GENESIS

ANALYSIS MANUAL

VERSION 17.0

May 2018

1767 SOUTH 8TH STREET, SUITE 200
COLORADO SPRINGS, CO 80905
Phone: (719) 473-4611 Fax: (719) 473-4638
http://www.vrand.com
email: genesis.support@vrand.com

TABLE OF CONTENTS

GENESIS

Analysis Manual

CHAPTER 7 Output Files

CHAPTER A Diagnostic Information

CHAPTER B VR&D Client Support

CHAPTER C Notices

1

CHAPTER 1

Introduction

1

1.1 Analysis Capabilities

GENESIS can solve analysis problems in static, vibration, dynamic and random linear elasticity where the structure is modeled as an assemblage of rod, beam, bending/membrane, shear, composite, scalar and solid elements. Multiple loading and multiple boundary conditions are considered. Responses that are calculated include internal forces, stresses, strains, joint displacements, velocities, accelerations, grid point stresses, reaction forces, system strain energies, mass, volume, system moments of inertia and vibration frequencies. Single and multipoint constraints are allowed.

Inertia relief is available for static analysis, and it can be used simultaneously with different support conditions.

Nonlinear static contact analysis can be performed using the CGAP element to model the potential contact points and/or BCPAIR data to identify potential surface-to-surface contact.

Nonlinear effects due to preloads on bolts can be considered using the BOLTSUB solution control command and BOLD elements.

Guyan reduction is available for vibration analysis and it can be used simultaneously with different boundary (ASET) conditions.

Buckling analysis is also available for checking the stability of the structure subject to statics loads.

GENESIS will also solve linear static heat transfer problems with heat flux, conduction, convection and adiabatic boundary conditions. Volumetric heat generation loads are available. Grid point temperatures and reaction fluxes are calculated. The resulting thermal loads can be automatically applied to a linear static structural analysis.

Two linear equation solvers are available; sparse matrix and skyline. The sparse matrix solver is normally used as the default. It is not available at all installations. In this case, the skyline solver is used. The user can choose the solver to be used by specifying the analysis parameter “SOLVER”.

Three eigenvalue solvers are available; subspace iteration, Lanczos and the SMS solver.

1

1.2 Availability

GENESIS is written to operate on everything from personal computers to supercomputers. This allows users to solve a wide variety of everyday design tasks at their desks, while sending only the largest problems to a mainframe or supercomputer. To make the best use of computer resources, problems of significant size may be solved using workstations at night, when they normally stand idle.

GENESIS is a 64-bit program that allows the use of large amounts of memory.

GENESIS can run in multithreaded mode on shared memory parallel computers.

CHAPTER 2

Finite Element Analysis

2

2

2.1 Grid Points

The structural model is created based on grid points which are the connection points between elements, points where the structure is constrained against movement, points where loads are applied, or points used to define coordinate systems. Displacements, velocities, accelerations and temperatures in the structure are calculated at the grid locations. Also, vibration modes of the structure are calculated in terms of grid movements. Therefore, the grid locations are fundamental to modeling the structure. Also, for continuum structures, the arrangement of the grids will have a profound effect on the efficiency and accuracy of the finite element solution. For design, the location of grid points can be changed by optimization to improve the structure.

Figure 2-1 shows how grids are used to define some very simple planar structures. In general, the intersections of lines on the figures represent grid points. It will be necessary to assign a unique identification number to each grid point. The dark rectangles are used to represent supports for the structures.

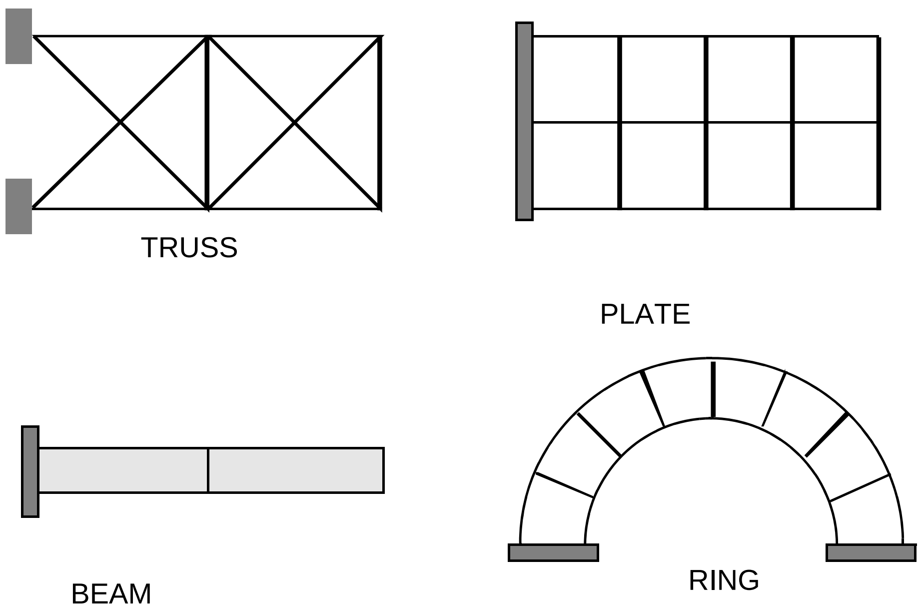


Figure 2-1 Simple Model

The definition of grids in the structural model forms the basis of all other analysis (and therefore, design) data. This is shown in **Figure 2-2**, where the arrows identify the direction in which the information references.

