Users Reference Manual

for the

MYSTRAN General Purpose Finite Element

Structural Analysis Computer Program

(Open Source Version)

By Dr Bill Case
www.mystran.com

(September 2022)
Most consistent with MYSTRAN version 14.0

Table of Contents

List of Figures

List of Tables

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 it's 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 insure 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.

  <u>Elastic Elements (1, 2 and 3D):</u>

  - 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 elements:

    - QUAD4 quadrilateral plate elements with plate membrane and bending stiffness, as well as transverse shear flexibility, based on Mindlin thick plate theory (References 5 and 9). These are essentially flat elements, however small distortion out of plane is accommodated.

    - TRIA3 flat triangular plate element with plate membrane and bending stiffness, as well as transverse shear flexibility, based on Mindlin thick plate theory (Reference 4)

    - QUAD4K quadrilateral plate element with plate 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.

    - TRIA3K flat triangular plate element with plate membrane and bending stiffness based on Kirchoff thin plate theory (Reference 6)

    - 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).

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 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.

---

[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.

3 The 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:

$$u_{1_g}, u_{2_g}, u_{3_g}, \theta_{1_g}, \theta_{2_g}, \theta_{3_g} \qquad \text{3-1}$$

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:

201   301   401   101   501   601   701

7
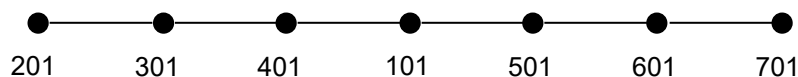
Assuming that the user has the initial order set with PARAM GRIDSEQ GRID then grid 101 would be sequenced 1st initially. However, for a minimum stiffness matrix bandwidth, it should be sequenced so that it is 4th. Using the SEQGP entry, grid 101 can be re-sequenced to be 4th by giving it a sequence number 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

8

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 elements

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 constant 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).

There are 2 versions of the QUAD4 quadrilateral plate element, referred to as MIN4 and MIN4T in MYSTRAN. The MIN4 version is described in Reference 5. Version 2.06 of MYSTRAN introduced the MIN4T version of the QUAD4 element described in Reference 9 to correct the deficiency in the MIN4 QUAD4 that could develop stresses in rigid body motion for elements that were not rectangular. The default QUAD4 is the MIN4T version. However, both versions are in MYSTRAN and 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 MIN4T uses the quad element in Reference 9. The MIN4T QUAD4 element is made up of 4 non-overlapping TRIA3 elements

- 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

- 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 $\pi^2/12$ (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, or lamina, of composite elements (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.

Figure 3-5 shows the triangular and quadrilateral element coordinate systems. Figure 3-6 shows the convention for plate force resultants which are the basis for calculating element stresses. These are standard definitions of plate force resultants that can be found in texts on the theory of plates and shells.

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 $x_e - y_e$ plane for the element (as shown in Figure 3-5) is the mean plane between the grids. Instead of allowing significant warp of quadrilateral elements, triangular elements should be used.

Output for the plate elements includes:

- Element engineering forces:

    - Membrane force resultants (force/length) as shown on Figure 3-6

    - Bending moment resultants (moment/length) as shown on Figure 3-6

    - Transverse shear force resultants (force/length) for the QUAD4 and TRIA3 as shown on Figure 3-6

- 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_e direction

    - Normal stress in the y_e 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 stresses can be output at the element center as well as at the corner nodes of the element. The TRIA3 element has constant stress so only one output per element is provided.

3.2.2.6   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:

- Displacenents 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 it's 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:

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

3-2

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:

$$U_G = \begin{Bmatrix} U_N \\ U_M \end{Bmatrix}$$ 3-3

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:

$$U_N = \begin{Bmatrix} U_F \\ U_S \end{Bmatrix}$$ 3-4

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:

$$U_F = \begin{Bmatrix} U_A \\ U_O \end{Bmatrix}$$ 3-5

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:

$$U_A = \begin{Bmatrix} U_L \\ U_R \end{Bmatrix}$$ 3-6

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

22

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 CaseControl)

- 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 3 solution types: SOL = 1 for statics, SOL = 3 for eigenvaluse and SOL = 31 for Craig-Bampton (CB) model generation. The first two of these are very similar to the static and eigenvalue solution types in NASTRAN and will not be elaborated upon. The third, CB model generation is a new analysis type and is discussed in more detail

4.1   Statics

SOL 1 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   Eigenvalues

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   Buckling and Differential Stiffness[2]

SOL 5 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 thebuckling load

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   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

---

[2] The BAR element is coded for buckling (SOL 5) or differential stiffness (SOL 4). The solid elements have also been coded for buckling and differential stiffness.

