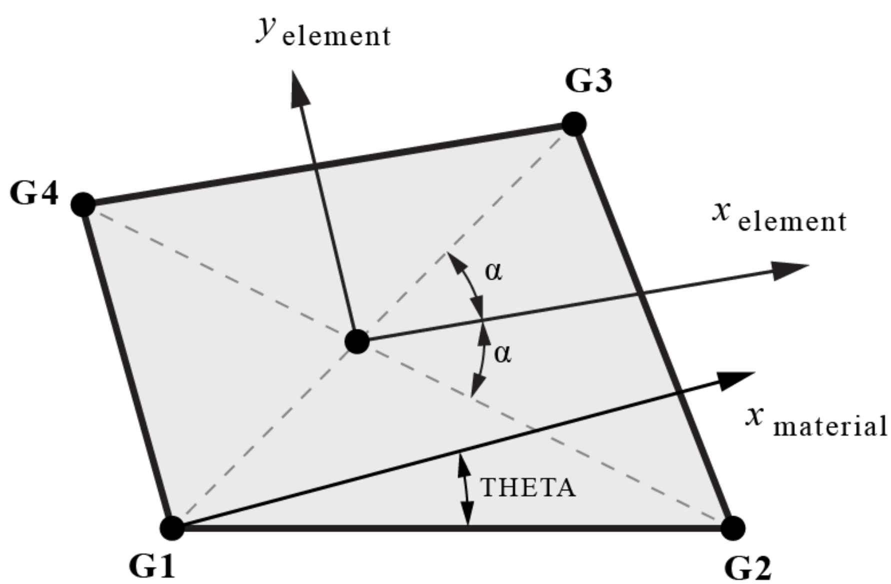
Remarks:

1. No other element in the model may have the same element ID.

2. The grids must be numbered in a clockwise or counter clockwise direction around the quadrilateral element.

3. The $z_{element}$ axis of the element coordinate system is in the direction of the cross-product of the diagonal from G1 to G3 with the diagonal from G2 to G4. If the element is rectangular, the $x_{element}$ axis of the element coordinate system is the projection of the vector from G1 to G2 onto the mean plane. If not rectangular, this is rotated to split the angle between the diagonals. The $y_{element}$ axis is in the direction of $z_{element}$ cross $x_{element}$.

4. See discussion in Section 3.2 regarding the three versions of the QUAD4 element. These are controlled by the parameter QUAD4TYP (Options are MIN4, MIN4T, MITC+).



7.25 CQUAD4K

Description:

This is a legacy element. The documentation may not be updated.

Thin quadrilateral plate element. This element has membrane and bending stiffness but does not include flexibility for transverse shear deformations.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD4K	EID	PID	G1	G2	G3	G4			

Example:

CQUAD4K	68	123	935	67	1357	2			

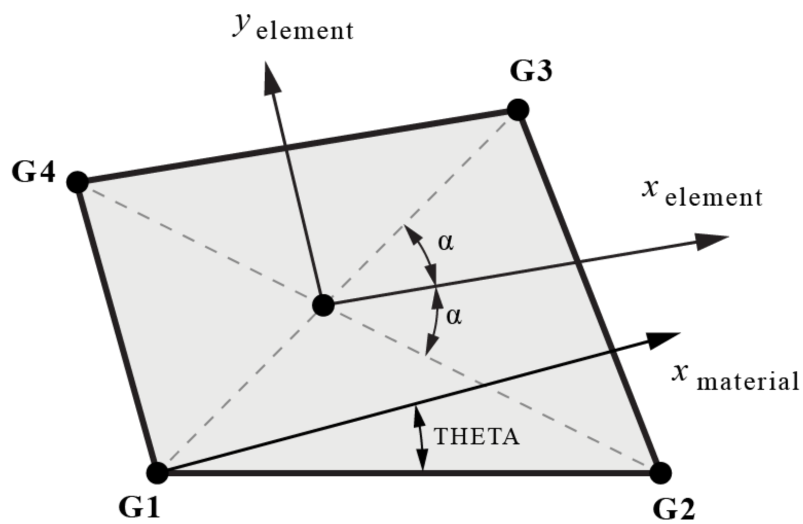
Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
PID	ID number of a PSHELL Bulk data entry	Integer > 0	EID
Gi	ID numbers of the grids to which the element is attached	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID.
2. The grids must be numbered in a clockwise or counter clockwise direction around the quadrilateral element.
3. The $z_{element}$ axis of the element coordinate system is in the direction of the cross-product of the diagonal from G1 to G3 with the diagonal from G2 to G4. If the element is rectangular, the $x_{element}$ axis of the element coordinate system is the projection of the vector from G1 to G2

onto the mean plane. If not rectangular, this is rotated to split the angle between the diagonals. The y_{element} axis is in the direction of z_{element} cross x_{element}.



7.26 CQUAD8

Description:

Quad element with eight nodes.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD8	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	T1	T2	T3	T4	THETA or MCID		

Example:

CQUAD8	68	123	935	67	1357	2			

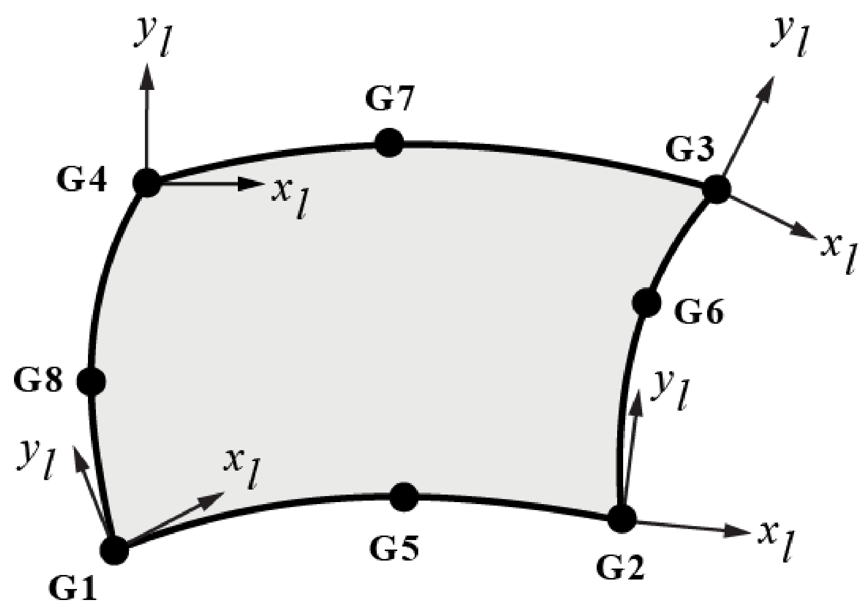
Data Description:

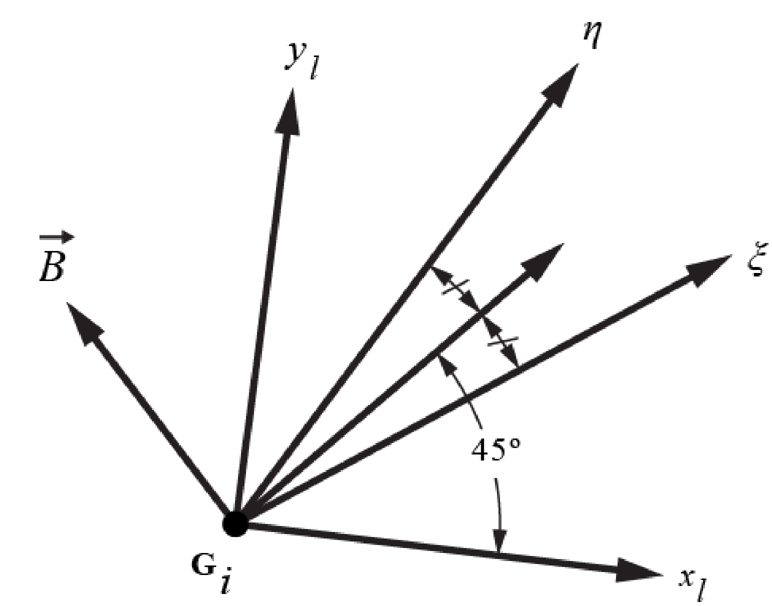
Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
PID	ID number of a PSHELL Bulk data entry	Integer > 0	EID
Gi	ID numbers of the grids to which the element is attached	Integer > 0	None
THETA	Material property orientation angle in degrees measured from axis connecting grids 1 and 2	Real	0.
ZOFFS	Offset of the grid plane to element reference plane	Real	0.

Remarks:

1. No other element in the model may have the same element ID.

2. Grids 1-4 and 5-8 must be numbered in a clockwise or counter clockwise direction around the quadrilateral element.





7.27 CROD

Description:

1D elastic rod element for axial load and torsion, with reference to a property entry. Noe that the CONROD entry does not require a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CROD	EID	PID	G1	G2					

Example:

CROD	98	43	1234	56					

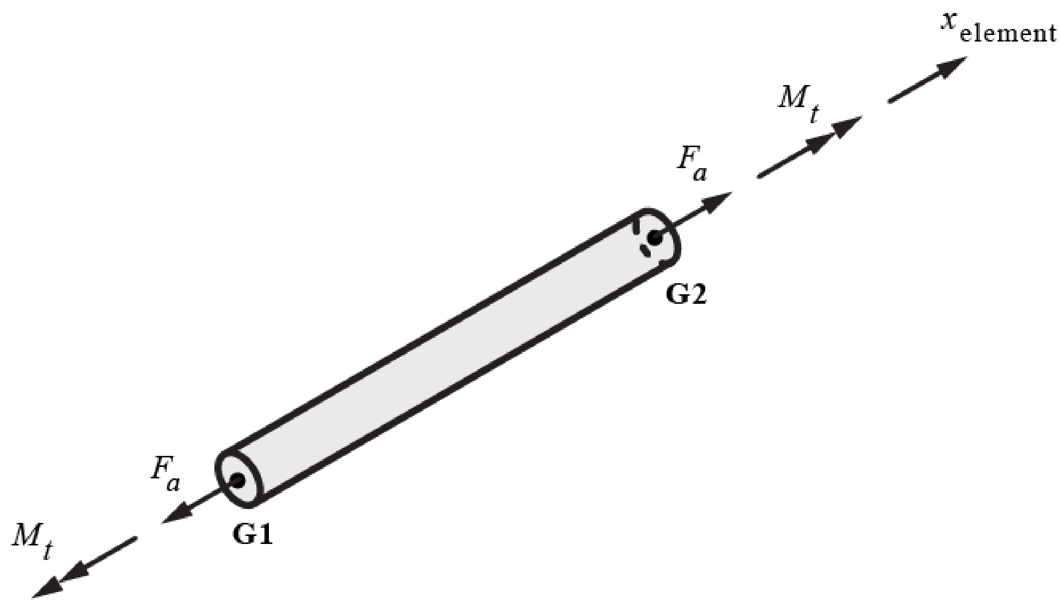
Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
PID	ID number of a PROD Bulk data entry	Integer > 0	EID
G1, G2	ID numbers of the grids to which the element is attached	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID.

2. The local x_{element} axis of the element is a vector from G1 through G2.



F_a = axial load

M_t = torque

x_{element} = rod axis (positive from grid G1 through G2)

7.28 CSHEAR

Description:

Defines a quadrilateral shell element that carries only in-plane shear.

Format:

1	2	3	4	5	6	7	8	9	10
CSHEAR	EID	PID	G1	G2	G3	G4			

Example:

CSHEAR	98	43	978	564	94	465			

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
PID	ID number of a PSHEAR Bulk data entry	Integer > 0	EID
Gi	ID numbers of the grids to which the element is attached	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID.
2. The local x_e axis of the element is defined the same as for the QUAD4 element.

7.29 CTETRA

Description:

3D solid tetrahedron element.

Format:

1	2	3	4	5	6	7	8	9	10
CTETRA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10					

Example:

CTETRA	98	43	101	123	254	12			

Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
PID	ID number of a PSOLID Bulk data entry	Integer > 0	None
G1-G10	ID numbers of the grids to which the element is attached. Specify G1-G4 for a 4 node TETRA and all 10 for a 10 node TETRA	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID.
2. Continuation entries are only needed for the 15-node element.

7.30 CTRIA3

Description:

Thick triangular plate element. This element has membrane and bending stiffness and can include flexibility for transverse shear deformations.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIA3	EID	PID	G1	G2	G3	THETA	ZOFFS		

Example:

CTRIA3	68	123	935	67	1357				

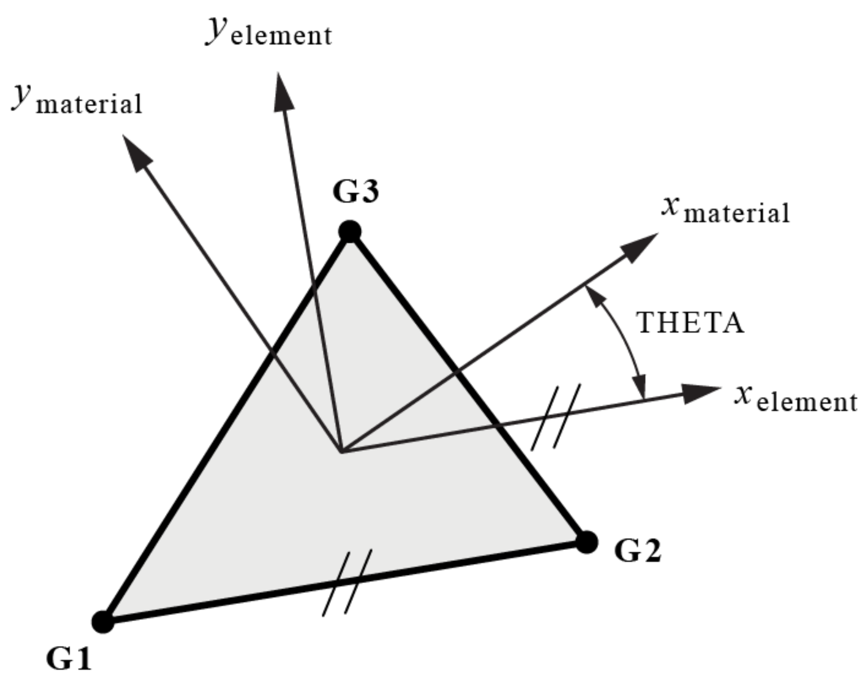
Data Description:

Field	Contents	Type	Default
EID	Unique element identification (ID) number	Integer > 0	None
PID	ID number of a PSHELL Bulk data entry	Integer > 0	EID
Gi	ID numbers of the grids to which the element is attached	Integer > 0	None
THETA	Material property orientation angle in degrees measured from axis connecting grids 1 and 2	Real	0.
MCID	Material coordinate system identification number. The *x*-axis of the material coordinate system is the projection of the 1st axis (*x* or *r*) of the MCID coordinate system onto the surface of the shell element at the element center.		
ZOFFS	Offset of the grid plane to element reference plane	Real	0.

Remarks:

1. No other element in the model may have the same element ID

2. The local x_{element} axis of the element is in the direction from G1 to G2. The local z_{element} axis is in the direction of the cross product of the vector from G1 to G2 with the vector from G1 to G3. The local y_{element} axis is in the direction of z_{element} cross x_{element}.



7.31 CTRIA3K

Description:

Thin triangular plate element. This element has membrane and bending stiffness but does not include flexibility for transverse shear deformations.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIA3K	EID	PID	G1	G2	G3				

Example:

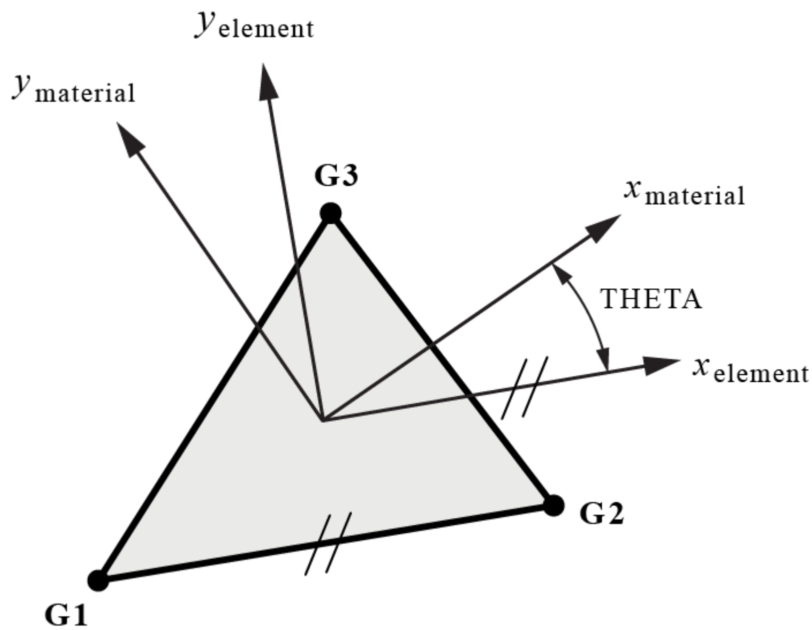
CTRIA3K	68	123	935	67	1357				

Data Description:

Field	Contents	Type	Default
EID	Element identification (ID) number	Integer > 0	None
PID	ID number of a PSHELL Bulk data entry	Integer > 0	EID
Gi	ID numbers of the grids to which the element is attached	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID.
2. The local $x_{element}$ axis of the element is in the direction from G1 to G2. The local $z_{element}$ axis is in the direction of the cross product of the vector from G1 to G2 with the vector from G1 to G3. The local $y_{element}$ axis is in the direction of $z_{element}$ cross $x_{element}$.



y_{element}
y_{material}
G3
x_{material}
THETA
x_{element}
G2
G1

7.32 CUSERIN

Description:

User defined element for which the user will supply the mass and stiffness matrices via NASTRAN formatted INPUTT4 files.