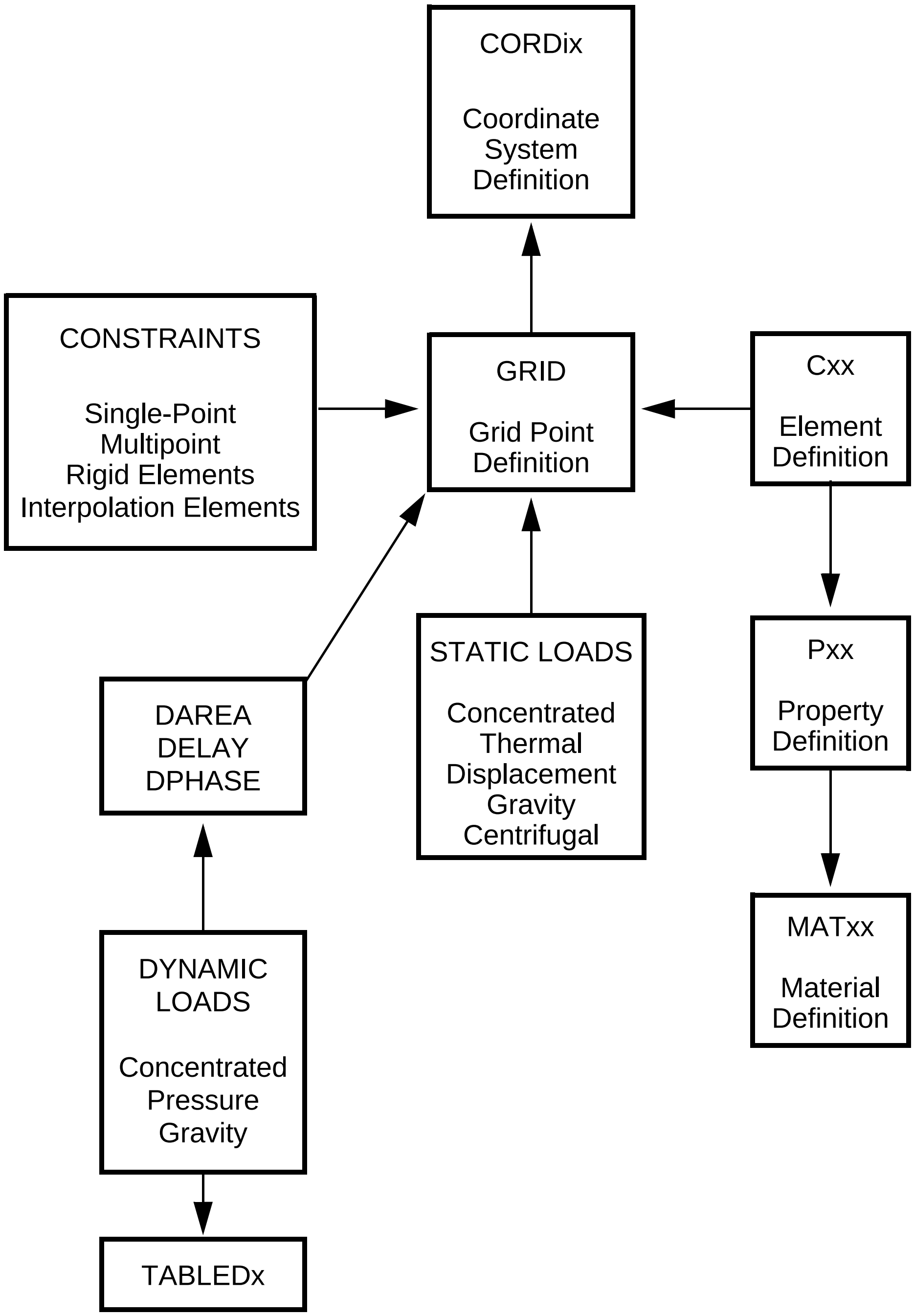


Figure 2-2 The Structural Analysis Model

Note: The grid point definition is central to the analysis model. Constraint definitions over-ride any constraints defined on individual grids. That is, any additional constraints, not defined on the grid point definition, are automatically imposed by the constraint definitions. The only case where grid point definitions reference other information is for coordinate systems. The grid point definition statements point to the coordinate system definition if it is different from the basic coordinate system. Defaults for grid point constraints and coordinate systems can be set. Finally, all elements reference the grid point definitions. The Cxxx information defines the connectivity of the elements to the grids. Additionally, the element definition points to the properties for the individual elements which, in turn, point to the material information.

2.2 Coordinate Systems

All grid points must reference a coordinate system. There are three different types of coordinate systems that are related to the grid points. Also, two additional coordinate systems, being the element coordinate system and the material coordinate system, are used for stiffness, mass, stress, strain and internal force calculations.

- **BASIC COORDINATE SYSTEM** is a fixed cartesian coordinate system to which all others are referenced. The basic coordinate system represents a fixed point and orientation in space, to which any other coordinate systems are related, either directly or indirectly.
- **LOCAL COORDINATE SYSTEM** is a unique coordinate system that can be used to define grid locations and displacements. A local coordinate system may be defined relative to another local coordinate system. However, all local coordinate systems must be referenced to the basic coordinate system, either directly, or by referencing other local coordinate systems in a chain that eventually references the basic coordinate system.

  All grid point locations must be specified in the basic or a local coordinate system (the input system). The grid point displacements must also be defined in the basic or local coordinate system (the output system). The input and output systems for a grid point may be different. The default input and output coordinate systems are the basic coordinate system.
- **GENERAL COORDINATE SYSTEM** is the collection of all the grid point output coordinate systems. Grid point constraints and displacements, velocities and accelerations are measured in the general coordinate system, not in the basic coordinate system (unless no local coordinate systems are specified, in which case the basic and general coordinate systems are the same). What this means is that, if you define the displacements of some grid points relative to a local spherical coordinate system, the displacements will be calculated in that coordinate system. Note that the general coordinate system for each grid is a cartesian system. For example, displacements in the θ direction in a spherical system have units of length, not degrees. The displacement direction is the θ direction at the grid point.
- **ELEMENT COORDINATE SYSTEM** is the coordinate system attached to an individual element which is used to define its material or structural axes. Also, stiffness and mass properties are calculated in this system and then transformed to the general coordinate system. Finally, stresses, strains and forces are calculated in the element coordinate system, or the material coordinate system in the case of solid elements, or in the layer coordinate system in the case of composite element stresses and strains.

- **MATERIAL COORDINATE SYSTEM** defines the orientation of non-isotropic properties relative to the element coordinate system. Stresses and strains in solid and axisymmetric elements are calculated in this coordinate system. Stresses, strains and forces of plate/shell elements can be calculated in the material coordinate system.
- **LAYER COORDINATE SYSTEM** defines the orientation of non-isotropic properties relative to the material coordinate system. Stresses and strains in composite elements are calculated in this coordinate system.

2

2.2.1 Local Coordinate System Options

Three types of local coordinate systems are available. These are rectangular (cartesian), cylindrical and spherical systems. These three coordinate systems are shown in **Figure 2-3**.