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

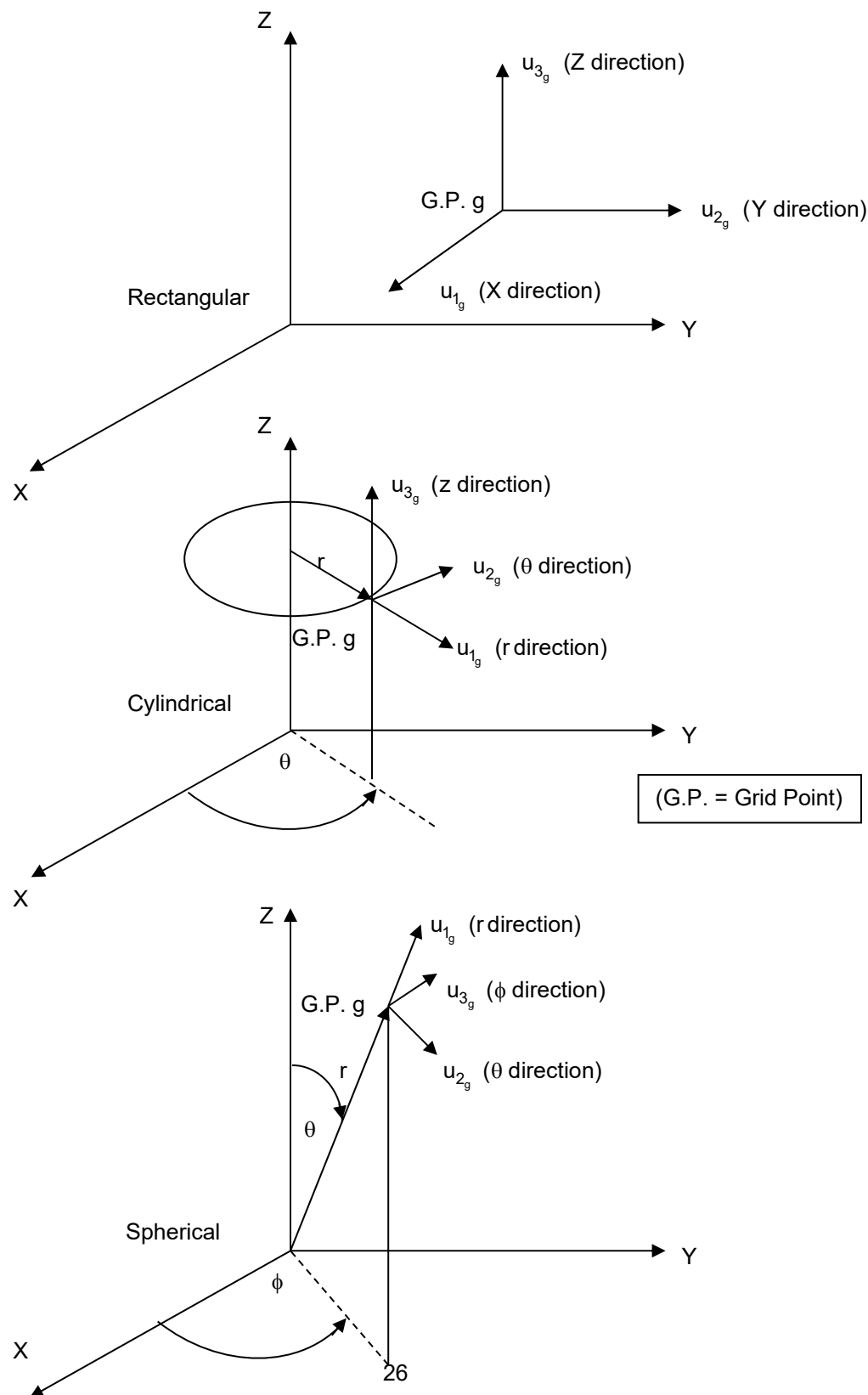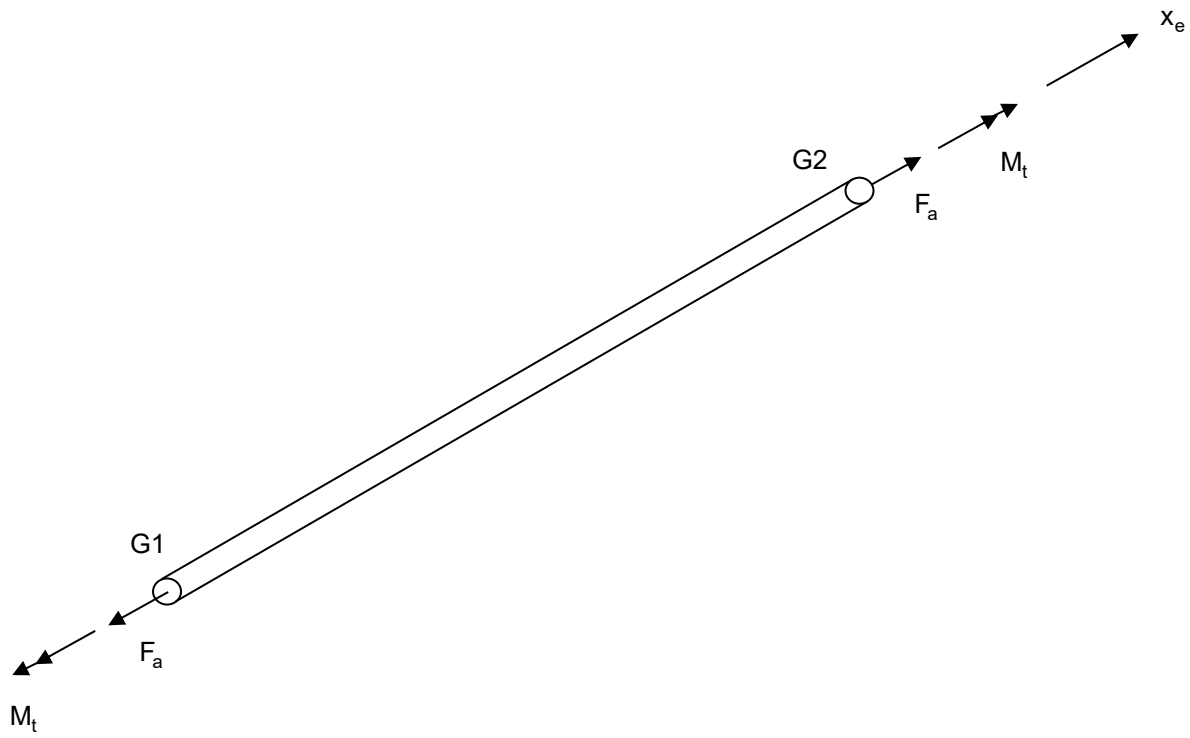**Figure 4-1: Rectangular, Cylindrical and Spherical Coordinate Systems**