Format 1:

1	2	3	4	5	6	7	8	9	10
CUSERIN	EID	PID	NG	NS	CID0				
	G1	C1	G2	C2	etc				
	S1	S2	S3	etc					

Format 2:

1	2	3	4	5	6	7	8	9	10
CUSERIN	EID	PID	NG	NS	CID0				
	G1	C1	G2	C2	etc				
	S1	THRU	S2						

Example:

CUSERIN	32	123	3	8	198				
	201	123	202	13	203	3			
	20001	THRU	20008						

Data Description:

Field	Contents	Type	Default
EID	Element identification (ID) number	Integer > 0	None

PID	ID number of a PUSERIN Bulk Data entry	Integer > 0	EID
NG	Number of grid points (GRID's) that the element is attached to	Integer >= 0	0
NS	Number of scalar points (SPOINT's) that the element is attached to	Integer >= 0	0
CID0	ID of the coordinate system that defines the basic coord system of this element relative to the basic coord system of the overall model	Integer >= 0	0
Gi, Ci	NG grid/component numbers for the grids and components that the element connects to (Ci have to be integers 1,2,3,4,5 and/or 6)	Integer > 0	None
Si	NS scalar points (Bulk Data SPOINT) that the element connects to	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID.
2. An example of how this element is used is in Craig-Bampton analyses where a system model is made up of one or more substructures (generated in CB model generation solution sequence, SO.
3. Each CB model's connection information is described by a CUSERIN element. The PUSERIN Bulk Data entry is required.

7.33 DEBUG

Description:

Define debug parameters.

Format:

1	2	3	4	5	6	7	8	9	10
DEBUG	i	VALUE							

Example:

DEBUG	31	1							

Data Description:

Field	Contents	Type	Default
i	Debug number (index in DEBUG array)	0 < Integer < 100	None
VALUE	The value for DEBUG(i)	Integer	0

Remarks:

1. No other element in the model may have the same element ID.
2. See table below for actions taken based on the various debug values. Unless otherwise stated, DEBUG(i) = 0 is the default and, for the "print" parameters, no printing is done.

Action Taken For DEBUG(I) Values

I	DEBUG(I)	Action (NOTE:default values are zero)
1	1	Print KIND parameters defined in module PENTIUM_II_KIND to F06 file
2	1	Print constants (parameters) defined in module CONSTANTS_1
3	1	Print machine parameters as determined by LAPACK function DLAMCH
4	1	Do not use BMEAN when calculating membrane quad element stiffness for warped elements
5	1	Print Gauss quadrature abscissas and weight s for plate elements
6	1 2	Print some quad elem data to BUG file (over and above what is printed with Case Control ELDATA) Print some hexa elem data to BUG file (over and above what is printed with Case Control ELDATA)
7	1	Print arrays ESORT1, ESORT2, EPNT, ETYPE in subr ELESORT before/after sorting elems
8	1 2 3	Print grid temperature data in subr TEMPERATURE_DATA_PROC Print elem temperature data in subr TEMPERATURE_DATA_PROC Print both grid and elem temperature data in subr TEMPERATURE_DATA_PROC
9	> 0	Prints debug info on BAR pin flag processing
10	11 or 33 12 or 32 13 or 33 21 or 33 22 or 32	Print data on algorithm to create STF stiffness arrays in subr ESP Print detailed data on algorithm to create STF arrays in subr SPARSE Print template of nonzero terms in KGG if PARAM SETLKTK = 1 or 2 Print data on algorithm to create EMS mass arrays in subr ESP Print detailed data on algorithm to create EMS mass arrays in subr SPARSE
11	1 2 3	Print individual 6x6 rigid body. displacement matrices in basic and global coordinates for each grid Print NGRID by 6 rigid body displacement matrix in global coordinates for the model Print both
12	1	Use area shear factors in computing BAR stiffness matrix regardless of I_{12} value
13	1	Print grid sequence tables in subr SEQ
14	1	Print matrices generated in the rigid element generation subr's
15	1	Print concentrated mass data in subr CONM2_PROC_1
16	1	Use static equivalent instead of work equivalent pressure loads for the QUAD4, TRIA3
17	> 0 > 1	Print some info in subr KGG_SINGULARITY_PROC for grids that have AUTOSPC'd components Do above for all grids (not just ones that have AUTOSPC's)
18	> 0	Print diagnostics in subr QMEM1 regarding checks on the BMEAN matrix satisfying R.B. motion
19	1	Print debug output from subr STOKEN
20	0 1	Use simple solution for GMN if RMM is diagonal. Bypass the simple solution for GMN if RMM is diagonal and use subr SOLVE_GMN instead
21	0 1	Use MATMULT_SFF to multiply stiffness matrix times rigid body displs in STIFF_MAT_EQUIL_CHK Use LAPACK subroutine DSBMV
22	1	Print RBMAT in subr STIFF_MAT_EQUIL_CHK
23	> 0	Do equilibrium checks on stiffness matrix even though model has SPOINT's

24	1 or 3 2 or 3	Print KFSe matrix in subr REDUCE_KNN_TO_KFF Print KSSe matrix in subr REDUCE_KNN_TO_KFF
25	1 or 3 2 or 3	Print PFYS matrix in subr REDUCE_N_FS Print QSYS matrix in subr REDUCE_N_FS
26	1	Print YS matrix (S-set enforcorced displs) in LINK2 (LAPACK)
32	1	Print PL load matrix in LINK3-LAPACK
33	1	Print UL displacement matrix before refining sulotion in LINK3_LAPACK
34	1 or 3 2 or 3	Print ABAND matrix (KLL in band form) before equilibrating it in LINK3 (LAPACK Print ABAND matrix after equilibrating it in LINK3 (LAPACK)
35	1	Print ABAND's decomp matrix (KLL triangular factor) in LINK3 (LAPACK)
36	1	Print grid 6x6 mass for every grid in LINK2
I	**DEBUG(I)**	**Action (NOTE:default values are zero)**
40	1 or 3 2 or 3 1 1	Print banded stiffness matrix ABAND in subr EIG_GIV_MGIV Print banded mass matrix ABAND in subr EIG_GIV_MGIV print RFAC = KLL - sigma*MLL in subr EIG_INV print RFAC = KLL - sigma*MLL in subr EIG_LANCZOS
41	1	Print KLL stiffness matrix in LINK4
42	1	Print MLL stiffness matrix in LINK4
43	1	Print eigenvectors in LINK4 (normally not printed until LINK9)
46	1	Print debug info for Inverse Power eigenvalue extraction
47	1	Print eigenvalue estimates at each iteration in Lanczos
48	1	Do not calculate off-diag terms in generalized mass matrix
49	1	Print diagnostics in ARPACK subroutine DSBAND
55	1 2 3	Print PHIXG in full format in EXPAND_PHIXA_TO_PHIXG Print PHIZG in full format in LINK5 Do both

I	DEBUG(I)	Action (NOTE:default values are zero)
80	> 0	Print LAPACK_S scale factors, in subr EQUILIBRATE, used to equilibrate the stiffness matrices
81	1 2 3	Print data on how subr MATADD_SSS_NTERM determines no. terms to allocate for matrix add Print data on progress of matrix add in subr MATADD_SSS Print data from both subroutines
82	1	Print data on progress of matrix multiply in subr MATMULT_SFF
83	1 2 3	Print data on how subr MATMULT_SFS_NTERM determines no. terms to allocate for matrix multiply Print data on progress of matrix multiply in subr MATMULT_SFS Print data from both subroutines
84	1 2 3	Print data on how subr MATMULT_SSS_NTERM determines no. terms to allocate for matrix multiply Print data on progress of matrix multiply in subr MATMULT_SSS Print data from both subroutines
85	1	Print data on matrix transposition in subr MATTRNSP_SS

86	1 2 3	Print data on how subr PARTITION_SS_NTERM determines no. terms to allocate for matrix partition Print data on progress of matrix partition in subr PARTITION_SS Print data from both subroutines
87	1	Print data on algorithm to convert sparse CRS matrix to sparse CCS in subr SPARSE_CRS_SPARSE_CCS
88	1	Do not write separator line between grids several places(matrix diagonal output, equil check)
89	1	Write row numbers where there are zero diag terms in subroutine SPARSE_MAT_DIAG_ZEROS
91	1	Print Information on how the maximum number of requests for grid or element related outputs is determined. This controls the allocation of memory in LINK9
92	1	Print OLOAD, SPCF, MPCF totals even if global coordinate systems for all grids are not the same
100	> 0 > 1	Check allocation status of allocatable arrays. Also write memory allocated to all arrays to F06 file.
101	> 0 > 1	Write sparse I_MATOUT array in subroutine READ_MATRIX_1. Call subroutine to check I_MATOUT array to make sure that terms are nondecreasing
102	> 0	Print debug info in subroutine MERGE_MAT_COLS_SSS
103	> 0	Do not use MRL (or MLR) in calc of modal participation factors and effective mass
104	> 0	Check if KRRcb is singular
105	> 0	write KLLs matrix to unformatted file
106	> 0	write info on all files in subr WRITE_ALLOC_MEM_TABLE (if 0 only write for those arrays that have memory allocated to them
107	> 0	Write allocated memory in F04 file with 6 decimal points (3 if DEBUG(107) = 0)
108	> 0	Write EDAT table
109	> 0	Write debug info in subr ELMDIS
110	> 0	Write debug info for BUSH elem in subrs ELMDAT1, ELMGM1
111	> 0	Write some debug info on RSPLINE
112	> 0	Write THETAM (plate element material angle) and the location in subr EMG where it was calculated
113	> 0	Write PBARL entries in a special format that has 1 line per PBAR entry
114	> 0	Write debug info in subr OU4_PARTVEC_PROC
115	> 0	Write debug info in subr READ_INCLUDE_FILNAM
116	= 1	Write debug info in Yale subr SFAC
	= 2	Write debug info in Yale subr NFAC
	= 3	Do both

I	**DEBUG(I)**	**Action (NOTE:default values are zero)**
172	> 0	Calc PHI_SQ for the MIN4T based on area weighting of the TRIA3's. Otherwise, use simple average
173	= 1	Write some debug info in subr PARSE_CSV_STRING
	= 2	Write some more detailed data
174	> 0	Print MPFACTOR, MEFFMASS values with 2 decimal places of accuracy rather than 6
175	> 0	Write debug output from subroutine SURFACE_FIT regarding the polynomial fit to obtain element corner stresses from Gauss point stresses
176	> 0	Calculate stresses using element SEi, STEi matrices and displacements rather than from BEi matrices and strains
177	> 0	Print BAR, ROD margins of safety whether or not they would otherwise be
178	= 1	Print info on user key if PROTECTED = 'N'
179	= 1	Print blank space at beg of lines of output for CUSERIN entries in the F06 file
180	> 0	Write debug info to F06 for USERIN elements
181	= 1	Include USERIN RB mass in subr GPWG even though user did not input 3rd matrix (RBM0) on IN4FIL
182	= 1	Print debug data in subr MGGS_MASS_MATRIX for scalar mass matrix
183	= 1	Write some debug data for generating TDOF array
184	> 0	Write L1M data to F06
185	> 0	Let eigen routines find and process all eigenval, vecs found even if NVEC > NDOFL - NUM_MLL_DIAG_ZEROS
186	> 0	Print debug info for pressure loads on faces of solid elements
187	> 0	Write list ao the number of various elastic elements in the DAT file to the F06 file
188	> 0	Do not abort in QPLT3 if KOO is reported to be singular
189	1 2 3	Print messages in subroutine ESP for KE in local coords if element diagonal stiffness < 0 Print these messages in subroutine ESP after transformation to global Do both
190	> 0	Do not round off FAILURE_INDEX to 0 in subr POLY_FAILURE_INDEX
191	= 0	Use temperatures at Gauss points for thermal loads in solid elements
192	> 0	Print some summary info for max abs value of GP force balance for each solution vector
193	= 1 = 2 = 3 = 4 = 5 = 6 = 9 = 100 = 999	call FILE_INQUIRE at end of LINK1 call FILE_INQUIRE at end of LINK2 call FILE_INQUIRE at end of LINK3 call FILE_INQUIRE at end of LINK4 call FILE_INQUIRE at end of LINK5 call FILE_INQUIRE at end of LINK6 call FILE_INQUIRE at end of LINK9 call FILE_INQUIRE at end of MAIN do all of the above
194	1 or 3 2 or 3 3	skip check on CW/CCW numbering of QUAD's 2 or 3 skip check on QUAD interior angles < 180 deg skip both
195	> 0	Print CB OTM matrices to F06 at end of LINK9

196	0 > 0	Matrix output filter SMALL = EPSIL(1) Matrix output filter SMALL = TINY (param defined by user with default = 0.D0)
197	> 0	Print debug info in subr EC_ENTRY_OUTPUT4 which reads Exec Control OUTPUT4 entries
198	> 0	Write debug info in subroutine QPLT3 (for QUAD4 element)
199	> 0	Check matrix times its inverse = identity matrix in several subroutines
201	> 0	Allow SOL = BUCKLING or DIFFEREN to run even if some elements are not coded for these soln's
202	> 0	Calculate rigid body and constant strain sanity checks on strain-displacement matrices
203	> 0	Print debug info in subroutine BAR1 (for the BAR element)
248	> 0	Override fatal error and continue with orthotropic material properties for MIN4T QUAD4
249	> 0	In subroutine BREL1 call code for Timoshenko (BART) instead of Euler (BAR1) BAR element

7.34 EIGR

Description:

Eigenvalue extraction data.

Format:

1	2	3	4	5	6	7	8	9	10
EIGR	SID	METH	F1	F2	NE	ND		CRIT	
	NORM	G	C	SIGMA					

Examples:

EIGR	98	GIV	0.1	20.				1.E-4	
	MAX								

EIGR	25	GIV	15.	20.				1.E-4	
	POINT	471	3						

Data Description:

Field	Contents	Type	Default
SID	Eigenvalue extraction set number	Integer > 0	None
METH	Method for eigenvalue extraction: (GIV, MGIV, INV)	Character	None
F1, F2	Frequency range of interest	Real	0.
NE	Number of estimated eigenvalues in range (not used for GIV)	Integer	0
ND	Number of desired eigenvalues in range (not used for GIV)	Integer	0
CRIT	Orthogonality criteria	Real	0.

NORM	Method of eigenvector renormalization (POINT, MAX, MASS)	Character	None
G	If NORM = POINT, the grid to be used in normalizing eigenvector to 1.0	Integer > 0 or blank	0
C	If NORM = POINT, the component (1-6) at G to be used in normalizing the eigenvector = 1.0	Integer 1-6 or blank	0
SIGMA	Shift eigenvalue (only used for METH = INV. Better convergence is obtained if this is close to the fundamental mode	Real or blank	0.

Remarks:

1. Givens (GIV) or Modified Givens (MGIV) methods of eigenvalue extraction are available. In addition, an Inverse Power (INV) method is also available, but only for the fundamental mode.

2. The EIGR set ID, SID, must be selected in Case Control with the entry METHOD = SID.

3. The three methods of eigenvector renormalization are:

   MASS: eigenvectors are normalized to unit generalized mass (1.0)

   MAX: eigenvectors are normalized to 1.0 for the largest term

   POINT: eigenvectors are normalized such that the value at grid G, component C is 1.0

4. For the GIV method the mass matrix must be positive definite (thus the mass matrix can have no zeros on its diagonal). For the MGIV method, the model must have the stiffness matrix positive definite (thus modes of a model that is not restrained from rigid body motion cannot be obtained).