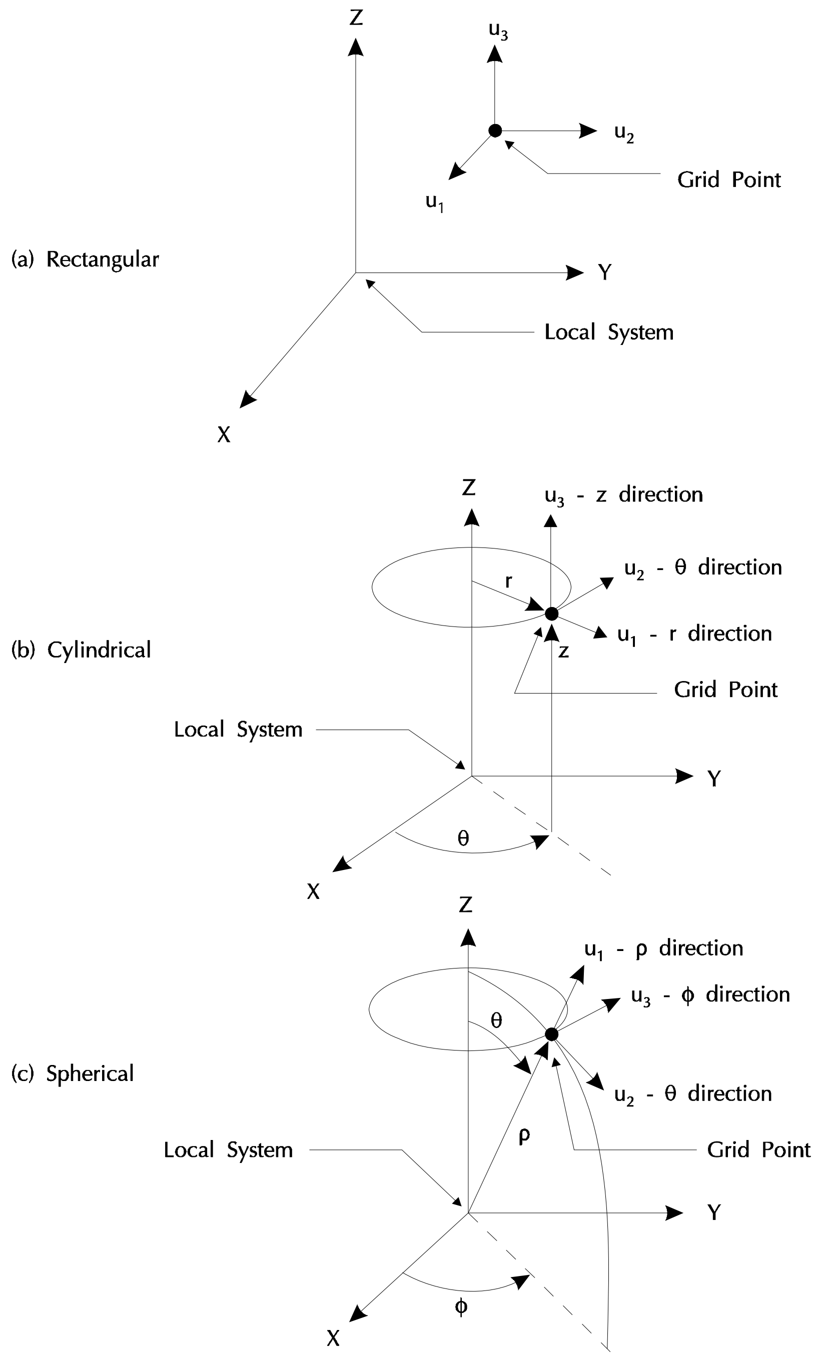


Figure 2-3 Displacement Coordinate Systems

More than one coordinate system of each type may be defined. This allows you to define grid points in separate parts of the structure in a convenient local system. It is only necessary that each local coordinate reference the basic system, either directly, or indirectly through it's reference to another local system. This is shown in **Figure 2-4**, where the dotted line between coordinate systems shows how each references another, back to the basic system.

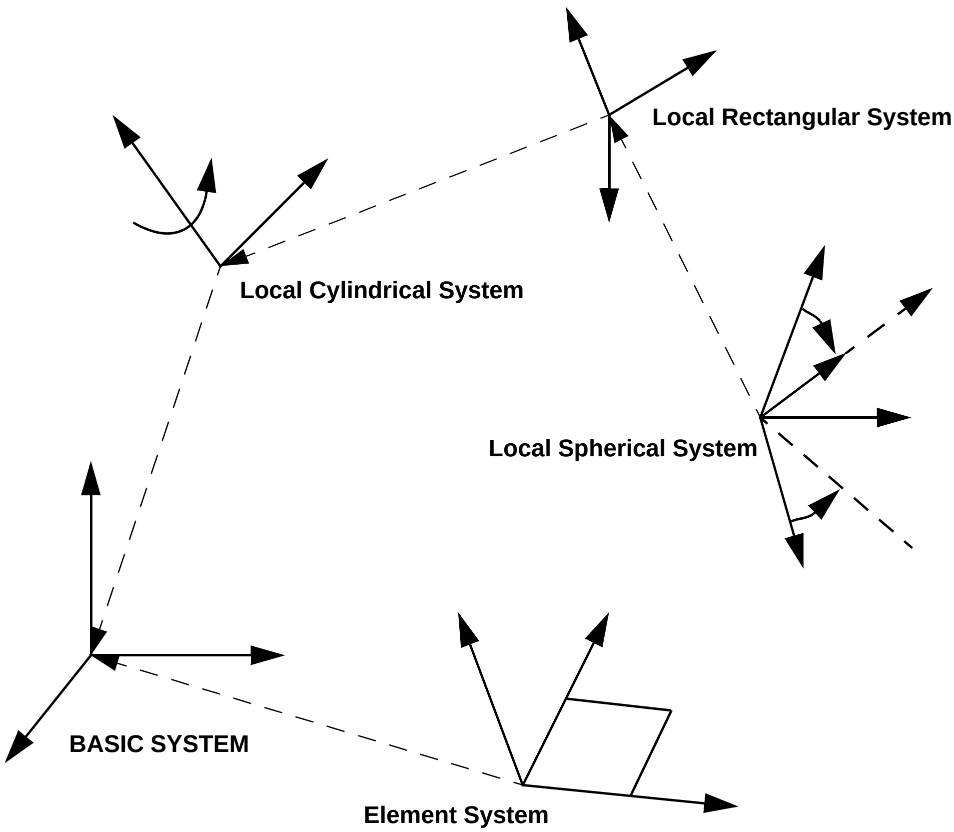


Figure 2-4 Multiple Coordinate Systems

Local coordinate systems may be defined by referencing three grid points or by providing three physical sets of coordinates. In each case, this will define the position of the local coordinate system origin, relative to the basic coordinate system or to another local coordinate system, and will define the orientation of this local system.

2.3 Boundary Conditions

Five types of boundary conditions may be imposed on the structure. These include rigid constraints at grid points (single point constraints), prescribed displacements at grid points (imposed displacements), and constraints that require two or more grid points to move in space according to a fixed relationship (multi-point constraints). In static analysis, for inertia relief load cases, support (reference) degrees of freedom can be specified. In frequency analysis, for Guyan reduction, free degrees of freedoms can be selected. In heat transfer analysis, prescribed temperatures and constraints that require two or more grid point temperatures having a fixed relationship can be used.

2.3.1 Single Point Constraints

Single point constraints are defined as restraints (fixed against movement) at a particular grid. If such constraints are defined on **GRID** definition data, then they apply to this grid under all loading conditions. In this case, the grid is constrained in the direction(s) specified for the output coordinate system that the GRID data references. Also, **GRDSET** data may be used to define default constraints for all grids under all loading conditions. Additional single point constraints may be activated in the Solution Control data, in which case they apply only to the specific load cases identified there. **SPC1** or **SPC** data is used to define such constraints. Heat transfer analysis single point constraints set the grid point temperature to zero. The user can combine SPC/SPC1 data sets using the bulk data command **SPCADD**.