Rectangular

u_{3_g} (Z direction)

G.P. g

u_{2_g} (Y direction)

u_{1_g} (X direction)

Z

Y

X

Cylindrical

u_{3_g} (z direction)

u_{2_g} (θ direction)

u_{1_g} (r direction)

r

G.P. g

Z

Y

θ

X

(G.P. = Grid Point)

Spherical

u_{1_g} (r direction)

u_{3_g} (ϕ direction)

u_{2_g} (θ direction)

G.P. g

r

θ

Z

Y

ϕ

X

26

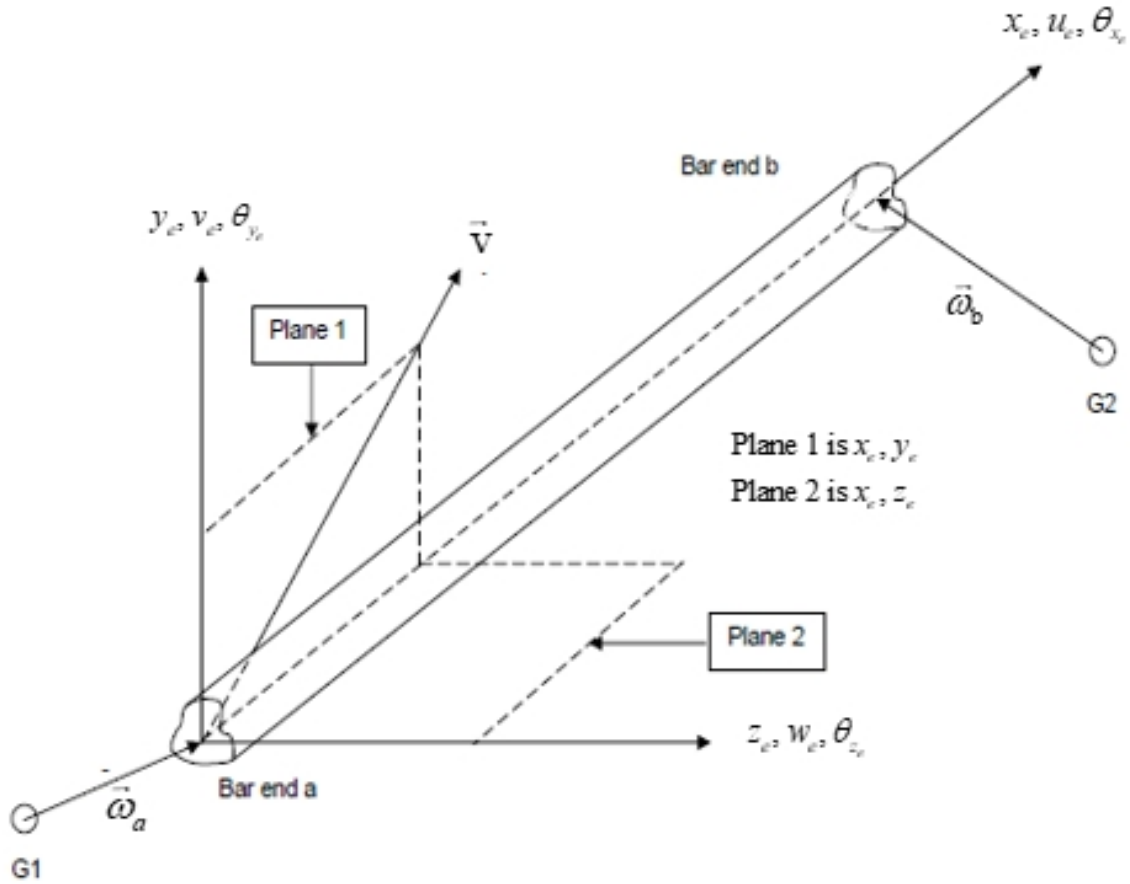**Figure 4-2: Rod Element Geometry, Coordinate System and Forces**