7.35 EIGRL

Description:

Eigenvalue extraction data for Lanczos method.

Format:

1	2	3	4	5	6	7	8	9	10
EIGRL	SID	F1	F2	N	MSGLVL	NCVFACL	SIGMA	NORM	
	MODE	TYPE							

Examples:

EIGRL	98	0.	50.						

Data Description:

Field	Contents	Type	Default
SID	Eigenvalue extraction set number	Integer > 0	None
F1, F2	Frequency range of interest	Real	0.
N	Number of desired eigenvalues	Integer	0
MSGLVL	Output message level (0 for none, or 1 or 2 for some	Integer	0
NCVFACL	Used to dimension several arrays in the Lanczos method. Must be > 1	Integer	2
SIGMA	Shift eigenvalue	Real	-1.0
NORM	Method of eigenvector renormalization (MAX, MASS)	Character	None
Mode	Lanczos mode for calculating eigenvalues	Integer	2
Type	Lanczos matrix type (DPB, DGB)	Character	DPB

Remarks:

1. The EIGRL set ID, SID, must be selected in Case Control with the entry METHOD = SID

2. Either F1 (and F2) or N must be specified. If both are specified, N will be used.

3. Mode refers to the Lanczos mode type to be used in the solution. In mode 3 the mass matrix, M_{aa}, must be nonsingular whereas in mode 2 the matrix $K_{aa} - \sigma M_{aa}$ must be nonsingular (where σ = SIGMA). See Bulk Data PARAM ART_MASS for use if the mass matrix is singular.

4. TYPE = DPB uses sym storage of the matrices (preferred) whereas DGB stores all nonzero terms.

5. In general, it is not recommended to request more than about 1/4 of the possible modes. This is only a concern for problems with a small number of DOF.

6. SIGMA is the shift eigenvalue. If free modes (rigid body modes) are desired, it should be set to a small negative number (such as between -1.0 and -10.0). The default value is -1.0.

7. The Lanczos solution may not report all negative eigenvalues (especially the lower modes closer to the fundamental, or first, negative eigenvalue). This can be an issue for buckling problems (the opposite direction load is actual buckling load). However, this is not expected to be an issue for natural frequency problems. *For the Lanczos method, negative eigenvalues should be treated with high caution.*

8. The Lanczos method cannot find the highest 3 possible modes in a model. This is only a concern for problems with a small number of DOF.

9. If SIGMA=0.0, models with free modes (rigid body modes) may cause an error in MYSTRAN. If SIGMA > 0.0, an error may also occur. These issues require further investigation.

7.36 FORCE

Description:

Static concentrated force at a grid point.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE	SID	GID	CID	F	N1	N2	N3		

Example:

FORCE	1234	567	89	1000.	1.5	2.5	3.5		

Data Description:

Field	Contents	Type	Default
SID	Load set ID number	Integer > 0	None
GID	ID of the grid at which this concentrated force acts	Integer >0	None
CID	ID of the coordinate system in which the Ni are specified	Integer >= 0	0
F	An overall scale factor for the force	Real	0.
Ni	Components of a vector in the direction of the force	Real	0.

Remarks:

1. The static concentrated force applied to the grid is the vector:

   $$\vec{P} = F\vec{N}$$

   with Ni in fields 6-8 the components of the vector N

2. In order for this load to be used in a static analysis the load set ID must either be selected in Case Control by LOAD = SID, or this load set ID must be referenced on a LOAD Bulk Data entry which itself is selected in Case Control.

3. A blank entry for CID implies the basic coordinate system.

7.37 GRAV

Description:

Gravity load definition.

Format:

1	2	3	4	5	6	7	8	9	10
GRAV	SID	CID	A	N1	N2	N3			

Example:

GRAV	975	246	386.	2.	3.	4.			

Data Description:

Field	Contents	Type	Default
SID	Load set ID number	Integer > 0	None
CID	ID of the coordinate system in which the Ni are specified	Integer >= 0	0
A	Acceleration value	Real	0.
Ni	Components of a vector in the direction of the force	Real	0.

Remarks:

1. GRAV causes a static load to be applied to the complete model that is calculated based on the acceleration vector on the GRAV entry and the mass properties of the model.
2. The acceleration vector applied to the model is the vector:

$$\vec{a} = A\vec{N}$$

with Ni in fields 5-7 the components of the vector N

3. In order for this load to be used in a static analysis the load set ID must either be selected in Case Control by LOAD = SID, or this load set ID must be referenced on a LOAD Bulk Data entry which itself is selected in Case Control.

4. A blank entry for CID implies the basic coordinate system.

7.38 GRDSET

Description:

Default values for the GRID entry.

Format:

1	2	3	4	5	6	7	8	9	10
GRDSET		CID1				CID2	PSPC		

Example:

GRDSET		12				42	245		

Data Description:

Field	Contents	Type	Default
CID1	Default value for the coordinate system ID in which grids will be located for GRID entries which have a blank in this field	Integer >= 0	0
CID2	Default value for the global coordinate system for GRID entries which have a blank in this field	Integer >= 0	0
PSPC	Default value for permanent single point constraints for GRID entries which have a blank in this field	Integers 1-6	0

Remarks:

1. Only one GRDSET entry is allowed in the data file. Any data entered on a GRDSET entry will be used for the corresponding field of any GRID entry that has that field blank. Thus, if the user desires to have CIDi be the basic system on a GRID entry, and a GRDSET entry is present with nonzero value for CIDi, the GRID entry in question must have 0 (not blank) for CIDi.

2. See the GRID entry for remarks on the above fields of this entry.

3. A blank entry for CIDi implies the basic coordinate system.

7.39 GRID

Description:

Grid point definition.

Format:

1	2	3	4	5	6	7	8	9	10
GRID	GID	CID1	X1	X2	X3	CID2	PSPC		

Example:

GRID	58	12	10.	20.	30.	42	245		

Data Description:

Field	Contents	Type	Default
GID	Grid point ID number	Integer > 0	None
CID1	ID of the coordinate system that the Xi are defined in	Integer >= 0	0
Xi	Coordinates of the grid defined in coordinate system CID1	Real	0.
CID2	ID of the global coordinate system for this grid point	Integer >= 0	0
PSPC	Permanent single point constraints at this grid point	Integers 1-6	Blank

Remarks:

1. Grid IDs must be unique among all GRID entries.
2. The word “permanent” in regards to the single point constraints (SPC’s) defined on the GRID entry is merely a designation given to SPC’s defined on GRID entries. The PSPC field does not have to be used. Any, or all, of the zero value (i.e., not enforced displacement) single point constraints used in a model can be specified on Bulk Data SPC or SPC1 entries or as PSPC’s on the GRID entry.
3. A blank entry for CIDi implies the basic coordinate system.

7.40 LOAD

Description:

This entry combines loads defined on FORCE, MOMENT, PLOAD2, GRAV entries.

Format:

1	2	3	4	5	6	7	8	9	10
LOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	(etc)						

Example:

LOAD	12345	1500.	151.5	25	290.2	33	780.3	24	
	2450.1	12							

Data Description:

Field	Contents	Type	Default
SID	Load set ID number	Integer > 0	None
S	An overall scale factor for the load combination	Real	0.
Si	Scale factor for load set Li	Real	0.
Li	Load set ID number for loads defined on FORCE, MOMENT, PLOAD2, GRAV entries	Integer > 0	None

Remarks:

1. The static load applied to the model is the vector:

$$\vec{P} = S\sum_i S_i \vec{P}_{Li}$$

where P_{Li} is the load defined on the FORCE, MOMENT, PLOAD2 or GRAV that has Li load set ID.

2. In order for this load to be used in a static analysis the load set ID must be selected in Case Control by the command LOAD = SID.

3. Any number of continuation entries may be included.

7.41 MAT1

Description:

Linear isotropic material definition.

Format:

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	ALPHA	TREF	GE	
	TA	CA	SA						

Example:

MAT1	10	1.E7		0.33	0.1	2.E-5	21.		
	10000.	20000.	15000.						

Data Description:

Field	Contents	Type	Default
MID	Material ID number	Integer > 0	None
E	Young’s modulus	Real > 0. or blank	See remarks
G	Shear modulus	Real > 0. or blank	See remarks
NU	Poisson’s ratio	Real > 0. or blank	See remarks
RHO	Material mass density	Real > 0. or blank	0.
ALPHA	Coefficient of thermal expansion	Real > 0. or blank	0.

TREF	Reference temperature	Real > 0. or blank	0.
GE	Damping coefficient	Real > 0. or blank	0.
TA	Tension allowable for the material	Real > 0. or blank	0.
CA	Compression allowable for the material	Real > 0. or blank	0.
SA	Shear allowable for the material	Real > 0. or blank	0.

Remarks:

1. MID must be unique among all material property entries.
2. The continuation entry is not required.
3. The following action is taken if one or more of the fields E, G and NU are blank:
   a) If one of E, G or NU is blank it will be calculated using the relationship E = 2(1 + NU)G
   b) If E and NU are blank or if G and NU are blank, these two are set to 0.
   c) If E and G are blank (or zero) a fatal error occurs
4. A warning is given if NU < 0 or if NU > 0.5.
5. A warning is given if if E, G and NU are all input and do not satisfy the relationship:

$$\left| 1 - \frac{E}{2(1+NU)G} \right| < 0.01$$

7.42 MAT2

Description:

Linear anisotropic material definition for 2D plate elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT2	MID	G11	G12	G13	G22	G23	G33	RHO	
	A1	A2	A3	TREF	GE	ST	SC	SS	

Example:

MAT2	10	9.9+6	3.+6	2.+6	10.1+6	3.2+6	8.9+6	.00025	
	2.-5	3.-5	1.5-5	21.	.001	30000.	20000.	25000	

Data Description:

Field	Contents	Type	Default
MID	Material ID number	Integer > 0	None
Gij	Terms in the 3x3 material property matrix	Real	0.
RHO	Material mass density	Real	0.
Ai	Thermal expansion coefficients	Real	0.
TREF	Reference temperature	Real	0.
GE	Structural damping coefficient	Real	0.
ST	Tension stress limit	Real	0.
SC	Compression stress limit	Real	0.
SS	Shear stress limit	Real	0.

Remarks:

1. MID must be unique among all material property entries.

2. The continuation entry is not required.

3. If this entry is used for the transverse shear properties (MID3 on PSHELL) then G13, G23 and G33 are ignored.

4. The stress strain relationship for an element using the MAT2 is:

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} & G_{13} \\ G_{12} & G_{22} & G_{23} \\ G_{13} & G_{23} & G_{33} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \end{Bmatrix} - (T - T_{ref}) \begin{Bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \end{Bmatrix}$$

$$and$$

$$\begin{Bmatrix} \tau_{xz} \\ \tau_{yz} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} \\ G_{12} & G_{22} \end{bmatrix} \begin{Bmatrix} \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix}$$

7.43 MAT8

Description:

Linear orthotropic material definition for plate elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	
	A1	A2	TREF	Xt	Yc	Yt	Yc	S	
	GE	F12	STRN						

Example:

MAT8	10	9.+6	11.+6	0.29	4.+6	3.+6	5.+6	.00258	
	20.-5	22.-5	21.0						

Data Description:

Field	Contents	Type	Default
MID	Material ID number	Integer > 0	None
E1	Elastic modulus in longitudinal direction	Real > 0.	0.
E2	Elastic modulus in lateral direction	Real > 0.	0.
G12	In-plane shear modulus	Real >= 0.	0.
G1Z	Transverse shear modulus in the 1-Z plane	Real >= 0.	0.
G2Z	Transverse shear modulus in the 2-Z plane	Real >= 0.	0.
NU12	Poisson’s ratio	Real >= 0.	0.

RHO	Material mass density	Real >= 0.	0.
A1	Coefficient of thermal expansion in the longitudinal direction	Real >= 0.	0.
A2	Coefficient of thermal expansion in the lateral direction	Real >= 0.	0.
TREF	Reference temperature	Real	0.
Xt		Real > 0.	0.
Xc		Real > 0.	0.
Yt		Real > 0.	0.
Yc		Real > 0.	0.
S		Real > 0.	0.
GE	Damping coefficient	Real > 0.	0.
F12		Real > 0.	0.
STRN	Compression allowable for the material	Real > 0.	0.

Remarks:

1. MID must be unique among all material property entries.
2. The continuation entries are not required.
3. If G1Z and G2Z are zero (or blank) transverse shear flexibility is zero (infinite transverse shear stiffness).

7.44 MAT9

Description:

Linear anisotropic material definition for 3D solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT9	MID	G11	G12	G13	G14	G15	G16	G22	
	G23	G24	G25	G26	G33	G34	G35	G36	
	G44	G45	G46	G55	G56	G66	RHO	A1	
	A2	A3	A4	A5	A6	TREF	GE		

Example:

MAT9	10	8.+6	4.+4	3.2+6	2.5+6			9.+6	
					10.+6				
	4.+6			5.+6		3.+6	.003	20.-5	
	22.-5	18.-5							

Data Description:

Field	Contents	Type	Default
MID	Material ID number	Integer > 0	None
Gij	Elements of the 6x6 material matrix	Real > 0.	0.
RHO	Material mass density	Real >= 0.	0.
AI	Coefficients of thermal expansion	Real >= 0.	0.
TREF	Reference temperature	Real	0.

GE	Damping coefficient	Real > 0.	0.

Remarks:

1. MID must be unique among all material property entries.
2. The first two continuation entries are required but the third continuation entry is not required.
3. The Gij are the transformation of strains to stresses as in:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} & G_{13} & G_{14} & G_{15} & G_{16} \\ & G_{22} & G_{23} & G_{24} & G_{25} & G_{26} \\ & & G_{33} & G_{34} & G_{35} & G_{36} \\ & & & G_{44} & G_{45} & G_{46} \\ & sym & & & G_{55} & G_{56} \\ & & & & & G_{66} \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix}$$

7.45 MOMENT

Description:

Static concentrated moment at a grid point.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT	SID	GID	CID	M	N1	N2	N3		

Example:

MOMENT	1234	567	89	1000.	1.5	2.5	3.5		

Data Description:

Field	Contents	Type	Default
SID	Load set ID number	Integer > 0	None
GID	ID of the grid at which this concentrated moment acts	Integer >0	None
CID	ID of the coordinate system in which the Ni are specified	Integer >= 0	0
M	An overall scale factor for the moment	Real	0.
Ni	Components of a vector in the direction of the moment	Real	0.

Remarks:

1. The static concentrated moment applied to the grid is the vector:

$$\vec{P} = M\vec{N}$$

with Ni in fields 6-8 the components of the vector N

2. In order for this load to be used in a static analysis the load set ID must either be selected in Case Control by LOAD = SID, or this load set ID must be referenced on a LOAD Bulk Data entry which itself is selected in Case Control.

3. A blank entry for CID implies the basic coordinate system.

7.46 MPC

Description:

Multi point constraints define a linear dependence of one degree of freedom (that becomes a member of the M-set) on other degrees of freedom.

Format:

1	2	3	4	5	6	7	8	9	10
MPC	SID	G1	C1	D1	G2	C2	D2		
		G3	C3	S3	G4	C4	D4		
		G6	C5	D6	etc…				

Example:

As an example, consider the following equation relating several degrees of freedom (in global coordinates):

$$1.2w_{101} + 4.5v_{201} - 0.63\theta_{y_{623}} + 12.7\theta_{z_{76}} = 0$$

where w_{101} is the displacement in the global z direction at grid 101, v_{201} is the displacement in the global y direction at grid 201, and the remaining two terms are the rotation about the global y and z directions at grids 623 and 76 respectively. Assuming that w_{101} has been chosen as the M-set degree of freedom for this MPC equation, the input would be:

MPC	56	101	3	1.2	201	2	4.5		
		623	5	-.63	76	6	12.7		

Data Description:

Field	Contents	Type	Default
SID	ID number of the multi point constraint set	Integer > 0	None

Gi	ID numbers of the grids involved in the constraint. Grid G1, component C1 is, by definition, the dependent (M-set) degree of freedom	Integer > 0	None
Ci	Component numbers at grids Gi involved in the MPC equation	Integers 1-6	None
Di	The value for coefficient D for grid Gi, component Ci	Real	0.

Remarks:

1. Multi point constraint sets must be selected in Case Control with the entry MPC = SID in order for them to be applied.

2. Degrees of freedom defined as dependent on MPC entries will be members of the M-set and cannot be defined as being members of any other mutually exclusive set.

3. G1/C1 is the degree of freedom eliminated (M-set) due to the MPC equation and the remaining terms in the MPC equation can be for degrees of freedom belonging to any displacement set.