2.3.2 Prescribed Displacements

In a similar way, enforced non-zero displacements may be applied to grid points. These are activated in the Solution Control data and apply only to the specified loading cases. **SPC** or **SPCD** data is used to define prescribed displacements. In heat transfer analysis, prescribed grid point temperatures are defined with SPC or SPCD data. Prescribed displacements are not available for dynamic response.

2.3.3 Multi Point Constraints

Multi-point constraints require that a linear combination of displacements at several grids must add to zero. This is accomplished by making the first degree of freedom a dependent variable when solving the displacement equations. Multi-point constraints are activated in the Solution Control data and apply only to the specified loading cases. **MPC** data is used to define multi-point constraints. A linear combination of grid point temperatures can be required to add to zero by using multipoint constraints in heat transfer analysis.

2.3.4 Degrees of Freedom for Reduction

Free degrees of freedom for calculating frequencies and mode shapes using the Guyan reduction technique or for specifying boundary degrees of freedom for superelement reduction can be selected. Guyan reduction is activated in the Solution Control data and applies only to the specified eigenvalue load cases. **ASET2** and/or **ASET3** data is used to specify the free degrees of freedom.

2.3.5 Free Body Support for Inertia Relief

Inertia relief analysis can be activated using the solution control command **SUPORT**.

Two options are available: Automatic or manual. The automatic option is obtained by using SUPORT=AUTO. When using AUTO the structure must consist of exactly one connected component that has exactly 6 rigid body modes (i.e., the structure cannot be constrained to move or rotate in any of the 3 coordinate directions).

The maual option requires the user to select a reference set of degrees of freedom just sufficient to restrain all rigid body modes (typically 6). This is accomplished by using the **SUPORT1** data.

Inertia relief can also be activated using PARAM, INREL, -2. This will set the default for all static loadcases to SUPORT=AUTO. Individual static loadcases can override this default with an explicit SUPORT entry.

2.3.6 Contact Boundary Conditions

Nonlinear surface-to-surface contact in static analysis can be activated using the solution control command **BCONTACT**. **BCPAIR** data is used to define a contact interface set, with each contact interface selecting a pair of surfaces to put into potential contact. The user can combine BCPAIR data sets using the bulk data command **BCPADD**. Surfaces are defined by **BSURFE**, **BSURFM** and/or **BSURFP** data entries. The **NLPARM** solution control command is required along with BCONTACT to control the nonlinear solution process.

Nonlinear contact boundary condition can also be simulated using the **CGAP** element.

The gap element only behaves nonlinearly in static loadcases that have nonlinearity activated using the **NLPARM** solution control command. In all other cases, the gap element is a simple linear spring.

2.3.7 Glue Connections

Glue connections enforce compatible displacements across surface-to-surface boundaries or between a point collection and a surface for all structural loadcase types. The glue connection represents a permanent physical bonding of the interface. This is most useful to connect components modeled with non-conforming meshes.

Surface-to-surface glue adds a linear penalty stiffness to connect the displacements of the degrees of freedom on one surface to the degrees of freedom of the opposite surface. The **CGLUE** bulk data is used to identify pairs of surfaces that are to be glue-connected. Surfaces are defined by **BSURFE**, **BSURFM** and/or **BSURFP** data entries.

Point collection-to-surface glue uses internally generated **RBE3** elements to connect the degrees of freedom of grid points to the degrees of freedom of the opposite surface. The **CGLUE1** bulk data is used to identify a point collection and a surface that are to be glue-connected. Point collections are defined by **BPOINTG** data entries.

Note that CGLUE1 connections are ignored in heat transfer analysis.

2.4 Elastic Elements

Here, we briefly define the elastic elements available in *GENESIS*. Each element must be assigned a unique number for reference. This is required even for elements of different types. That is no two elements in the structure may be assigned the same identification number.

The elements are defined here in a simple outline form for clarity. Additional details are provided with the actual input data describing the elements.

2.4.1 Rod Element (CROD, CTUBE)

The **CROD** element has stiffness only in the axial direction (tension and compression). Either a consistent or a lumped mass formulation can be used to generate the element mass matrix. Geometric (differential) stiffness is calculated for this element for buckling analysis. The element material, area, and nonstructural mass are specified in the **PROD** input data. Only isotropic (**MAT1**) materials may be referenced. Thermal, centrifugal and gravity loads can be applied to CROD elements. Element axial forces and stresses are recovered at each end of the CROD elements.

The figure below defines the sign convention for forces in the CROD elements.

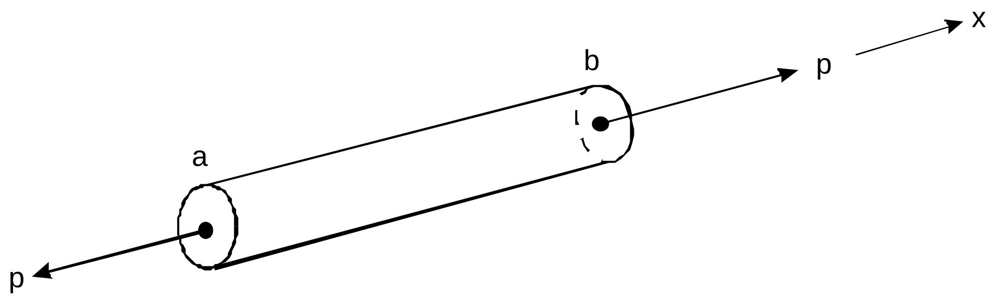


Figure 2-5

The **CTUBE** element is identical to the CROD element, except that it references property data defined on the **PTUBE** entry. Equivalent PROD data is calculated from the dimensions specified on PTUBE.

2.4.2 Bar Element (CBAR)

The **CBAR** element is a general frame element with axial, torsional, and bending stiffness. The element is uniform along its length. Shear deformation can be included in the stiffness formulation. The ends of the CBAR element may be offset from its connecting grid points by vectors defined in the General Coordinate system. The ends of the CBAR element may be pin jointed using the pin flag option. Warping is ignored. A consistent mass formulation which includes rotary inertia and shear deformation effects is used to generate the element mass matrix. Alternatively, a lumped mass formulation may be used.

Geometric (differential) stiffness is calculated for this element for buckling analysis.

The CBAR element properties are defined on the **PBAR** or **PBARL** input data. These include the material, element section properties, nonstructural mass per unit length, and stress recovery locations. Only isotropic (**MAT1**) materials may be referenced.

Thermal, centrifugal and gravity loads can be applied to CBAR elements. Linearly varying distributed tractions and moments along the element can be applied in the basic or element coordinate system (**PLOAD1**) or any local coordinate system (**PLOADA**). Six element forces (2 shear forces, 2 bending moments, axial force, and torque) and four stresses (bending plus axial at user defined locations) are recovered at each end of the CBAR element.

The figure below defines the geometric orientation of bar elements, as well as the sign convention for the forces on the elements.