F_a = Axial Load

M_t = Torque

x_e = Rod axis (positive from grid G1 through grid G2)

Figure 4-3: Bar Element Geometry and Coordinate System

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

$\overset{r}{v}$ = Vector specified on CBAR or BAROR entry used in defining Plane 1
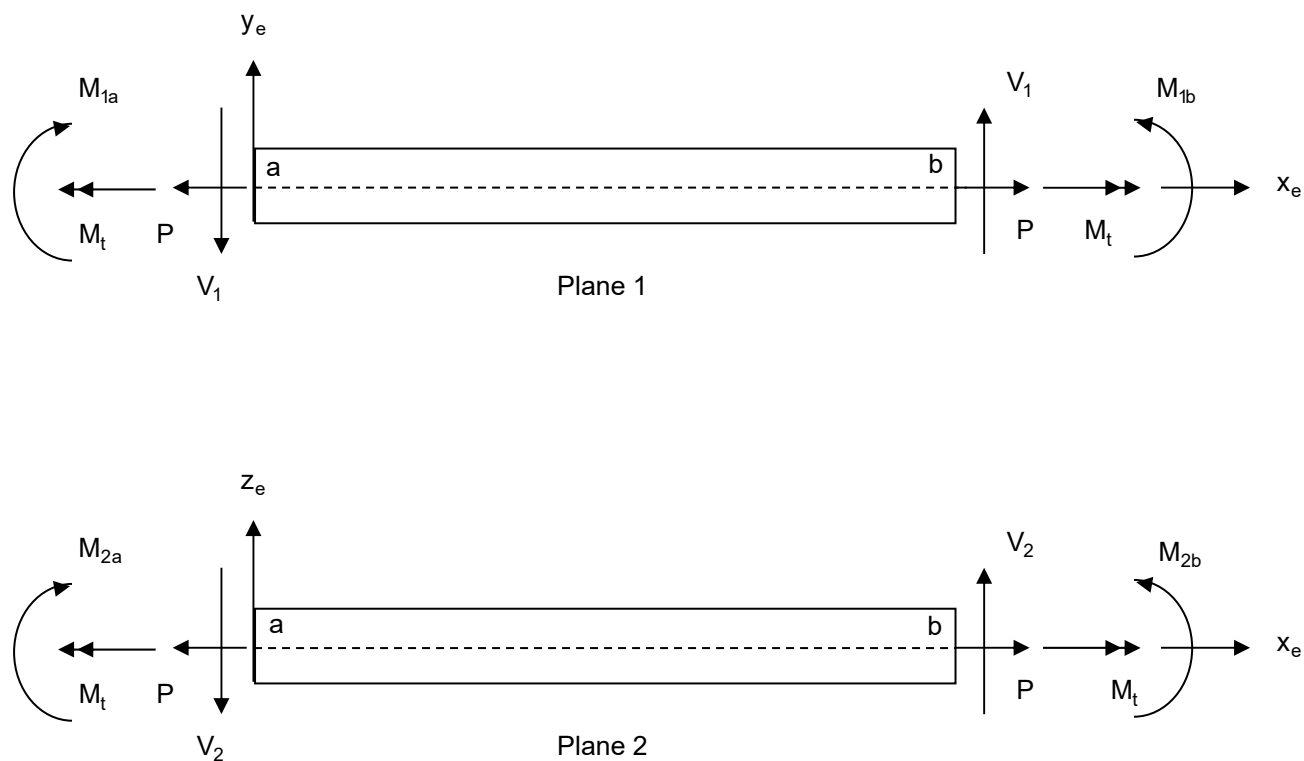
z_e = Axis in the plane defined by x_e and the vector cross product $x_e \otimes \overset{r}{v}$