7.47 MPCADD

Description:

Combine multi-point constraint sets defined on MPC entries.

Format:

1	2	3	4	5	6	7	8	9	10
MPCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	(etc)						

Example:

MPCADD	283	11	74	123	564				

Data Description:

Field	Contents	Type	Default
SID	Multi-point constraint set ID number	Integer > 0	None
Si	Set IDs of MPC Bulk Data entries	Integer > 0	None

Remarks:

1. Multi-point constraint sets must be selected in Case Control with the entry MPC = SID in order for them to be applied.
2. All multi-point constraints specified on MPC entries whose set IDs are the Si on the MPCADD will be applied to the model if MPC = SID is in Case Control.

7.48 OMIT

Description:

Define degrees of freedom to go into the omit set (O-set).

Format:

1	2	3	4	5	6	7	8	9	10
OMIT	G1	C1	G2	C2	G3	C3	G4	C4	

Example:

OMIT	19	1	28	2345	37	124	46	134	

Data Description:

Field	Contents	Type	Default
Gi	ID numbers of grids	Integer > 0	None
Ci	Displacement component numbers	Integers 1-6	None

Remarks:

1. The degrees of freedom defined by grids GI, components Ci will be placed in the mutually exclusive O-set. These degrees of freedom cannot have been defined to be in any other mutually exclusive set (i.e.. M, S or A sets).
2. If OMIT or OMIT1 are present in the data file, then all degrees of freedom not specified on these entries and also not in the M or S sets will be placed in the A-set. If both ASET (or ASET1) and OMIT (or OMIT1) are present, then all degrees of freedom not in the M and S sets must be explicitly defined on ASET (or ASET1) and OMIT (or OMIT1)
3. Up to four pairs of Gi, Si can be specified on one OMIT entry. For more pairs, use additional OMIT entries (i.e. there is no continuation entry for OMIT).

7.49 OMIT1

Description:

Define degrees of freedom to go into the omit set (O-set).

Format No. 1:

1	2	3	4	5	6	7	8	9	10
OMIT1	C	G1	G2	G4	G4	G5	G6	G7	
	G8	G9	(etc)						

Format No. 2:

1	2	3	4	5	6	7	8	9	10
OMIT1	C	G1	THRU	G2					

Example:

OMIT1	135	17934	THRU	19012					

Data Description:

Field	Contents	Type	Default
Gi	ID numbers of grids. G2 > G1	Integer > 0	None
C	Displacement component numbers	Integers 1-6	None

Remarks:

1. In Format No. 2, all grids in the range G1 through G2 will have component C defined in the O-set.

2. The degrees of freedom defined by grids GI, components C will be placed in the mutually exclusive O-set. These degrees of freedom cannot have been defined to be in any other mutually exclusive set (i.e.. M, S or A sets).

3. If OMIT or OMIT1 are present in the data file, then all degrees of freedom not specified on these entries and also not in the M or S sets will be placed in the A-set. If both ASET (or ASET1) and OMIT (or OMIT1) are present, then all degrees of freedom not in the M and S sets must be explicitly defined on ASET (or ASET1) and OMIT (or OMIT1)

7.50 PARAM

Description:

Provide values, other than default values, for parameters that control options during execution.

Format:

1	2	3	4	5	6	7	8	9	10
PARAM	NAME	V1	V2	V3	V4				

Example:

PARAM	PRTDOF	2							

Data Description:

Field	Contents	Type	Default
NAME	Parameter name	Char	None
Vi	Values for the parts of the parameter	Char, Integer or real	Various

Remarks:

1. See table below for a list of the various parameters and what action is taken based on their values. Unless otherwise stated, only value V1 is used. The parameter name always goes in field 2 and V1 always goes in field 3. When there is more than one Vi, the table explicitly states in what fields the Vi go.

Parameter Name	Data Type	Function of Parameter NOTE: Default values of parameters are: N for Char, 0 for Int and 0.0 for real
ARP_TOL	Real	Default = $1x10^{-6}$ Tolerance to use in Lanczos eigenvalue extraction method for convergence
ART_KED (for diff stiffness – not fully implemented)	Char	Field 3: ART_KED, default = N. If Y add artificial stiff to diag of KED stiff matrix Field 4: ART_TRAN_MASS: value for translation degrees of freedom, default $1x10^{-6}$ Field 5: ART_ROT_MASS: value for translation degrees of freedom, default $1x10^{-6}$
ART_MASS	Char	Field 3: ART_MASS, default = N. If Y add artificial mass to diag of MGG mass matrix Field 4: ART_TRAN_MASS: value for translation degrees of freedom, default $1x10^{-6}$ Field 5: ART_ROT_MASS: value for rotation degrees of freedom, default $1x10^{-6}$
AUTOSPC	Char Real Int Char Char	Field 3: AUTOSPC value, default = Y (AUTOSPC), N turns AUTOSPC off. Field 4: AUTOSPC_RAT, default = $1x10^{-8}$ (see Section 3.4.1.1) Field 5: AUTOSPC_NSET, default = 1 (see Section 3.4.1.1) Field 6: AUTOSPC_INFO, default = N. If Y then print messages about the AUTOSPC's Field 7: AUTOSPC_SPCF, default = N. If Y print AUTOSPC forces of constraint
BAILOUT	Int	Default = 1 If > 0 quit if a singularity in decomposing a matrix is detected. If <= 0 do not quit
CBMIN3	Real	Default = 2.0 CBMIN3 is the constant C_B used in tuning the shear correction factor in Ref 3 for the TRIA3 plate element. The default 2.0 is the value suggested by the author.
CBMIN4	Real	Default = 3.6 CBMIN4 is the constant C_B used in tuning the shear correction factor in Ref 4 for the QUAD4 plate element (QUAD4TYP = 'MIN4 '). See Ref 4

CBMIN4T	Real	Default = 3.6 CBMIN4T is the constant C_B used in tuning the shear correction factor in Ref 4 for the QUAD4 plate element (QUAD4TYP = 'MIN4T').
CHKGRDS	Char	Default = Y. If N do not check that all grids for all elements exist
CRS_CCS	Char	Default = CRS (compressed row storage of matrices). Also can be CCS
CUSERIN	Char Int Int Int Int Char Int	If this parameter is present, Bulk Data entries for Craig-Bampton (CB) reduced models will be written to the F06 file as a CUSERIN element (including grids, coord sys, etc) Field 3: element ID, default = 9999999 Field 4: property ID default = 9999999 Field 5: start index for SPOINT's to represent modes of the CB model, default = 1001 Field 6: IN4 file # on the PUSERIN entry for this CUSERIN elem, default = 9999999 Field 7: Set-ID for CUSERIN elem (typically the "R", or boundary, set), def is blank field Field 8: Format for how to write the comp numbers (1 thru 6) for each grid of the CUSERIN elem. If 0, write them in compact form (e.g. 1356). If > 0 write them in expanded form (1 3 56), default = 0
DARPACK	Int	Default = 2 how many extra modes to find above EIG_N2 on the EIGRL entry. These few highest mode are not used due to difficulty with getting good GP force balance.
DELBAN	Int	Default 1. If equal to 1 delete the bandit output files on exit
EIGESTL	Int	Default 5000 For eigenvalue problems by the Lanczos method, if the number of L-set DOF's exceed EIGESTL the method for specifying the search range will be changed from F1 to F2 to N (see EIGRL Bulk Data entry) to avoid excessive run times (since the code to estimate the number of eigens in the F1 to F2 range can be excessive).
EIGNORM2	Char	Default = N. if 'Y' then eigenvectors will be renormalized a last time by multiplying by a set of scale factors (1 per eigenvector) supplied in a file with the same name as the input file and extension 'EIN' (if it exists)

ELFORCEN	Char	Default = GLOBAL If ELFORCEN = GLOBAL, and nodal forces have been requested in Case Control, they will be output in the global coordinate system. If ELFORCEN = BASIC, and nodal forces have been requested in Case Control, they will be output in the basic coordinate system. If ELFORCEN = LOCAL, and nodal forces have been requested in Case Control, they will be output in the local element coordinate system.
EPSERR	Char	Default = Y. If N, do not calculate the NASTRAN like "epsilon error estimate"
EPSIL	Real	There are 3 EPSIL(i) values each of which requires a separate PAPAM EPSIL Bulk Data entry with the index (i) in field 3 and EPSIL(i) value in field 4. These are small numbers used in MYSTRAN for the purposes indicated below: 1) EPSIL(1) (default = $1x10^{-15}$) is used in MYSTRAN such that, in any real number comparisons, any real number whose absolute magnitude is less than EPSIL(1) is considered to be zero. If no PARAM EPSIL 1 entry is in the data file then this value is reset (from the default) in LINK1 to a value based on machine precision calculated using LAPACK BLAS function DLAMCH. If the user has a PARAM EPSIL 1 entry, this value will be used for EPSIL(1) instead of the LAPACK machine precision. 2) Currently not used 3) EPSIL(3) is used in the Inverse Power method of eigenvalue extraction to test convergence of an eigenvalue. The default value (% change) is $1x10^{-5}$ % 4) EPSIL(4) is used to calculate the maximum warp for quadrilateral plate elements, above which a warning message will be written. This maximum warp is EPSIL(2) times the average length of the quadrilateral's two diagonals. The default for EPSIL(2) is $1.x10^{-1}$. 5) EPSIL(5) (default $1.x10^{-6}$) is used in BAR and ROD margin of safety calculations. If a stress magnitude is less than EPSIL(5) a $1.x10^{10}$ margin of safety will printed out for that stress (in other words, an infinite margin of safety) EPSIL(6) (default $1.x10^{-15}$) is used in BAR margin of safety calculations

EQCHECK	Int Int Int Int Int Int Real Char	Field 3: Default = 0 (basic origin) or reference grid to use in calculating the rigid body displacement matrix for the equilibrium check Field 4: If nonzero, do equilibrium check on the G-set Field 5: If nonzero, do equilibrium check on the N-set Field 6: If nonzero, do equilibrium check on the F-set Field 7: If nonzero, do equilibrium check on the A-set Field 8: If nonzero, do equilibrium check on the L-set The value in fields 4-8 can be: 1: print loads due to rigid body displacements 2: print strain energy due to rigid body displacements 3: print both Field 9: EQCHK_TINY, default = $1x10^{-5}$. I Do not print grid forces smaller than this Field 10: Default = N. If Y, normalize the grid forces on diagonal stiffness
FILES		Equivalent to PRTALL
GRDPNT	Int	Default = -1. If not -1 then the value is interpreted as a grid number If GRDPNT /= 0, calculate total mass properties of the model relative to the basic coordinate system origin or relative to the specified grid.
GRIDSEQ	Char Char Char	Field 3: GRIDSEQ value (default = BANDIT). Other values are GRID and INPUT. BANDIT is automatic grid sequencing. GRID is sequencing in grid ID numerical order. INPUT is sequencing in the grid input order. Field 4: SEQQUIT, default = N. If Y, then quit in the sequence processor if BANDIT did not run correctly. Field 5: SEQPRT, default = N. If Y, print SEQGP card images generated by BANDIT to the F06 output file
HEXAXIS	Char	'SIDE12', use side 1-2 as the local elem x axis. 'SPLITD' (default), use angle that splits the 2 diags to define the elem x axis
IORQ1M	Int	Default = 2 Gaussian integration order for membrane direct stress terms for the QUAD4, QUAD4K quadrilateral elements
IORQ1S	Int	Default = 1 Gaussian integration order for membrane shear stress terms for all quad elements

IORQ1B	Int	Default = 2 Gaussian integration order for bending stress terms for the QUAD4K element
IORQ2B	Int	Default = 2 Gaussian integration order for bending stress terms for the QUAD4 element
IORQ2T	Int	Default = 3 Gaussian integration order for transverse shear stress terms for the QUAD4 element
ITMAX	Int	Default = 5 Max number of iterations in refining the solution when parameter UREFINE = Y
KLLRAT	Char	Default = Y to tell whether to calc ratio of max/min KLL diagonal terms
KOORAT	Char	Default = Y to tell whether to calc ratio of max/min KOO diagonal terms
LANCMETH	Char	Procedure to use for Lanczos eigenvalue extraction (Currently only ARPACK is available but it does require matrices to be stored in band form which can require an excessive amount of memory for large problems)
MATSPARS	Char	If = Y (default), use sparse matrix routines for add/multiply in all matrix operations. If N, use full matrix add/multiply (not recommended)
MAXRATIO	Real	Default =$1X10^7$ Max value of matrix diagonal to factor diagonal before messages are written and BAILOUT tested for aborting run
MEFMCORD	Int	Default = 0. The coordinate system in which to calculate modal mass and participation factors
MEFMLOC	Char	Reference location for calculating modal effective mass in Craig-Bampton (SOL 31) analyses. This only affects the rotational modal effective masses. Field 3 can be GRDPNT, GRID or CG: If field 3 = GRDPNT (default): ref point is the same as the one for PARAM GRDPNT If field 3 = CG: use the model center of gravity as the reference point If field 3 = GRID: use the grid point number in field 4 as the reference point Field 4: MEFMGRID (grid to use when field 3 is GRID)

MEMAFAC	Int	Default = 0.9. Factor to multiply the size request of memory to be allocated when looping to find an allowable amount of memory to allocate. Used when the initial request for memory (in subrs ESP or EMP) cannot be met and we know that the request is conservative.
MIN4TRED	Char	Default = STC. Defines the method for how the 5th node of the MIN4T element is reduced out (to get a 4 node quad element). STC (default) is static condensation. B54 (not implemented as of Version 3.0) uses a method developed by the element author (see Reference section, this manual for the element formulation paper)
MPFOUT	Char	(1) '6' (default) indicates to output modal participation factors (MPF) relative to the 6 DOF's at grid MEFMGRID (see PARAM MEFMLOC) (2) 'R' indicates to output MPF's for all of the R-set DOF's individually
MXALLOCA	Int	Default = 10. Max number of attempts to allow when trying to allocate memory in subroutine ALLOCATE_STF_ARRAYS
MXITERI	Int	Default = 50. Max number of iterations to use in the Inverse Power eigenvalue extraction method
MXITERL	Int	Default = 50. Max number of iterations to use in the Lanczos eigenvalue extraction method
OTMSKIP	Int	Number of lines to skip between segments of OTM text file descriptors
PBARLDEC	Int	Default = 5. Number of decimal digits when writing PBAR equivalents for PBARL entry real data
PBARLSHR	Char	Default = Y. Include K1, K2 for PBAR equiv to PBARL BAR properties
PCHSPC1	Char Int Char	Field 3: PCHSPC1 value (default = N, do not punch SPC1 card images for constraints generated by the AUTOSPC feature, use Y to punch these) Field 4: SPC1SID value (default = 9999999, the set ID to put on the SPC1 card images) Field 5: SPC1QUIT value (default = N, do not stop after SPC!'s are punched, or Y to stop processing)
PCMPTSTM	Real	Factor to multiply composite ply thickness for effective shear thickness
PCOMPEQ	Int	Default = 0. Indicator to write equiv PSHELL, MAT2 to F06 for PCOMP's. If > 0, write the equivalent PSHELL amd MAT2 Bulk Data entries for the PCOMP. If > 1 also write the data in a format with a greater number of digits of accuracy.
POST	Int	If = -1 then write FEMAP neutral file for post processing of MYSTRAN outputs

PRTANS	Char	Default = NO If YES, an ANS file will be created and populated with the data requested in the case control section. *This is a MYSTRAN exclusive param, not present in Nastran.
PRTALL	Char	Default = NO If YES, a F06, an OP2, a NEU, and an ANS file will be created and populated with the data requested in the case control section. If YES, this parameter will override any other output control parameters which suppress file data output. If NO, this param does not override any other output requests or output control parameters. *This is a MYSTRAN exclusive param, not present in Nastran.
PRTBASIC	Int	If = 1 print grid coordinates in the basic coordinate system
PRTCGLTM	Int	If = 1 print CB matrix for C.G. LTM loads
PRTCONN	Int	If = 1, print table of elements connected to each grid. If 2, more detailed data
PRTCORD	Int	If PRTCORD = 1 print coordinate system transformation data
PRTDISP	Int	PRTDISP(I), I=1-5 go in fields 3-7 of the PARAM PRTDISP entry that prints displacement matrices for various displacement sets: V1 = PRTDISP(1) = 1 print UG V2 = PRTDISP(2) = 1 or 3 print UN, 2 or 3 print UM V3 = PRTDISP(3) = 1 or 3 print UF, 2 or 3 print US V4 = PRTDISP(4) = 1 or 3 print UA, 2 or 3 print UO V5 = PRTDISP(5) = 1 or 3 print UL, 2 or 3 print UR
PRTDLR	Int	If = 1, the DLR matrix will be printed