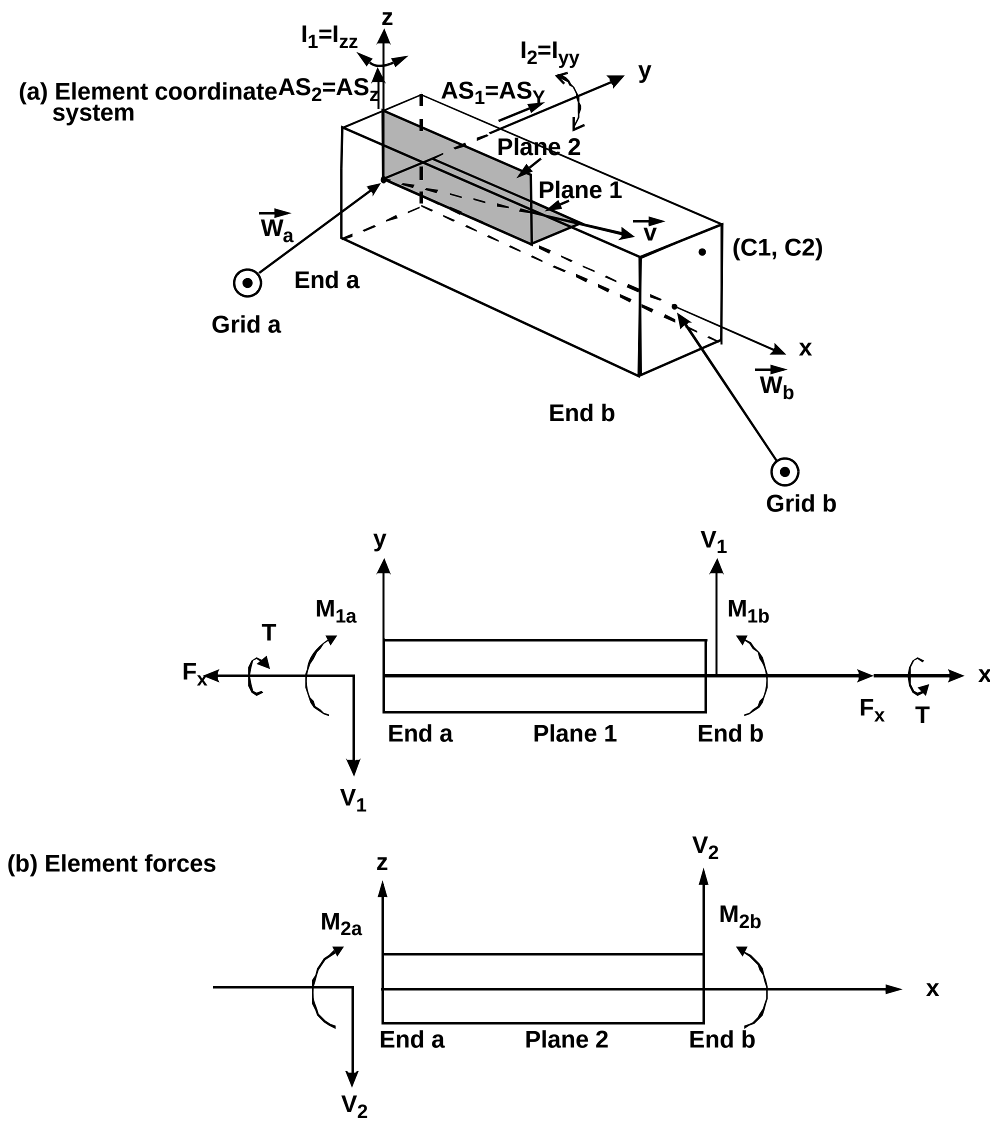


Figure 2-6

2.4.3 General Beam Element (CBEAM)

The **CBEAM** element is a general frame element with axial, torsion and bending stiffness. Shear deformation is included in the basic formulation. Warping is also included. The element can be non-uniform or uniform in its length. Section properties can be specified at up to 11 stations along the length of the element.

The ends of the CBEAM element may be offset from its connecting grid points by vectors defined in the general coordinate system. The ends of the CBEAM may be pin jointed using the pin flag options.

The neutral axis may be offset form the shear axis. This allows analysis of non-symmetric beams.

The mass center of gravity may be offset from the shear center axis.

A consistent mass formulation is used for the uniform beam. This formulation includes rotatory inertia. For a uniform beam, the lumped formulation is also available. For a non-uniform beam the lumped formulation is used to construct the mass matrix. This lumped mass includes the effect of rotatory inertia and allows for a mass center of gravity that is not coincident with the shear or neutral axis.

Geometric (differential) stiffness is calculated for this element for buckling analysis.

The CBEAM element properties are defined either on the **PBEAM** or **PBEAML** input data. These includes the material, element properties, nonstructural mass per unit length, the shear factors, the warping coefficients, the stress recovery locations, the neutral axis and the mass center of gravity axis.

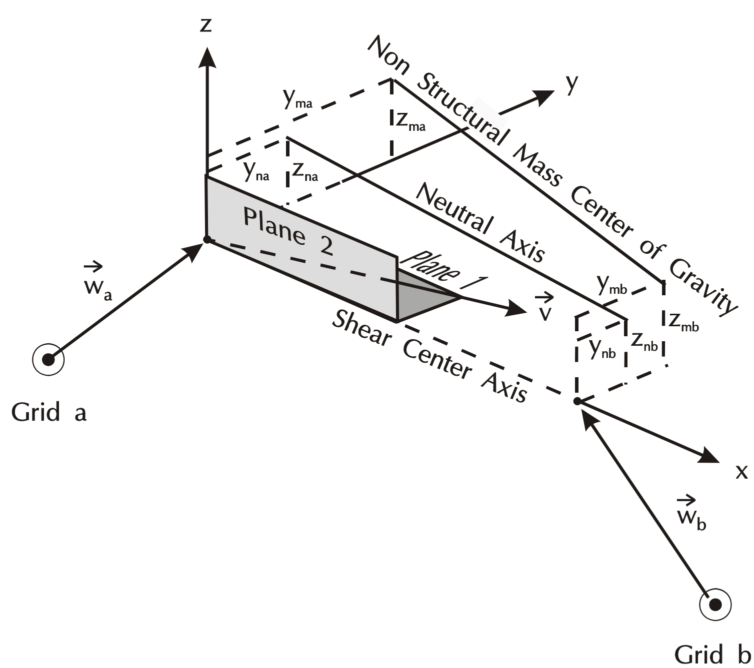


Figure 2-7

Thermal, centrifugal and gravity loads can be applied to the CBEAM elements. Linearly varying distributed tractions and moments along the entire element can be applied in the basic or element coordinate system (**PLOAD1**) or local coordinate system (**PLOADA**).

Six element forces (2 shear forces, 2 bending moments, axial and torque) and 4 stresses (bending plus axial at user defined locations) are recovered at each end and at each intermediate section.