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

$\overset{r}{\omega_a}$ = Vector from grid G1 on the CBAR entry to end a of the Bar (the offset at end a)

$\overset{r}{\omega_b}$ = Vector from grid G2 on the CBAR entry to end b of the Bar (the offset at end b)

Figure 4-4: Bar Element Forces

Plane 1

Plane 2

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 4-5: Plate Element Geometry and Coordinate Systems

y_e

G3

Triangular Plate Element

G1

G2

x_e

y_e

Gi is a grid point

G3

G4

α

α

x_e

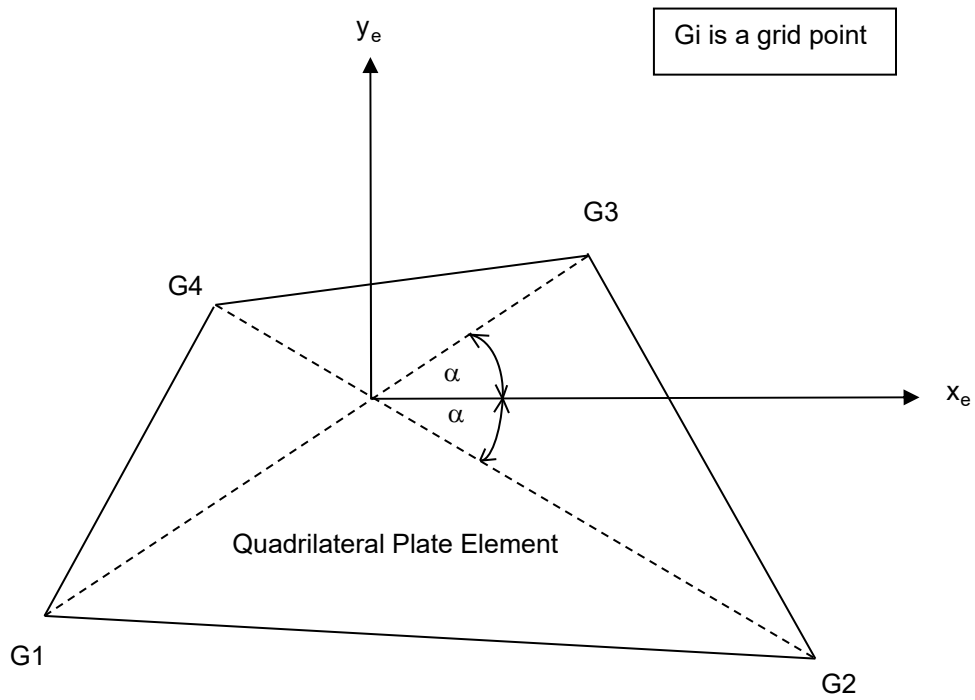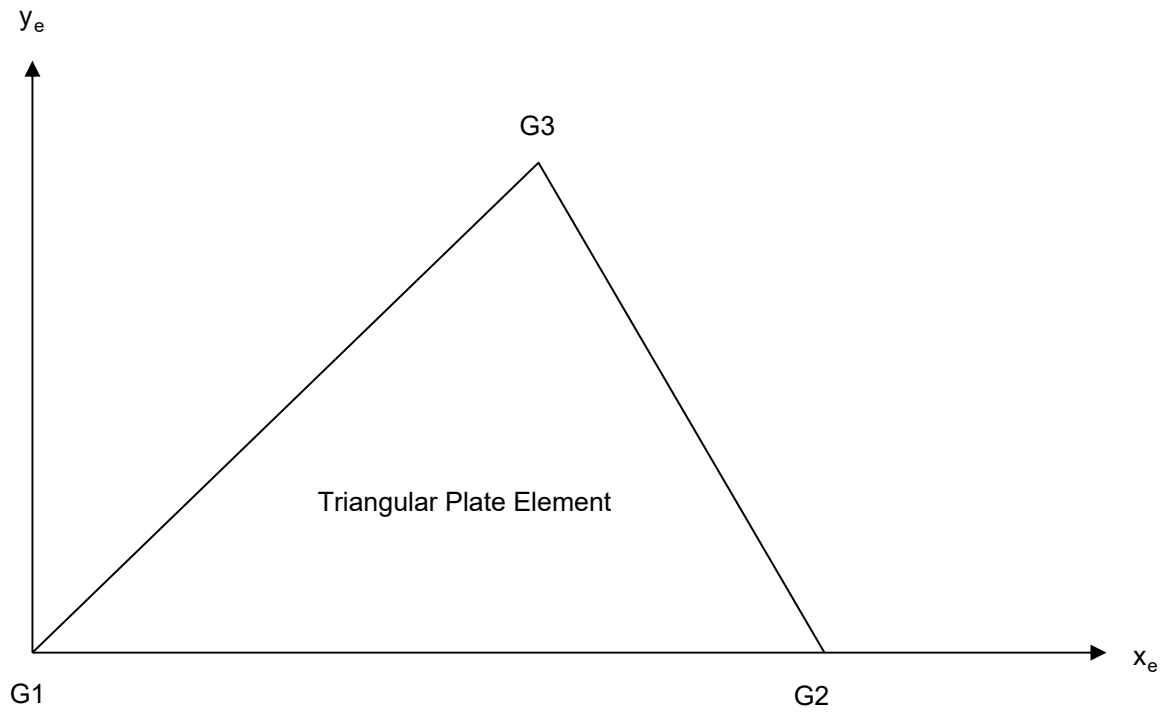Quadrilateral Plate Element

G1

G2

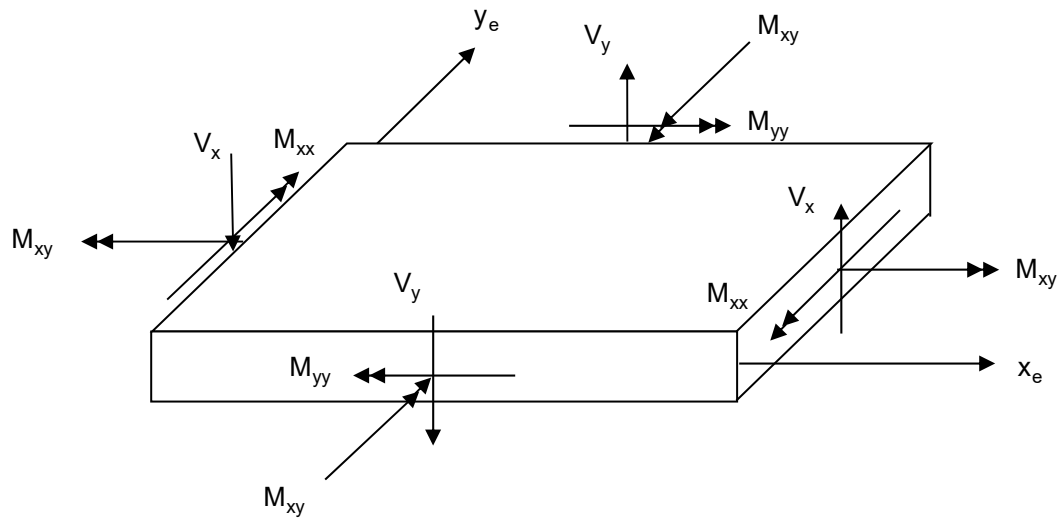**Figure 4-6: Plate Element Force Resultants**