PRTDOF	Int	If PRTDOF = 1 or 3 print TDOF table, in grid point ID numerical order, which gives a list of the degree of freedom numbers for each displacement set (size is number of degrees of freedom x number of displacement sets) If PRTDOF = 2 or 3 print TDOF table, in degree of freedom numerical order, which gives a list of the degree of freedom numbers for each displacement set (size is number of degrees of freedom x number of displacement sets)
PRTF06	Char	Default = NO If YES, a F06 file will be created and populated with the data requested in the case control section. *This is a MYSTRAN exclusive param, not present in Nastran. *A "zero" is used in "PRTF06"
PRTFOR	Int	PRTFOR(I), I=1-5 go in fields 3-7 of the PARAM PRTFOR entry that prints sparse force matrices for various displacement sets: V1 = PRTFOR(1) = 1 print sparse PG V2 = PRTFOR(2) = 1 or 3 print sparse PN, 2 or 3 print PM V3 = PRTFOR(3) = 1 or 3 print sparse PF, 2 or 3 print PS V4 = PRTFOR(4) = 1 or 3 print sparse PA, 2 or 3 print PO V5 = PRTFOR(5) = 1 or 3 print sparse PL, 2 or 3 print PR
PRTGMN	Int	If PRTGMN = 1, print GMN matrix
PRTGOA	Int	If PRTGOA = 1, print GOA matrix
PRTHMN	Int	If = 1 print HMN constraint matrix
PRTIFLTM	Int	If = 1 print CB matrix for Interface Forces LTM
PRTKXX	Int	If = 1 print CB matrix KXX
PRTMASSD	Int	Same as PRTMASS, except only print diagonal terms

PRTMASS	Int	PRTMASS(I), I=1-5 go in fields 3-7 of the PARAM PRTMASS entry that prints sparse mass matrices for various displacement sets: V1 = PRTMASS(1) = 1 print sparse MGG V2 = PRTMASS(2) = 1 or 3 print sparse MNN, 2 or 3 print MNM, MMM V3 = PRTMASS(3) = 1 or 3 print sparse MFF, 2 or 3 print MFS, MSS V4 = PRTMASS(4) = 1 or 3 print sparse MAA, 2 or 3 print MAO, MOO V5 = PRTMASS(5) = 1 or 3 print sparse MLL, 2 or 3 print MLR, MRR
PRTMXX	Int	If = 1 print CB matrix MXX
PRTNEU	Char	Default = NO If YES, a NEU file will be created and populated with the data requested in the case control section. *This is a MYSTRAN exclusive param, not present in Nastran.
PRTOP2	Char	Default = NO If YES, an OP2 file will be created and populated with the data requested in the case control section. *This is a MYSTRAN exclusive param, not present in Nastran.
PRTOU4	Int	If > 0 write all OU4 (OUTPUT4) matrices to F06 file
PRTPHIXA	Int	If = 1 print CB matrix PHIXA
PRTPHIZL	Int	If = 1 print CB matrix PHIZL
PRTPSET	Int	If > 0 print the OUTPUT4 matrix partitioning vector sets
PRTQSYS	Int	If = 1 print matrix QSYS
PRTRMG	Int	If PRTRMG = 1 or 3, print constraint matrix RMG If PRTRMG = 2 or 3, print partitions RMN and RMM of constraint matrix RMG
PRTSCP	Int	If PRTSCP = 1 print data generated in the subcase processor
PRTSTIFD	Int	Same as PRTSTIFF, except only print diagonal terms

PRTSTIFF	Int	Defaults = 0 for PRTSTIFF(I), I=1-5 which go in fields 3-7 of the PARAM PRTSTIFF entry that prints sparse stiffness matrices for various displacement sets: V1 = PRTSTIFF(1) = 1 print sparse KGG V2 = PRTSTIFF(2) = 1 or 3 print sparse KNN, 2 or 3 print KNM, KMM V3 = PRTSTIFF(3) = 1 or 3 print sparse KFF, 2 or 3 print KFS, KSS V4 = PRTSTIFF(4) = 1 or 3 print sparse KAA, 2 or 3 print KAO, KOO V5 = PRTSTIFF(5) = 1 or 3 print sparse KLL, 2 or 3 print KLR, KRR
PRTTSET	Int	If PRTSET = 1 print TSET table which gives the character name of the displacement sets that each degree of freedom belongs to (size is number of grids x 6)
PRTUO0	Int	If = 1 print UO0
PRTUSET	Int	If > 0 print the user defined set (U1 or U2) definitions
PRTYS	Int	If = 1 print matrix YS
Q4SURFIT	Int	Default = 6. Polynomial order for the surface fit of QUAD4 stress/strain when stresses are requested for other than corner locations
QUAD4TYP	Char	Formulation for the QUAD4 element Default = MIN4: Reissner-Mindlin element This element may exhibit spurious modes See Section 3.2 for a detailed description of capabilities MITC4+: Mixed Interpolation of Tensorial Components (Reference 15). The MITC4+ is a newer version of the MITC4 element that offers further improvements in convergence behavior. See Section 3.2 for a detailed description of capabilities
QUADAXIS	Char	Default = SIDE12 This determines how the quad element local x axis is defined. 'SIDE12' means that the axis between grids 1 and 2 of the quad define the local x axis. 'SPLITD' means that the axis is defined as the direction that splits the angle between the quad diagonals
RCONDK	Char	If RCONDK = Y, then LAPACK calculates the condition number of the A-set stiffness matrix. This is required if LAPACK error bounds on the A-set displacement solution are desired. This can require significant solution time.

SKIPMGG	Char	Default = N. 'Y', 'N' indicator to say whether to skip calculation of MGG, KGG in which case MGG, KGG will be read from previously generated, and saved, files (LINK1L for KGG, LINK1R for MGG)
SOLLIB	Char	Field 3: Denotes which library to use for matrix decomposition and equation solution. Options are: 1) SPARSE: default (matrices stored with only nonzero terms) 2) BANDED: (matrices stored in band form. Uses LAPACK/ARPACK routines) Field 4: (only if SPARSE SOLLIB) denotes which SPARSE library to use: 5. SUPERLU (default) uses the SuperLU method of sparse matrix decomp and solve.
SORT_MAX	Int	Default = 5 Max number of times to run algorithm when sorting arrays before fatal message.
SPARSTOR	Char	Default = SYM If SYM, symmetric matrices are stored with only the terms on and above the diagonal. If NONSYM all terms are stored. SYM requires less disk storage but NONSYM can save significant time in sparse matrix partitioning and multiply operations.
STR_CID	Int	Default = -1. Indicator for the coordinate system to use ID for element stress, strain and engineering force output: -1 is local element coordinate system (default) 0 is basic coordinate system j (any other integer) is a defined coordinate system for output
SUPINFO	Char	Default = Y Indicator of whether some information messages should be suppressed in the F06 output file. N indicates to suppress, Y indicates to not suppress messages in the file.
SUPWARN	Char	Default = Y Indicator of whether warning messages should be suppressed in the F06 output file. N indicates to suppress, Y indicates to not suppress messages in the file.

THRESHK	Real	Default = 0.1 User defined value for the threshold in deciding whether to equilibrate the A-set stiffness matrix in LAPACK subroutine DLAQSB. Default value 0.1, LAPACK suggests
TINY	Real	Do not print matrix values whose absolute value is less than this parameter value
TSTM_DEF	Real	Default = 5/6 = 0.833333 Value for TS/TM on PSHELL Bulk data entry when that field on the PSHELL is blank
USETSTR	Char	Requests output of the internal sequence order for displacement sets (e.g. G-set, etc). See section 3.6 for a discussion of displacement sets. In addition to the sets in section 3.7, the user displacement sets U1 and U2 (see Bulk Data entry USET and USET1) can also have the internal sort order output to the F06 file. As an example, to obtain a row oriented tabular output of the internal sort order for the R-set, include the Bulk data entry: PARAM, USETSTR, R
USR_JCT	Int	User supplied value for JCT - used in shell sort subroutines. If USR_JCT = 0, internal values for JCT will be used in the shell sort.
WINAMEM	Real	Default = 2.0 GB. Max memory Windows allows for any array. If it is exceeded, a message is printed out and execution is aborted. This is used to avoid a failure which aborts MYSTRAN catastrophically (due to a system fault).
WTMASS	Real	Default = 1.0 Multiplier for mass matrix after the model total mass is output in the Grid Point Weight Generator (GPWG). This allows user to input mass terms as weight to get model mass properties in weight units and then to convert back to mass units after the GPWG has run. For example, if the model units are lb-sec^2/inch for mass and inches for length and the input data file has lb for “mass” (read weight), then 1/386, or 0.002591 would be the value for WTMASS needed to convert the “mass” matrix from weight units to mass units.

7.51 PARVEC

Description:

Defines a partitioning vector to be used in partitioning an OUTPUT4 matrix. See the Exec Control statements OUTPUT4 and PARTN.

Format:

1	2	3	4	5	6	7	8	9	10
PARVEC	NAME	G1	C1	G2	C2	G3	C3		

Example:

PARVEC	COLVEC	101	3	201	2				

Data Description:

Field	Contents	Type	Default
NAME	Name of a row or column partitioning vector specified in a PARTN Exec Control command	Char	None
GI	ID numbers of the grids that will be partitioned	Integer > 0	None
C	Component numbers at grids Gi that will be partitioned	Integers 1-6	None

Remarks:

1. The Gi, Ci must be members of the displacement set for the matrix being partitioned. For example, if the OUTPUT4 matrix being partitioned is K_{RL} the row partitioning vector grid/component values must be members of the R-set and the column partitioning vector must be a member of the L-set.

7.52 PARVEC1

Description:

Defines a partitioning vector to be used in partitioning an OUTPUT4 matrix. See the Exec Control statements OUTPUT4 and PARTN.

Format No. 1:

1	2	3	4	5	6	7	8	9	10
PARVEC1	NAME	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	(etc)					

Format No. 2:

1	2	3	4	5	6	7	8	9	10
PARVEC1	U1	C	G1	THRU	G2				

Examples:

PARVEC1	52	135	1001	1002	103	1004	2001	2002	
	2003	2004							

PARVEC1	52	135	1001	THRU	1004				

Data Description:

Field	Contents	Type	Default
NAME	Name of a row or column partitioning vector specified in a PARTN Exec Control command	Char	None
Gi	ID numbers of the grids that will be partitioned	Integers 1-6	None

C	Component numbers at grids Gi that will be partitioned	Integer > 0	None

Remarks:

1. The Gi, Ci must be members of the displacement set for the matrix being partitioned. For example, if the OUTPUT4 matrix being partitioned is K_{RL} the row partitioning vector grid/component values must be members of the R-set and the column partitioning vector must be a member of the L-set.

7.53 PBAR

Description:

Property definition for BAR element.

Format:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	MPL		
	Y1	Z1	Y2	Z2	Y3	Z3	Y4	Z4	
	K1	K2	I12	CT					

Example:

PBAR	5	2	1.44	.144	.1	.005	0.1		
	0.5	0.6	-0.5	0.6	-0.5	-0.6	0.5	-0.6	
	.833	.833							

Data Description:

Field	Contents	Type	Default
PID	Property ID number	Integer > 0	None
MID	Material ID number	Integer > 0	None
A	Bar cross-sectional area	Real	0.
I1	Section moment of inertia about the element z axis (bending in element plane xy)	Real	0.
I2	Section moment of inertia about the element y axis (bending in element plane xz)	Real	0.
J	Torsional constant	Real	0.

MPL	Mass per unit length	Real	0.
Yi, Zi	Element y, z coordinates, in the bar cross-section, of four points at which to recover stresses	Real	0.
K1, K2	Area factors for shear in element planes xy and xz respectively	Real	0.
I12	Section cross-product of inertia	Real	0.
CT	Torsional stress recovery coefficient	Real	0

Remarks:

1. PID must be unique among all PBAR, PBARL property ID's
2. Neither continuation entry is required
3. The shear center and neutral axis of the beam coincide.
4. See Figure 4-3 for bar element axes
5. Torsional stress is CT/J times the torsion load in the CBAR
6. K1 and K2 are used to calculate the transverse shear flexibility of the bar. For infinite shear stiffness (zero shear flexibility), K1 and K2 must be infinite by beam element theory. In order to implement this, and avoid dealing with very large numerical values for K1 and K2, MYSTRAN interprets zero K1 and K2 to indicate zero transverse shear flexibility

7.54 PBARL

Description:

Property definition for a CBAR element via reference to a cross-section shape (whose dimensions are specified).

Format:

1	2	3	4	5	6	7	8	9	10
PBARL	PID	MID		TYPE					
	DIM1	DIM2	DIM3	DIM4	DIM5	DIM6	DIM7	DIM8	
	DIM9	etc	NSM						

Example:

PBARL	5	2		CHAN					
	0.5	1.6	0.2	0.1					

Data Description:

Field	Contents	Type	Default
PID	Property ID number	Integer > 0	None
MID	Material ID number	Integer > 0	None
TYPE	Cross section type	Real	0.
DIMi	Cross-section dimensions	Real	0.
NSM	Nonstructural mass per unit length	Real	0.

Remarks:

1. PID must be unique among all PBAR, PBARL property ID's

2. If ECHO /= NONE the equivalent PBAR entries will be printed in the F06 file

3. Allowable cross-section types are:

BAR	BOX	BOX1	CHAN	CHAN1	CHAN2
CROSS	H	HAT	HEXA	I	I1
ROD	T	T1	T2	TUBE	Z

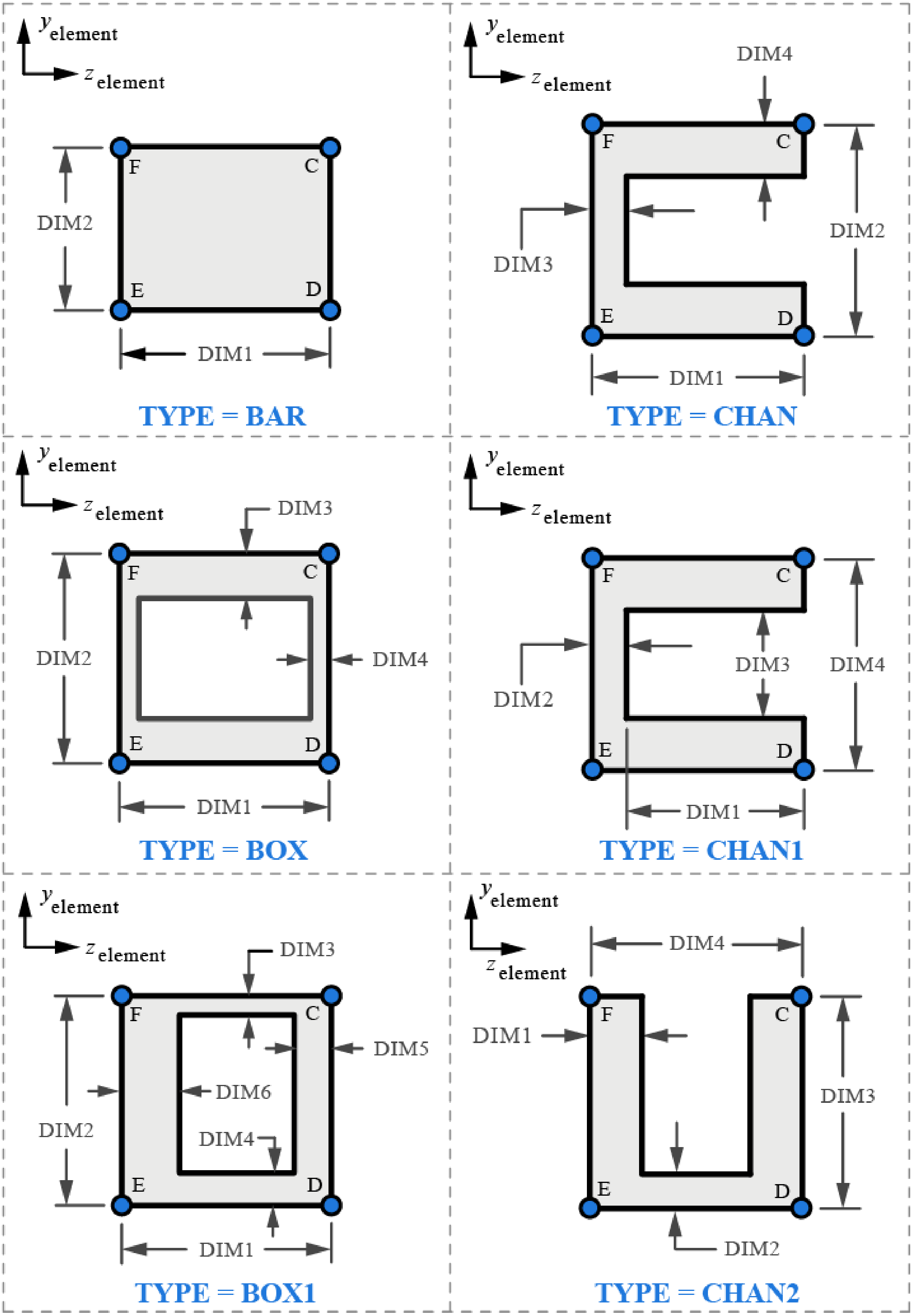
4. The figures on the following 3 pages show the above cross-section types along with the dimension variables (DIMi) and the cross-section axes. The axes are centered on the cross-section shear center. Points C, D E F are where stresses will be recovered.