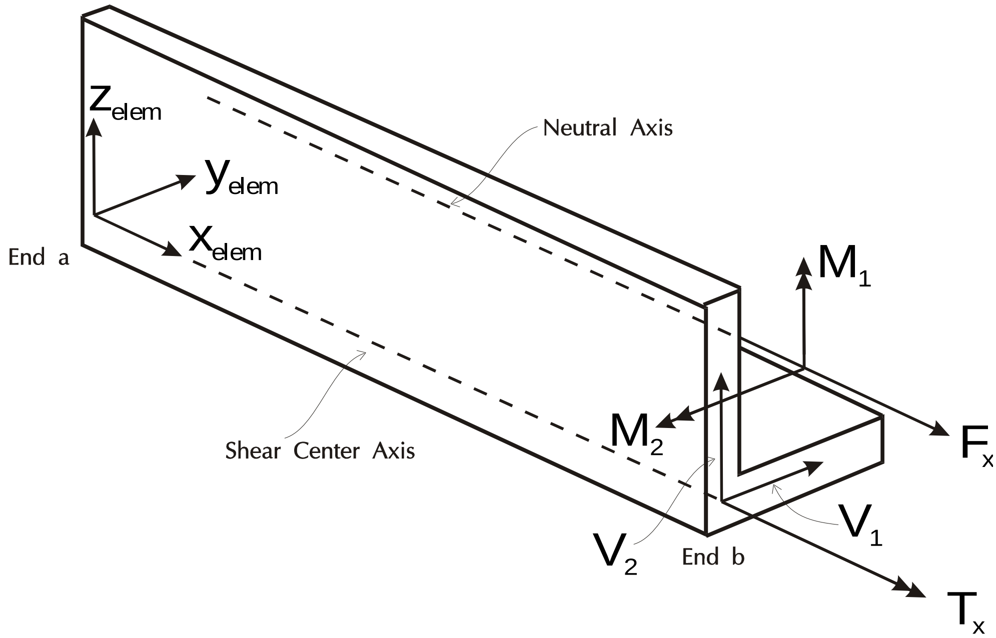


Figure 2-8

2.4.4 Shear Panel (CSHEAR)

The **CSHEAR** element connects four grid points and resists tangential (shearing) forces along its edges. It can also have extensional stiffness. The element thickness, material, nonstructural mass, and extensional stiffness parameters are defined on the **PSHEAR** input data. Only isotropic (**MAT1**) materials can be referenced. Only a lumped mass matrix is generated. This element is ignored in heat transfer analysis.

Geometric (differential) stiffness is calculated for this element for buckling analysis.

Thermal, gravity, centrifugal, and pressure (**PLOAD2** and **PLOAD5**) loads can be applied to the shear panel element. Thermal loads are only calculated if the element has extensional stiffness.

The shear stress in the element coordinate system is calculated at each grid point. In addition the average and largest (in absolute value) of the four corners stresses are calculated.

Element forces at each corner along both connecting edges are calculated. Kick forces that are normal to the element surface are also calculated. In addition, the shear flow forces along each edge are calculated.

The figure below defines the grid numbers associated with the four node quadrilateral shear panel element.

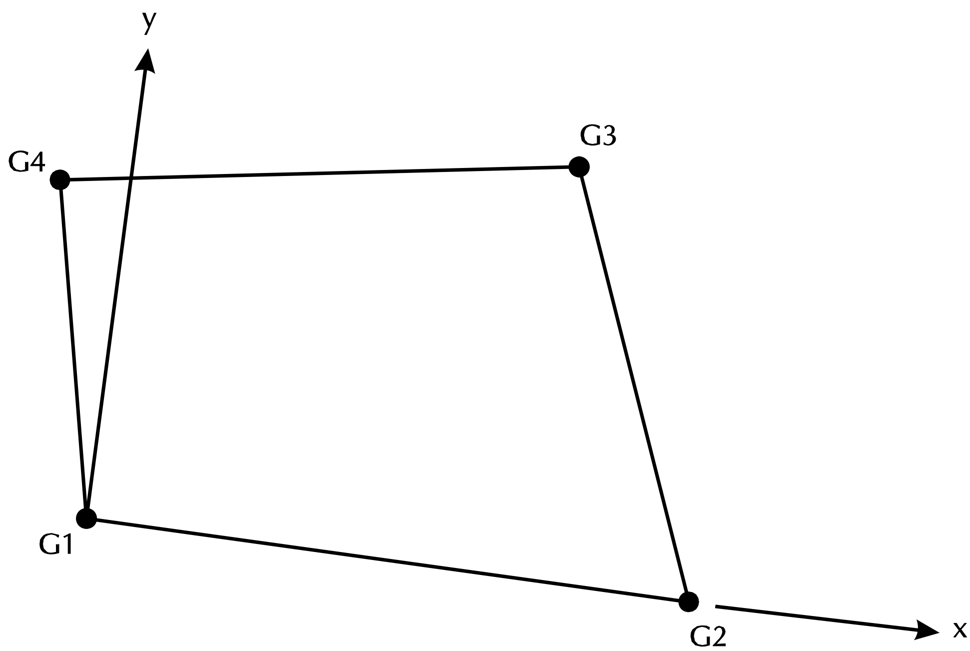


Figure 2-9

The figure below defines the sign convention for the forces acting on the CSHEAR element.