Plate Bending Moment and Transverse Shear Force Resultants
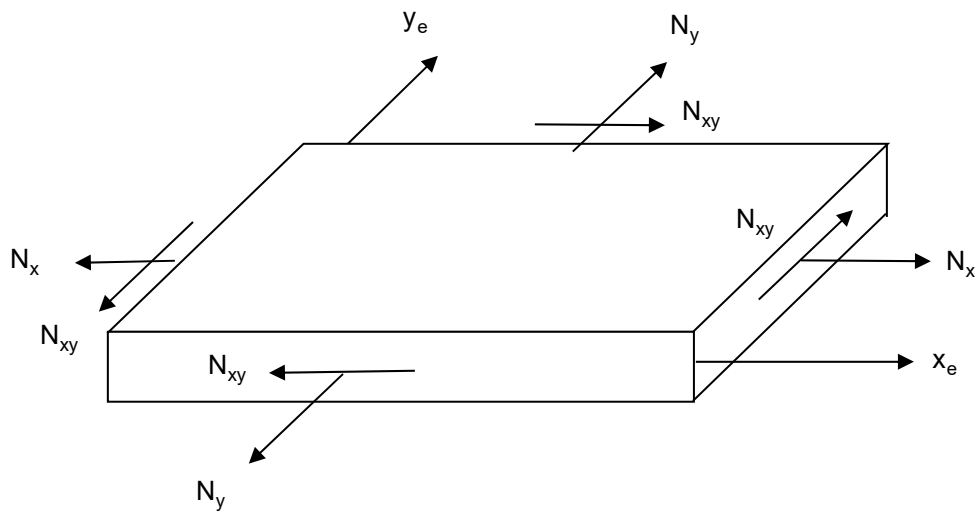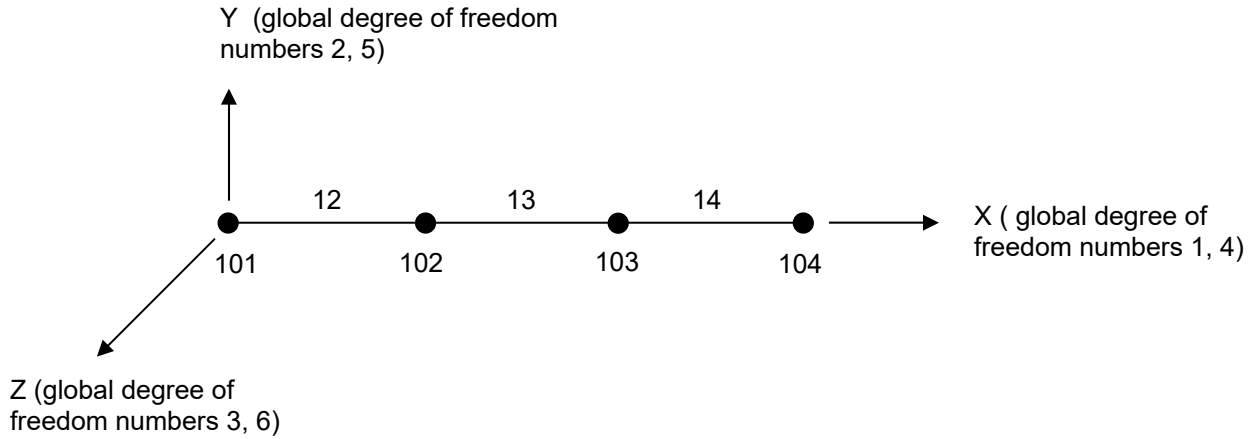

Plate Membrane Force Resultants

Figure 4-7: 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, $\quad \theta_{x_i}$ = rotation of grid i about X axis

v_i = displ of grid i in the Y direction, $\quad \theta_{y_i}$ = rotation of grid i about Y axis

w_i = displ of grid i in the Z direction, $\quad \theta_{z_i}$ = 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 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. 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 (see https://portal.nersc.gov/project/sparse/superlu/)