5. Complex bending is calculated. Complex bending occurs when…

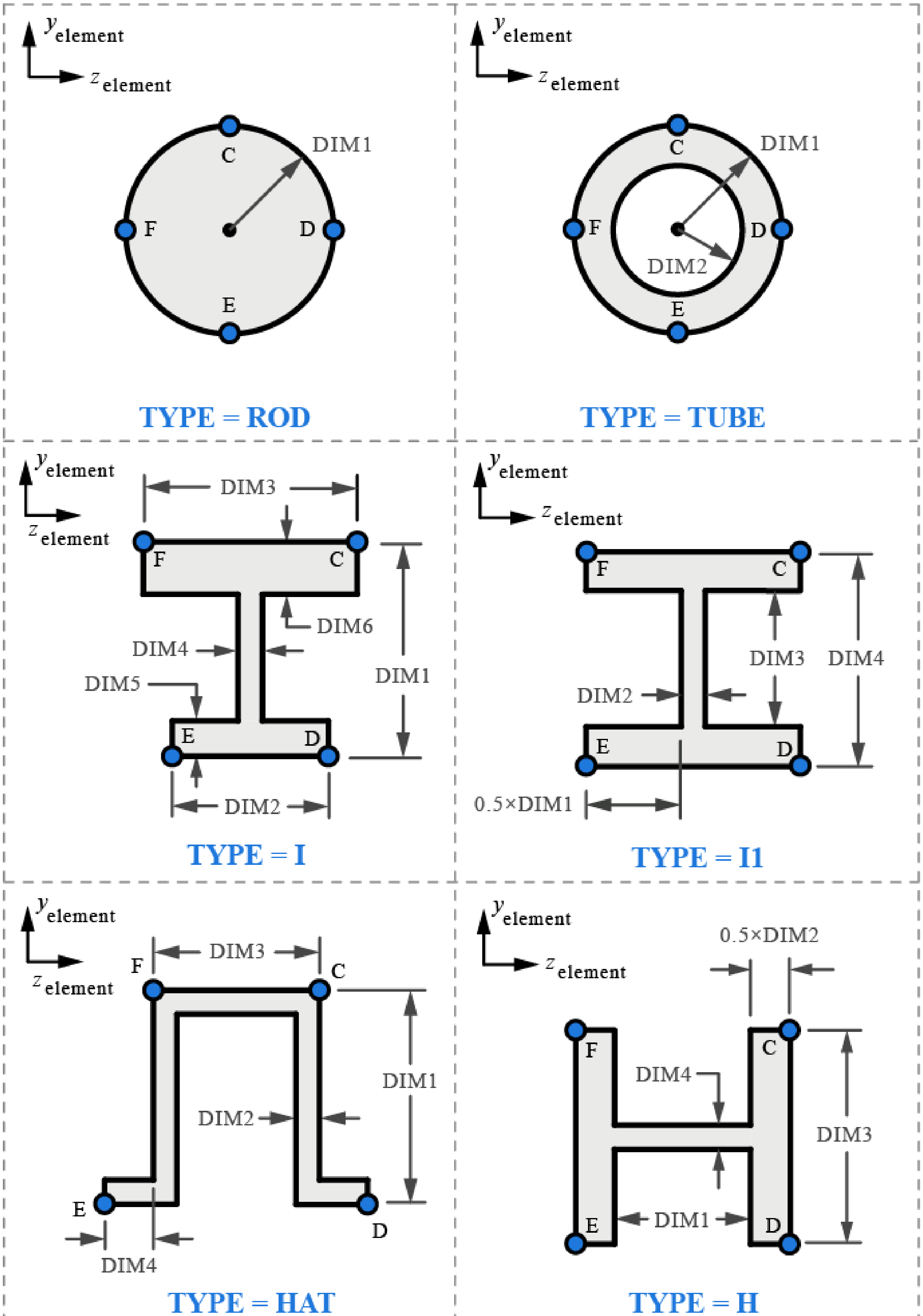
6. The load/moment is assumed to be applied at the shear center for bending and the centroid for axial loads. No twisting occurs due to axial or bending loads.

7. The element coordinate system is shown to be in the upper left corner to define the basic directions for bending. Placing a coordinate system at the neutral axis, centroid, or shear center would imply an incorrect position because of the limitations noted in Remark 6.

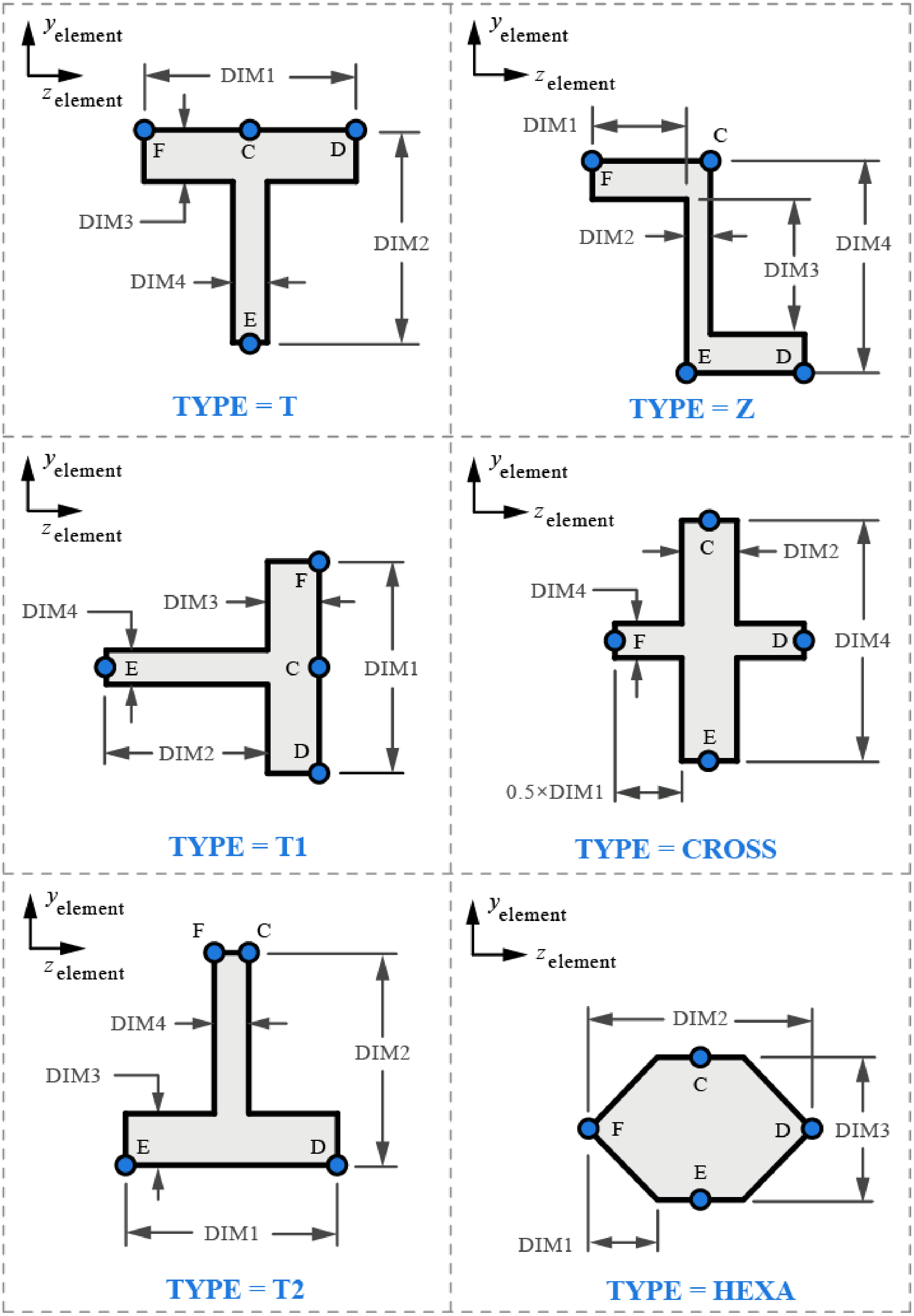
8. If a dimension is not explicitly defined, it may be assumed based on the geometry proportions of the figure.



1 OF 3 - PBARL Cross-Sections



2 OF 3 - PBARL Cross-Sections



3 OF 3 - PBARL Cross-Sections

PBUSH

7.55 PBUSH

Description:

Property definition for a spring element defined by a CBUSH entry.

Format:

1	2	3	4	5	6	7	8	9	10
PBUSH	PID	“K”	K1	K2	K3	K4	K5	K6	
		“RCV”	SA	ST	EA	ET			

Example:

PBUSH	136	K	10000.	20000.	30000.	4000.	50000.	60000.	
		RCV	30.	40.	.01	.02			

Data Description:

Field	Contents	Type	Default
PID	Property ID number	Integer > 0	None
“K”	Indicates that the next 6 fields are stiffness values	Char	None
Ki	Stiffness values	Real	0.
“RCV”	Indicates that the next 4 values are stress/strain recovery coefficients	Real	0.
SA	Stress recovery coefficient in the 3 translational directions		
ST	Stress recovery coefficient in the 3 rotational directions		
EA	Strain recovery coefficient in the 3 translational directions		
ET	Strain recovery coefficient in the 3 rotational directions		

Remarks:

1. Element stresses and strains are calculated by multiplying element engineering forces times the RCV coefficients

7.56 PCOMP

Description:

Property definition for a composite 2D plate/shell element made up of one or more plies.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	MID1	T1	THETA1	SOUT1	MID2	T2	THETA2	SOUT2	
	MID3	(etc)							

Example:

PCOMP	136	-1.02	.0003	30000	TSAI	21.	.002	SYM	
	91	.02	30.						

Data Description:

Field	Contents	Type	Default
PID	Property ID number	Integer > 0	None
Z0	Distance from reference plane to bottom surface of the element	Real	Remark 2
NSM	Non structural mass	Real	0.
SB	Allowable interlaminar shear stress	Real	0.
FT	Failure theory	Char	None
TREF	Reference temperature	Real	0.
GE	Structural damping coefficient	Real	0.
LAM	Symmetric lamination option	Char	NONSYM

MIDi	Ply material ID (MID1 must be specified)	Integer	Last one
Ti	Ply thickness (T1 must be specified)	Real	Last one
THETAi	Material angle of ply relative to element material axis	Real	0.
SOUTi	Not currently used in MYSTRAN		

Remarks:

1. PID must be unique among all PCOMP/PSHELL property entries
2. The default for Z0 is 0.5 times the laminate thickness
3. The failure index for the interlaminar shear is the maximum transverse shear stress divided by SB
4. The allowable failure theories are FT = HILL, HOFF, TSAI or STRN
5. If LAM = SYM only plies on one side of the laminate are to be specified. If an odd number of plies are desired with LAM = SYM then the center ply should have a thickness equal to one-half the actual thickness.
6. The default for MIDi is the previous defined MID. The same holds true for Ti.
7. In order for a ply to be defined, at least one of the 4 ply fields on continuation entries must be present.

7.57 PCOMP1

Description:

Property definition for a composite 2D plate/shell element made up of one or more plies where all plies are the same thickness and same material.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMP1	PID	Z0	NSM	SB	FT	MID	T	LAM	
	THETA1	THETA2	THETA3	etc					

Example:

PCOMP1	136	-1.02	.0003	30000	TSAI	21.	.002	SYM	
	91	.02	30.						

Data Description:

Field	Contents	Type	Default
PID	Property ID number	Integer > 0	None
Z0	Distance from reference plane to bottom surface of the element	Real	Remark 2
NSM	Non-structural mass	Real	0.
SB	Allowable interlaminar shear stress	Real	0.
FT	Failure theory	Char	None
MID	Material ID for all plies	Integer > 0	None
T	Thickness for all plies	Real	0.
LAM	Symmetric lamination option	Char	NONSYM

THETAi	Material angle of ply relative to element material axis	Real	0.

Remarks:

1. PID must be unique among all PCOMP/PSHELL property entries
2. The default for Z0 is 0.5 times the laminate thickness
3. The failure index for the interlaminar shear is the maximum transverse shear stress divided by SB
4. The allowable failure theories are FT = HILL, HOFF, TSAI or STRN
5. If LAM = SYM only plies on one side of the laminate are to be specified. If an odd number of plies are desired with LAM = SYM then the center ply should have a thickness equal to one-half the actual thickness.

7.58 PELAS

Description:

Stiffness definition for CELAS spring elements.

Format:

1	2	3	4	5	6	7	8	9	10
PELAS	PID	K	GE	S					

Example:

PELAS	63	1.55E6		.015					

Data Description:

Field	Contents	Type	Default
PID	Property ID number	Integer > 0	None
K	Spring stiffness	Real	0.
GE	Damping coefficient	Real	0.
S	Stress recovery coefficient	Real	0.

Remarks:

1. PID must be unique among all PELAS property entries.
2. Stress is output for this element as S times the elongation of the spring.

7.59 PLOAD2

Description:

Uniform pressure load for 2D bending plate elements.

Format No. 1:

1	2	3	4	5	6	7	8	9	10
PLOAD2	SID	P	EID1	EID2	EID3	EID4	EID5	EID6	

Format No. 2:

1	2	3	4	5	6	7	8	9	10
PLOAD2	SID	P	EID1	THRU	EID2				

Examples:

PLOAD2	267	.05	12	23	56	124	9789		

PLOAD2	345	.167	269	THRU	9823				

Data Description:

Field	Contents	Type	Default
SID	Load set ID number	Integer > 0	None
P	Pressure value	Real	0.
EIDi	ID numbers of elements that are to have this pressure as a load	Integer > 0	None

Remarks:

1. A positive value of P will result in a pressure being applied in the positive direction of the local z axis for the element (perpendicular to the elements' average midplane)

2. If the THRU option is used EID2 must be greater than EID1. All elements whose ID's are in the range EID1 through EID2 will have the pressure load (if SID selected in Case Control directly or via the load combining LOAD Bulk Data entry).

3. In order for this load to be used in a static analysis the load set ID must either be selected in Case Control by LOAD = SID, or this load set ID must be referenced on a LOAD Bulk Data entry which itself is selected in Case Control.

4. Up to six elements can have their pressure specified on one PLOAD2 entry in Format No 1. For more elements, use additional PLOAD2 entries (i.e. there is no continuation entry for PLOAD2).

7.60 PLOAD4

Description:

Pressure load on the face of 2D bending plate elements, CTRIA3, CTRIA3K, CQUAD4, CQUAD4K.

Format No. 1:

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P1	P2	P3	P4			

Format No. 2:

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID1	P1	P2	P3	P4	THRU	EID2	

Examples:

PLOAD4	267	987	1.1	1.5	1.25	1.4			

PLOAD4	345	101	2.4	2.25	2.1	2.0	THRU	200	

Data Description:

Field	Contents	Type	Default
SID	Load set ID number	Integer > 0	None
Pi	Pressure value at up to 4 grid locations	Real	0.
EIDi	ID numbers of elements that are to have this pressure as a load	Integer > 0	None

Remarks:

1. A positive value of P will result in a pressure being applied in the positive direction of the local z axis for the element (perpendicular to the elements' average midplane)

2. If the THRU option is used EID2 must be greater than EID1. All elements whose ID's are in the range EID1 through EID2 will have the pressure load (if SID selected in Case Control directly or via the load combining LOAD Bulk Data entry).

3. In order for this load to be used in a static analysis the load set ID must either be selected in Case Control by LOAD = SID, or this load set ID must be referenced on a LOAD Bulk Data entry which itself is selected in Case Control.

   If the fields for P2, P3 and/or P4 are blank that pressure is set equal to P1. P4 has no meaning for triangular elements.