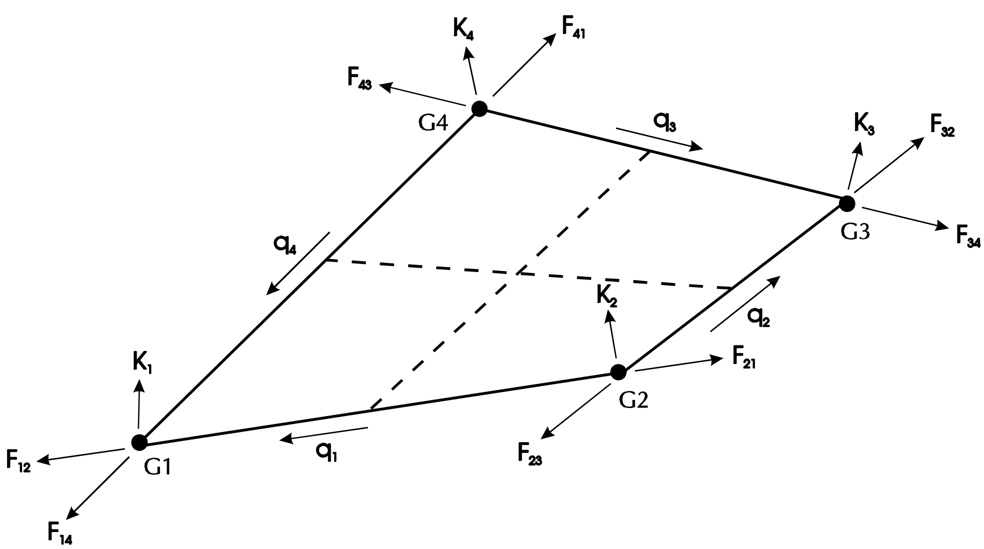


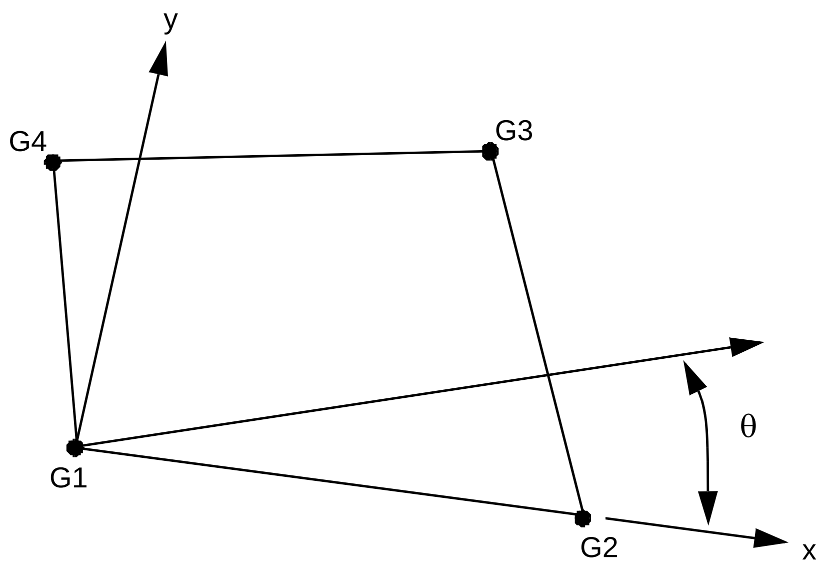
Figure 2-10

2.4.5 Plate/Shell Elements (CQUAD4, CTRIA3, CQUAD8 and CTRIA6 referencing PSHELL data)

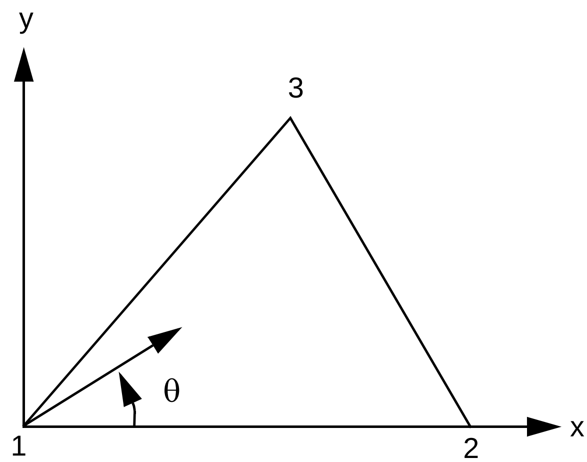
The **CQUAD4** and **CTRIA3** elements have in-plane (membrane) and bending stiffness. Transverse shear deformation is optionally included. The **CQUAD8** and **CTRIA6** elements are general purpose curved shell elements suitable for thick and thin shell analysis and require the specification of membrane, bending and transverse shear material properties. All four elements can be offset from their grid points.

For homogeneous elements, the element thickness, bending stiffness, shear deformation thickness, material, and nonstructural mass per unit area are defined in the **PSHELL** input data. Isotropic (**MAT1**), orthotropic (**MAT8**), and anisotropic (**MAT2**) materials can be used. The material orientation can be defined in the element coordinate system, or the basic or local grid point coordinate systems. The element stresses, strains and forces can be calculated in the element, material, basic, or any defined coordinate system.

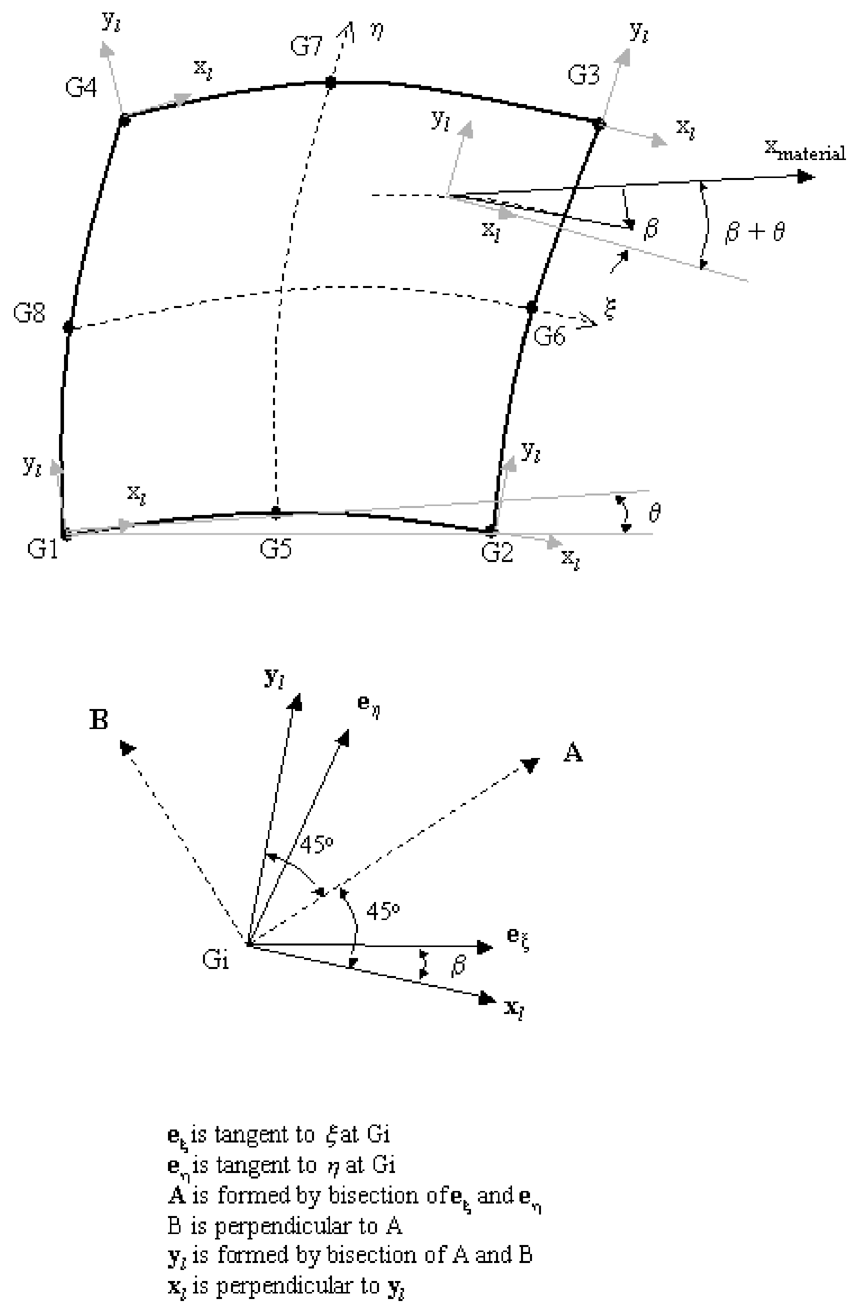
The figure below defines the ordering of the grid numbers associated with the four node quadrilateral plate/shell element. On the CQUAD4 data, the grid point identifiers must be specified in this order. The local material coordinate system is defined by the angle, θ. On the CQUAD4 data, if θ is defined by an integer value, it will refer to the coordinate system with that CID.