7.61 PLOTEL

Description:

1-dimensional dummy element that only serves the purpose of plotting a line. It has no elastic properties.

Format:

1	2	3	4	5	6	7	8	9	10
PLOTEL	EID	G1	G2						

Example:

PLOTEL	63	1001	2365	.					

Data Description:

Field	Contents	Type	Default
EID	Element ID number	Integer > 0	None
Gi	Grid point ID's	Integer > 0	None

Remarks:

1. EID must be unique among all element ID's
2. This element does not result in any stiffness or mass. It's purpose is only to plot a line between 2 grids

7.62 PROD

Description:

Property definition for ROD element.

Format:

1	2	3	4	5	6	7	8	9	10
PROD	PID	MID	A	J	C	MPL			

Example:

PROD	49	2	.175	.093	1.5	0.0175			

Data Description:

Field	Contents	Type	Default
PID	Property ID number	Integer > 0	None
MID	Material ID number	Integer > 0	None
A	Bar cross-sectional area	Real	0.
J	Torsional constant	Real	0.
C	Torsional stress recovery coefficient	Real	0.
MPL	Mass per unit length	Real	0.

Remarks:

1. PID must be unique among all PROD property entries
2. The torsional stress is calculated as:

$$\tau = C\frac{M_t}{J}$$

where M_t is the torsional moment in the rod element.

7.63 PSHEAR

Description:

Property definition for SHEAR element.

Format:

1	2	3	4	5	6	7	8	9	10
PSHEAR	PID	MID	T	NSM					

Example:

PSHEAR	49	2	.175	.093					

Data Description:

Field	Contents	Type	Default
PID	Property ID number	Integer > 0	None
MID	Material ID number	Integer > 0	None
T	Shear panel thickness	Real > 0.	None
NSM	Non-structural mass per unit area	Real	0.

Remarks:

1. PID must be unique among all PSHEAR property entries

7.64 PSHELL

Description:

Property definition for 2D plate elements.

Format:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	TM	MID2	12I/TM**3	MID3	TS/TM	MPA	
	Z1	Z2							

Examples:

PSHELL	987	234	0.10	123	125.	45	20.	.005	
	0.5	-0.5							

PSHELL	78	234	0.10	234		45			

Data Description:

Field	Contents	Type	Default
PID	Property ID number	Integer > 0	None
MID1	Material ID number for membrane material properties	Integer > 0 or blank	None
TM	Membrane thickness	Real or blank	0.
MID2	Material ID number for bending material properties	Integer > 0 or blank	None
12I/TM**3	Ratio of actual bending moment inertia (I) to bending inertia of a solid plate of thickness TM	Real or blank	1.0

MID3	Material ID number for transverse shear material properties	Integer > 0 or blank	None
TS/TM	Ratio of shear to membrane thickness	Real or blank	Remark 3
MPA	Mass per unit area	Real	0.
Z1, Z2	Distances from the neutral plane of the plate to locations where stress is calculated	Real	Remark 4

Remarks:

1. PID must be unique among all PSHELL property entries

2. Continuation entry is not required. If Z1 and Z2 are not input, then stresses are calculated at +/-TM/2.

3. Default value for TS/TM is 5/6 = 0.83333 unless a PARAM Bulk data entry with parameter name TSTM_DEF is in the data file, in which case the TSTM_DEF value on the PARAM entry is used.

4. The following holds for the cases of MIDi blank:

   If MID1 is blank, no membrane stiffness is calculated

   If MID2 is blank, no bending or transverse shear stiffness is calculated

   If MID3 is blank, no transverse shear flexibility is included (Kirchoff plate theory: plate is assumed infinitely stiff in transverse shear) so that normals to the mid-plane remain normal after bending)

7.65 PSOLID

Description:

Property definition for 3D solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
PSOLID	PID	MID	CID	IN		ISOP			

Examples:

PSOLID	987	234	23	3		FULL			

Data Description:

Field	Contents	Type	Default
PID	Property ID number	Integer > 0	None
MID1	Material ID number for membrane material properties	Integer > 0 or blank	None
CID	Material coordinate system ID	Integer or blank	0.
IN	Indicator for integration order (see table below)	Integer = 2,3	2
ISOP	Integration scheme (whether to use FULL or REDUCED integration	Character	REDUCE D

Remarks:

1. See table below for values of IN and ISOP to use

PSOLID entries IN and ISOP for solid elements – only use ones that have comment: OK

(based on test runs by the author)

(bold, underline indicates default which can also be blank)

HEXA	Integration	IN	ISOP	Comments
8 node	**2x2x2 reduced shear**	**2**	**REDUCED**	OK
	2x2x2 standard isopar.	2	FULL or 1	(1)
	3x3x3 reduced shear	3	REDUCED	(1)
	3x3x3 standard isopar	3	FULL or 1	(1)
20 node	2x2x2 reduced shear	2	REDUCED	(2)
	2x2x2 standard isopar.	2	FULL or 1	OK
	3x3x3 reduced shear	**3**	**REDUCED**	OK
	3x3x3 standard isopar	3	FULL or 1	OK

PENTA	Integration	IN	ISOP	Comments
6 node	**2x3 reduced shear**	**2**	**REDUCED**	OK
	2x3 standard isopar.	2	FULL or 1	(1)
	3x7 reduced Shear	3	REDUCED	(1)
	3x7 standard isopar	3	FULL or 1	(1)
15 node	2x3 reduced shear	2	REDUCED	(2)
	2x3 standard isopar.	2	FULL or 1	OK
	3x7 reduced shear	**3**	**REDUCED**	OK
	3x7 standard isopar	3	FULL or 1	OK

TETRA	Integration	IN	ISOP	Comments
4 node	**1 point standard**	**2**	**FULL**	(1)
	4 point standard isopar	3	FULL	(1)

10 node	1 point standard isopar **4 point standard**	 **3**	FULL **FULL**	(2) OK

Notes: (1) Answers degrade for aspect ratio (AR) above AR =1

(2) Answers are nonsense

OK means answers are good

Reduced integration is used for shear strains to avoid shear locking. For HEXA 2x2x2 and PENTA 2x3 integration it uses selective substitution. For HEXA 3x3x3 reduced integration it uses 2x2x2 for shear. For PENTA 3x7 reduced integration it uses 2x3 for shear

7.66 PUSERIN

Description:

Property definition for CUSERIN elements.

Format:

1	2	3	4	5	6	7	8	9	10
PUSERIN	PID	IN4_ID	KNAME	MNAME	RBNAME	PNAME			

Examples:

PUSERIN	101	95	KRRGN	MRRGN					

Data Description:

Field	Contents	Type	Default
PID	Property ID number	Integer > 0	None
IN4_ID	ID of an Exec Control IN4 entry that specifies the NASTRAN formatted INPUTT4 file containing the stiffness and mass matrices (whose name are KNAME, MNAME)	Integer > 0 or blank	None
KNAME	Name of the stiffness matrix which was written to the INPUTT4 file when it was created. This can be up to 8 characters long	Char	None
MNAME	Name of the mass matrix which was written to the INPUTT4 file when it was created. This can be up to 8 characters long	Char	None
RBNAM E	Name of a 6x6 rigid body mass matrix which specifies the rigid body mass relative to the C.G. of the CUSERIN element in its basic coordinate system. This can be up to 8 characters long	Char	None
PNAME	Name of the load matrix which was written to the INPUTT4 file when it was created. This can be up to 8 characters long.	Char	None

Remarks:

1. PID must be unique among all PUSERIN property entries
2. IN4_ID is required. In the example above, an Exec Control entri IN4 with ID = 234 is required
3. The matrix whose name is RBNAME is not required. However, the rigid body mass properties (PARAM GRDPNT) for the overall model will be in error unless the element has the same basic coordinate system as the overall model.
4. The matrix whose name is PNAME is only used for statics solutions.

7.67 RBE2

Description:

Rigid element that has specified components at a number of grids dependent on the six degrees of freedom at one other grid.

Format:

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4	GM5	
	GM6	GM7	(etc)						

Example:

RBE2	43	1021	346	1031	1033	1035	1041	1043	
	1045								

Data Description:

Field	Contents	Type	Default
EID	Element ID number	Integer > 0	None
GN	ID number of the grid that will have all 6 components as the 6 independent degrees of freedom for this rigid element	Integer > 0	None
CM	The component numbers of the dependent degrees of freedom at grid points GMi	Integers 1-6	None
GMi	The components CM at grids GMi are the dependent degrees of freedom that will be eliminated due to this rigid element	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID

2. All of the degrees of freedom defined by components CM at each of the grids GMi are made members of the M-set and their displacements will be rigidly dependent on the six degrees of freedom at grid GN.

Dependent degrees of freedom defined by RBE2 elements cannot be defined as members of any other mutually exclusive set (i.e., cannot appear on SPC, SPC1, OMIT, OMIT1, ASET or ASET1 entries, nor can they appear as dependent degrees of freedom on other rigid elements)

7.68 RBE3

Description:

Element used to distribute loads or mass from one grid point (denoted as the dependent grid) to other grids in the model. The element is defined based on the grids/components that it connects. The resulting multi-point constraints (MPC's) generated internally in MYSTRAN, will eliminate the dependent degrees of freedom and will distribute any loads or mass from the dependent grid to the remaining grids defined on the RBE3. Unlike the NASTRAN RBE3, the MYSTRAN RBE3 does not support the "UM" option at the current time.

Format:

1	2	3	4	5	6	7	8	9	10
RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
	G1,3	WT2	C2	G2,1	G2,2	G2,3	G2,4	WT3	
	C3	G3,1	G3,2	etc					

Example:

RBE3	43		9001	123456	1.0	123	1001	1002	
	1003	1004							

Data Description:

Field	Contents	Type	Default
EID	Element ID number	Integer > 0	None
REFGRID	Grid that will be the dependent (or reference) grid	Integer > 0	None
REFC	The component numbers of the dependent degrees of freedom at grid point REFGRID	Integers 1-6	None
WTi	Weighting factors for the grids/components that follow	Real	None

Ci	Displacement components at the following Gi,j that have weighting factor WTi	Integers 1-6	None
Gi,j	Grids that REFGRID depend on	Integer > 0	None

Remarks:

1. No other element in the model may have the same element ID

2. For most applications only the translation displacement components (1,2,3) should be defined for the Ci. If REFGRID and a Gi,j are coincident then rotation components (4,5,6) can be defined for Ci.

3. Dependent degrees of freedom defined by RBE3 elements can not be defined as members of any other mutually exclusive set (i.e., cannot appear on SPC, SPC1, OMIT, OMIT1, ASET or ASET1 entries, nor can they appear as dependent degrees of freedom on other rigid elements)

7.69 RFORCE

Description:

Defines rigid body rotational velocity, and optional rotational acceleration, of the model about some specified grid for the purpose of generating inertia forces on the finite element model.

Format:

1	2	3	4	5	6	7	8	9	10
RFORCE	SID	GID	CID	V	N1	N2	N3		
	A								

Example:

TBD									

Data Description:

Field	Contents	Type	Default
SID	Load set ID number (must be selected in Case Control)	Integer > 0	None
GID	ID of the grid at which this concentrated moment acts	Integer >0	None
CID	ID of the coordinate system in which the Ni are specified	Integer >= 0	0
V	An overall scale factor for the angular velocity in revolutions per unit time	Real	0.
Ni	Components of a vector in the direction of the angular velocity and angular acceleration	Real	0.
A	An overall scale factor for the angular acceleration in revolutions per unit time squared	Real	0.

Remarks:

1. The force at grid i due to the angular velocity and acceleration is:

$$F_i = [M_i][\omega \times (\omega \times (r_i - r_a) + a \times (r_i - r_a)]$$

where

i = grid point

M_i = 6x6 mass matrix at grid i

ω = rigid body angular velocity of the model

a = rigid body angular acceleration of the model

r_i = distance from basic system origin to grid i

r_a = distance from basic system origin to reference grid about which the model rotates

2. The load set ID (SID) is selected by the Case Control entry LOAD:

3. GID = 0 signifies that the rotation vector acts through the basic system origin.

4. CID = 0 indicates that the rotation vector is defined in the basic coordinate system

7.70 RSPLINE

Description:

Interpolation element. A spline fit using the 2 independent end points (GI1, GI2) is applied to the locations of the dependent points (defined by GDi/CDi) to rigidly constrain the GDi/CDi.

Format:

1	2	3	4	5	6	7	8	9	10
RSPLINE	EID		GI1	GD1	CD1	GD2	CD2	GD3	
	CD3	GD4	CD4	etc	GI2				

Example:

RSPLINE	43		1001	2001	123456	2002	123456	2003	
	123456	2004	123456	2005	123456	1002			

Data Description:

Field	Contents	Type	Default
EID	Element ID number	Integer > 0	None
GIi	Grid numbers of the 2 independent end points	Integer > 0	None
GDi	Grid numbers of the dependent grids	Integers > 0	None
CDi	Displacement component numbers at the GDi	Integer 1-6	None

Remarks:

1. No other element in the model may have the same element ID
2. Displacements at the GDi are interpolated using the following rules applied to the line between the 2 end points:

Displacements along the line and rotations about the line are linear

Displacements perpendicular to the line are cubic

Rotations normal to the line are quadratic

7.71 SEQGP

Description:

Manual re-sequencing of grids.

Format:

1	2	3	4	5	6	7	8	9	10
SEQGP	G1	S1	G2	S2	G3	S3	G4	S4	

Example:

SEQGP	1001	1.5	1011	1.	1021	2.	1031	3.5	

Data Description:

Field	Contents	Type	Default
Gi	ID number of a grid point	Integer > 0	None
Si	The sequence number for Gi	Integer or Real > 0	None

Remarks:

1. The SEQGP entry is used to manually re-sequence grids. See the Bulk Data PARAM GRIDSEQ entry for the starting sequence MYSTRAN uses in manual grid sequencing.

2. Either integer or real sequence numbers are allowed but all are converted to real internally. Thus, if the user has two grids sequenced consecutively, say with integer sequence numbers 10 and 11, then some other grid can be inserted in the sequence between the two with a real sequence number anywhere in the range:

$$10. < Si < 11.$$

3. Up to four pairs of Gi, Si can be specified on one SEQGP entry. For more pairs, use additional SEQGP entries (i.e. there is no continuation entry for SEQGP).

4. If automatic grid point sequencing by BANDIT, any used defined SEQGP entries are ignored.

7.72 SLOAD

Description:

Defines the existence of a scalar load on a scalar point.

Format:

1	2	3	4	5	6	7	8	9	10
SLOAD	SID	Si	FMAG						

Example:

SPOINT	56	101	125.6						

Data Description:

Field	Contents	Type	Default
SID	Load set ID number	Integer > 0	None
Si	Scalar point ID	Integer > 0	None
FMAG	Magnitude of the force on scalar point Si	Real	0.

Remarks:

1. In order for this load to be used in a static analysis the load set ID must either be selected in Case Control by LOAD = SID, or this load set ID must be referenced on a LOAD Bulk Data entry which itself is selected in Case Control.

7.73 SPC

Description:

Single point constraints that are defined by specifying the degree of freedom and its displacement (either zero or some enforced nonzero value).

Format:

1	2	3	4	5	6	7	8	9	10
SPC	SID	G1	C1	D1	G2	C2	D2		

Example:

SPC	56	101	3	1.2E-3	201	2	0.0		

Data Description:

Field	Contents	Type	Default
SID	ID number of the single point constraint set	Integer > 0	None
GI	ID numbers of the grids that will have component number Ci constrained	Integer > 0	None
CI	Component numbers at grids Gi that will be constrained	Integers 1-6	None
DI	The value for the displacement at grid Gi, component Ci	Real	0.

Remarks:

1. Single point constraint sets must be selected in Case Control with the entry SPC = SID in order for them to be applied.
2. Degrees of freedom defined on SPC entries will be members of the S-set and cannot be defined as being members of any other mutually exclusive set.
3. Up to two gid/component pairs can be specified as being single point constrained on one SPC entry (i.e. continuation entries are not allowed). Additional SPC entries can have the same SID.

4. If a Gi/Ci pair is constrained more than once (with the same SID), the last value read for Di will be used.

5. A degree of freedom may be specified redundantly as a permanent single point constraint on a GRID Bulk Data entry and on an SPC or SPC1 Bulk Data entry. If it is defined on the GRID entry and on an SPC Bulk Data entry, Di must be zero on the SPC entry or a fatal error will occur.

7.74 SPC1

Description:

Single point constraints that are defined by specifying the degree of freedom to be constrained to zero displacement.

Format No. 1:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	(etc)					

Format No. 2:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	THRU	G2				

Examples:

SPC1	52	135	1001	1002	103	1004	2001	2002	+CONT
	2003	2004							

SPC1	52	135	1001	THRU	1004				
SPC1	52	135	2001	THRU	2004				

Data Description:

Field	Contents	Type	Default
SID	ID number of the single point constraint set	Integer > 0	None
C	Component numbers at grids Gi that will be constrained	Integers 1-6	None

GI	ID numbers of the grids that will have component number Ci constrained	Integer > 0	None
DI	The value for the displacement at grid Gi, component Ci	Real	0.

Remarks:

1. Single point constraint sets must be selected in Case Control with the entry SPC = SID in order for them to be applied.
2. Degrees of freedom defined on SPC entries will be members of the S-set and cannot be defined as being members of any other mutually exclusive set.
3. For format 2, all grids in the model that are in the range G1 through G2 will have component C constrained
4. A degree of freedom may be specified redundantly as a permanent single point constraint on a GRID Bulk Data entry and on an SPC or SPC1 Bulk Data entry.

7.75 SPCADD

Description:

Combine single point constraint sets defined on SPC, SPC1 entries.

Format:

1	2	3	4	5	6	7	8	9	10
SPCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	(etc)						

Example:

SPCADD	283	11	74	123	564				

Data Description:

Field	Contents	Type	Default
SID	Single point constraint set ID number	Integer > 0	None
Si	Set IDs of SPC and/or SPC1 Bulk Data entries	Integer > 0	None

Remarks:

1. Single point constraint sets must be selected in Case Control with the entry SPC = SID in order for them to be applied.
2. All single point constraints specified on the SPC and/or SPC1 entries whose set IDs are the Si on the SPCADD will be applied to the model if SPC = SID is in Case Control.

7.76 SPOINT

Description:

Defines the existence of a scalar point (1 component of displacement) in the model.

Format 1:

1	2	3	4	5	6	7	8	9	10
SPOINT	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8	
	ID9	etc							

Format 2:

1	2	3	4	5	6	7	8	9	10
SPOINT	ID1	THRU	ID2						

Example:

SPOINT	56	101	3	1.2E-3	201	2	0.0		

Data Description:

Field	Contents	Type	Default
IDi	ID of an SPOINT	Integer > 0	None

Remarks:

1. SPOINT ID's must be unique among all other SPOINT's and among all GRID's
2. SPOINT's are like GRID's but have only 1 component of displacement and their outputs are scalar, not vector, quantities. In the F06 output file, however, the output quantities are reported under the T1 headings.

7.77 SUPORT

Description:

Defines degrees of freedom that are to be in the R-set (for Craig-Bampton model generation).

Format:

1	2	3	4	5	6	7	8	9	10
SUPORT	GID	C	GID	C	GID	C	GID	C	

Example:

SUPORT	4981	12	695	123	5647	456			

Data Description:

Field	Contents	Type	Default
GID	ID of a grid whose components in the next field will be put into the R-set	Integer > 0	None
C	Displacement component numbers (digits 1 through 6)	Integer > 0	None

Remarks:

1. This Bulk Data entry is meant for use in Craig-Bampton analyses. The degrees of freedom specified on this entry will be treated the same as Single Point Constraints (SPC's) in all other analyses

7.78 TEMP

Description:

Grid point temperature definition for purposes of calculating thermal loads on the model.

Format:

1	2	3	4	5	6	7	8	9	10
TEMP	SID	G1	T1	G2	T2	G3	T3		

Example:

TEMP	4	1011	25.	1012	32.	1013	28.		

Data Description:

Field	Contents	Type	Default
SID	ID number of the temperature set	Integer > 0	None
GI	ID numbers of the grids whose temperature is being defined	Integer > 0	None
Ti	Temperature of grid Gi	Real	0.

Remarks:

1. Temperature sets must be selected in Case Control with the entry TEMP = SID in order for them to be used in calculating thermal loads

2. Every element in the model must have its temperature defined for set SID, either explicitly through an element temperature entry on TEMPRB, TEMPP1 Bulk Data entry or implicitly using grid temperatures on TEMP, TEMPD Bulk Data entries. Element temperatures defined on element TEMPRB, TEMPP1 entries take precedence over any that might be defined using grid temperatures. If no element temperature is explicitly defined, the element temperature is taken to be the average of the temperatures of the grids to which the element is connected.

3. Thermal loads for the model are calculated using element temperatures defined via TEMP, TEMPD, TEMPRB, TEMPP1 Bulk data entries, the element properties and the material

properties (including coefficient of thermal expansion and reference temperature). The thermal loads calculated are based on element temperatures that are the difference between those defined on TEMP, TEMPD, TEMPRB, TEMPP1 and the reference temperature defined on the material entry for the element.

4. Only three grids may have their temperature defined for set SID in one TEMP entry. Additional grid temperatures can be specified using more TEMP Bulk Data entries with the same SID.

7.79 TEMPD

Description:

Default grid point temperature definition for purposes of calculating thermal loads on the model.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPD	SID1	T1	SID2	T2	SID3	T3	SID4	T4	

Example:

TEMPD	4	46.2	33	52.1					

Data Description:

Field	Contents	Type	Default
SIDi	ID number of a temperature set	Integer > 0	None
Ti	The default temperature for grids for set SIDi	Real	0.

Remarks:

1. Temperature sets must be selected in Case Control with the entry TEMP = SID in order for them to be used in calculating thermal loads

2. All grids whose temperature is not defined on a TEMP Bulk Data entry will have the default temperature T, if there is one defined on a TEMPD for set SID.

3. Every element in the model must have its temperature defined for set SID, either explicitly through an element temperature entry on TEMPRB, TEMPP1 Bulk Data entry or implicitly using grid temperatures on TEMP, TEMPD Bulk Data entries. Element temperatures defined on element TEMPRB, TEMPP1 entries take precedence over any that might be defined using grid temperatures. If no element temperature is explicitly defined, the element temperature is taken to be the average of the temperatures of the grids to which the element is connected.

4. Thermal loads for the model are calculated using element temperatures defined via TEMP, TEMPD, TEMPRB, TEMPP1 Bulk data entries, the element properties and the material properties (including coefficient of thermal expansion and reference temperature). The thermal loads calculated are based on element temperatures that are the difference between those defined on TEMP, TEMPD, TEMPRB, TEMPP1 and the reference temperature defined on the material entry for the element.

5. Only four pairs of SIDi/Ti may be defined on one TEMPD entry. Additional pairs can be specified using more TEMPD Bulk Data entries.

7.80 TEMPP1

Description:

Defines temperatures and temperature gradients for 2D plate elements.

Format No. 1:

1	2	3	4	5	6	7	8	9	10
TEMPP1	SID	EID1	TBAR	TPRIME					
	EID2	EID3	EID4	EID5	(etc)				

Format No. 2:

1	2	3	4	5	6	7	8	9	10
TEMPP1	SID	EID1	TBAR	TPRIME					
	EID2	THRU	EID3	EID4	THRU	EID5			

Examples:

TEMPP1	13	2101	35.7	10.1					
	2679	3201	1104	32	5555				

TEMPP1	13	2101	35.7	10.1					
	2304	THRU	6789	12	THRU	46			

Data Description:

Field	Contents	Type	Default
SID	ID number of the temperature set	Integer > 0	None

EIDi	Element ID numbers	Integer > 0	None
TBAR	Average temperature of the element	Real	0.
TPRIME	Linear thermal gradient through the thickness of the element	Real	0.

Remarks:

1. Any number of continuation entries can be used
2. For format number 2, the THRU ranges must have the second element ID greater than the first.
3. Temperature sets must be selected in Case Control with the entry TEMP = SID in order for them to be used in calculating thermal loads.
4. Every element in the model must have its temperature defined for set SID, either explicitly through an element temperature entry on TEMPRB, TEMPP1 Bulk Data entry or implicitly using grid temperatures on TEMP, TEMPD Bulk Data entries. Element temperatures defined on element TEMPRB, TEMPP1 entries take precedence over any that might be defined using grid temperatures. If no element temperature is explicitly defined, the element temperature is taken to be the average of the temperatures of the grids to which the element is connected.
5. Thermal loads for the model are calculated using element temperatures defined via TEMP, TEMPD, TEMPRB, TEMPP1 Bulk data entries, the element properties and the material properties (including coefficient of thermal expansion and reference temperature). The thermal loads calculated are based on element temperatures that are the difference between those defined on TEMP, TEMPD, TEMPRB, TEMPP1 and the reference temperature defined on the material entry for the element.

7.81 TEMPRB

Description:

Defines temperatures and temperature gradients for 1D bar elements.

Format No. 1:

1	2	3	4	5	6	7	8	9	10
TEMPRB	SID	EID1	TA	TB	TP1A	TP1B	TP2A	TP2B	
	EID2	EID3	EID4	EID5	(etc)				

Format No. 2:

1	2	3	4	5	6	7	8	9	10
TEMPRB	SID	EID1	TA	TB	TP1A	TP1B	TP2A	TP2B	
	EID2	THRU	EID3	EID4	THRU	EID5			

Examples:

TEMPRB	13	2101	35.7	10.1					
	67	89	2	13	1	789			

TEMPRB	13	2101	35.7	10.1					
	68	THRU	97	2101	THRU	4009			

Data Description:

Field	Contents	Type	Default
SID	ID number of the temperature set	Integer > 0	None
EIDi	Element ID numbers	Integer > 0	None

TA	Average temperature of the element at end a	Real > 0.	0.
TB	Average temperature of the element at end b	Real > 0.	0.
TP1A	Linear temperature gradient in element y axis at end a	Real	0.
TP1B	Linear temperature gradient in element y axis at end b	Real	0.
TP2A	Linear temperature gradient in element z axis at end a	Real	0.
TP2B	Linear temperature gradient in element z axis at end b	Real	0.

Remarks:

1. Any number of continuation entries can be used

2. For format number 2, the THRU ranges must have the second element ID greater than the first

3. Temperature sets must be selected in Case Control with the entry TEMP = SID in order for them to be used in calculating thermal loads

4. Every element in the model must have its temperature defined for set SID, either explicitly through an element temperature entry on TEMPRB, TEMPP1 Bulk Data entry or implicitly using grid temperatures on TEMP, TEMPD Bulk Data entries. Element temperatures defined on element TEMPRB, TEMPP1 entries take precedence over any that might be defined using grid temperatures. If no element temperature is explicitly defined, the element temperature is taken to be the average of the temperatures of the grids to which the element is connected.

5. Thermal loads for the model are calculated using element temperatures defined via TEMP, TEMPD, TEMPRB, TEMPP1 Bulk data entries, the element properties and the material properties (including coefficient of thermal expansion and reference temperature). The thermal loads calculated are based on element temperatures that are the difference between those defined on TEMP, TEMPD, TEMPRB, TEMPP1 and the reference temperature defined on the material entry for the element.

$$TA=\frac{1}{A}\int_A T_a(y,z)dA$$

$$TB=\frac{1}{A}\int_A T_b(y,z)dA$$

6. The average temperatures TA and TB at ends a and b respectively are:

   where A is the cross-sectional area and $T_a(y,z)$ and $T_b(y,z)$ are the temperature distributions at ends a and b respectively.

7. The linear gradients through the thickness, TP1A, TP1B, TP2A and TP2B, are:

$$TP1A = \frac{1}{I1}\int_A T_a(y,z)\,y\,dA$$

$$TP1B = \frac{1}{I1}\int_A T_b(y,z)\,y\,dA$$

$$TP2A = \frac{1}{I2}\int_A T_a(y,z)\,z\,dA$$

$$TP2B = \frac{1}{I2}\int_A T_b(y,z)\,z\,dA$$

where I1 and I2 are the bending moments of inertia for the bar (on the PBAR entry) and $T_a(y,z)$ and $T_b(y,z)$ are the temperature distributions at ends a and b respectively.

7.82 USET

Description:

Defines a set of degrees of freedom that belong to a user defined set (named either “U1” or “U2”). The purpose is for the user to get an output listing that defines the internal degree of freedom order for the members of the set.

Format:

1	2	3	4	5	6	7	8	9	10
USET	NAME	G1	C1	G2	C2	G3	C3		

Example:

USET	U1	101	3	201	2				

Data Description:

Field	Contents	Type	Default
NAME	A user defined set. The name must be either “U1” or “U2”	Char	None
GI	ID numbers of the grids that the user wants to be members of the set	Integer > 0	None
CI	Component numbers at grid Gi that will be members of the set	Integers 1-6	None

Remarks:

1. The Gi, Ci are defined as members of the displacement set named SNAME.
2. A row oriented tabular output showing the internal sort order of the members of the set (named SNAME) can be output if a PARAM, USETSTR, Ui Bulk Data entry is present (I = 1 or 2).
3. In order to get a listing of the internal sort order, a Bulk Data PARAM, USETSTR, Ui (i=1 or 2) must be included

7.83 USET1

Description:

Defines a set of degrees of freedom that belong to a user defined set (named either “U1” or “U2”). The purpose is for the user to get an output listing that defines the internal degree of freedom order for the members of the set.

Format No. 1:

1	2	3	4	5	6	7	8	9	10
USET1	SNAME	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	(etc)					

Format No. 2:

1	2	3	4	5	6	7	8	9	10
USET1	SNAME	C	G1	THRU	G2				

Examples:

USET1	U2	135	1001	1002	103	1004	2001	2002	
	2003	2004							

USET1	U2	135	1001	THRU	1004				

Data Description:

Field	Contents	Type	Default
SNAME	A user defined set. The name must be either “U1” or “U2”	Char	None
GI	ID numbers of the grids that are members of the user defined set	Integers 1-6	None

C	Component numbers at grids Gi that are part of the user defined set	Integer > 0	None

Remarks:

1. The Gi, C are defined as members of the displacement set named SNAME.
2. A row oriented tabular output showing the internal sort order of the members of the set (named SNAME) can be output if a PARAM, USETSTR, Ui Bulk Data entry is present (I = 1 or 2).
3. In order to get a listing of the internal sort order, a Bulk Data PARAM, USETSTR, Ui (i=1 or 2) must be included

8 REFERENCES

1. LAPACK Users' Guide, 3rd edition, SIAM, 1999
   (see website at http://www.netlib.org/lapack).

2. ARPACK Users' Guide, 3rd edition, SIAM, 1998
   (see website at http://www.caam.rice.edu/software/ARPACK/).

3. Everstine, G. C., "Recent improvements to Bandit", NASTRAN: Users' Experiences, Volume NASA TM X-3278 pages 511-521, Washington, DC, 1975. National Aeronautics and Space Administration.

4. Tessler, A. and Hughes, T.J.R., "A three-node Mindlin plate element with improved transverse shear", Computer Methods In Applied Mechanics And Engineering 50 (1985) 71-101.

5. Tessler, A. and Hughes, T.J.R., "An improved treatment of transverse shear in the Mindlin-type four-node quadrilateral element", Computer Methods In Applied Mechanics And Engineering 39 (1983) 311-335.

6. Batoz, J., "An explicit formulation for an efficient triangular plate-bending element", International Journal For Numerical Methods In Engineering, Vol. 18 (1982), 1077-1089.

7. Batoz, J. and Tahar, M.B., "Evaluation of a new quadrilateral thin plate", International Journal For Numerical Methods In Engineering, Vol. 18 (1982), 1655-1677.

8. Case, William R., "A NASTRAN DMAP procedure for calculation of base excitation modal participation factors", 11th NASTRAN User's Colloquium, May 5-6, 1983.

9. Liu, J, Riggs, H.R. and Tessler, A., "A four-node, shear-deformable shell element developed via explicit Kirchoff constraints", International Journal For Numerical Methods In Engineering, Vol. X, 2000, 49, pp 1065-1086.

10. MacNeal, Richard H., "Finite Elements. Their Design and Performance", Marcel Dekker, 1993.

11. Case, William R., DMAP for generating Craig-Bampton Models, notes from a course given at the Goddard Space Flight Center (contact author for copy of paper).

12. MYSTRAN-Demo-Problem-Manual (contained in the MYSTRAN setup file downloaded from www.MYSTRAN.com along with this manual.

13. Li, X.S. et al. “SuperLU Users Guide”, Sept 1999. (https://portal.nersc.gov/project/sparse/superlu/)

14. Ashton, J.E., “Approximate Solutions for Unsymmetrically Laminated Plates”, General Dynamics Corporation, Fort Worth, Texas, October 28, 1968.

15. Yeongbin Ko, Phill-Seung Lee, Klaus-Jürgen Bathe, “A New MITC4+ Shell Element”, Computers and Structures, 182, 2017.

16. Bath, Klaus-Jurgen, “A formulation of general shell elements—the use of mixed interpolation of tensorial components”, March 1986.