The figure below defines the ordering of the grid numbers associated with the three node triangular plate element. On the CTRIA3 data, the grid point identifiers must be specified in this order. The local material coordinate system is defined by the angle, θ. On the CTRIA3 data, if θ is defined by an integer value, it will refer to the coordinate system with that CID.



The element coordinate systems of the CQUAD8 and CTRIA6 curved shell elements are shown in the figures below. On the CQUAD8 and CTRIA6 data, the grid point numbers must be specified in the order shown. The angle, θ, is the orientation angle for material properties.



G1
G2
G3
G4
G5
G6
G7
G8
$x_{material}$
β
$\beta + \theta$
θ
ξ
η
x_l
y_l
B
A
e_η
e_ξ
45°
45°
Gi
$\mathbf{e}_\xi$ is tangent to ξ at Gi
$\mathbf{e}_\eta$ is tangent to η at Gi
$\mathbf{A}$ is formed by bisection of $\mathbf{e}_\xi$ and $\mathbf{e}_\eta$
B is perpendicular to A
$\mathbf{y}_l$ is formed by bisection of A and B
$\mathbf{x}_l$ is perpendicular to $\mathbf{y}_l$

Figure 2-11

2

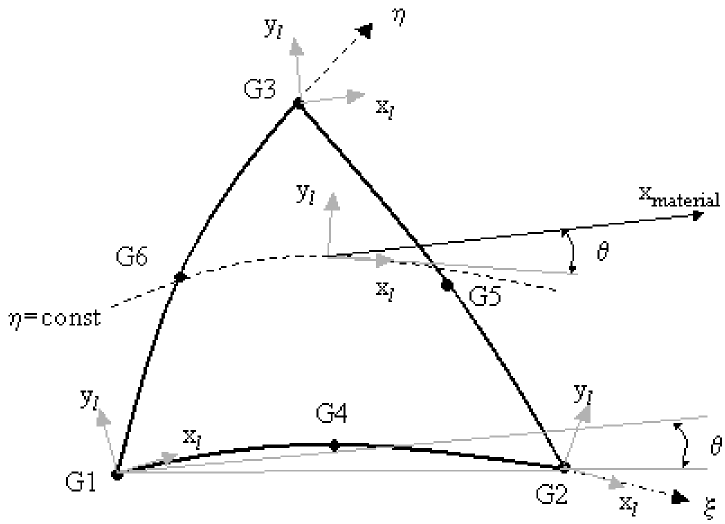


Figure 2-12

The element coordinate system for the curved shell elements is defined locally at each point (ξ,η) such that the plane containing $\mathbf{x}_{elem}$ and $\mathbf{y}_{elem}$ is tangent to the surface of the element. Additionally, for the CQUAD8 element, $\mathbf{x}_{elem}$ and $\mathbf{y}_{elem}$ are obtained by the procedure described in figure below. For the CTRIA6 element, $\mathbf{x}_{elem}$ is tangent to the line of constant η.

For CQUAD4 and CTRIA3 elements, consistent or lumped mass formulation can be used to generate the element mass matrix. The inplane (drilling) rotation is used in the membrane stiffness formulation. For CQUAD8 and CTRIA6 elements, consistent, lumped or coupled mass matrices can be used to generate the element mass matrices.

Geometric (differential) stiffness is calculated for all four elements for buckling analysis.

Thermal, centrifugal and gravity loads can be applied to plate/shell elements. Uniform pressure loads over the entire surface can be applied normal to the surface (**PLOAD2**) or in a direction specified in the basic or in a local coordinate system (**PLOAD4**).

Six midplane strains are recovered in the coordinate system specified in the PSHELL data (default is the element coordinate system), (3 inplane strains: ε_x^0, ε_y^0, γ_{xy}^0 and 3 bending curvatures: κ_x, κ_y and κ_{xy}). From these, the inplane strains on the lower and upper surface are calculated using the relationships:

$$\varepsilon_x = \varepsilon_x^0 - z\kappa_x \tag{2-1}$$

$$\varepsilon_y = \varepsilon_y^0 - z\kappa_y \tag{2-2}$$

$$\gamma_{xy} = \gamma_{xy}^0 - z\kappa_{xy} \tag{2-3}$$

where z is the fiber distance and the positive direction is determined using the right hand rule applied to the element's grid points.

In static analysis, the principal, maximum shear, and von Mises strains are calculated on each surface using the relationships:

$$\varepsilon_1 = \frac{\varepsilon_x + \varepsilon_y}{2} + \sqrt{\frac{(\varepsilon_x - \varepsilon_y)^2}{4} + \frac{\gamma_{xy}^2}{4}} \tag{2-4}$$

$$\varepsilon_2 = \frac{\varepsilon_x + \varepsilon_y}{2} - \sqrt{\frac{(\varepsilon_x - \varepsilon_y)^2}{4} + \frac{\gamma_{xy}^2}{4}} \tag{2-5}$$

$$\gamma_{max} = \sqrt{(\varepsilon_x - \varepsilon_y)^2 + \gamma_{xy}^2} \tag{2-6}$$

$$\gamma_{vm} = \sqrt{\frac{4(\varepsilon_x^2 + \varepsilon_y^2 - \varepsilon_x\varepsilon_y)}{9} + \frac{\gamma_{xy}^2}{3}} \tag{2-7}$$

Six element forces (3 inplane forces: N_x, N_y, and N_{xy} and 3 bending moments: M_x, M_y, and M_{xy}) are recovered in the coordinate system specified on the PSHELL data.

The surface stresses are recovered from the element forces using the relationships:

$$\sigma_x = \frac{N_x}{t} - \frac{zM_x}{D} \tag{2-8}$$

$$\sigma_y = \frac{N_y}{t} - \frac{zM_y}{D} \tag{2-9}$$

$$\sigma_{xy} = \frac{N_{xy}}{t} - \frac{zM_{xy}}{D} \tag{2-10}$$

where z is the fiber distance and the positive direction is determined using the right hand rule applied to the grid points and D is the plate bending stiffness. For homogeneous isotropic plates, $D = t^3/12$.

In static analysis, the principal, maximum shear, and von Mises stresses are calculated on each surface using the relationships:

$$\sigma_1 = \frac{\sigma_x + \sigma_y}{2} + \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \sigma_{xy}^2} \tag{2-11}$$

$$\sigma_2 = \frac{\sigma_x + \sigma_y}{2} - \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \sigma_{xy}^2} \tag{2-12}$$

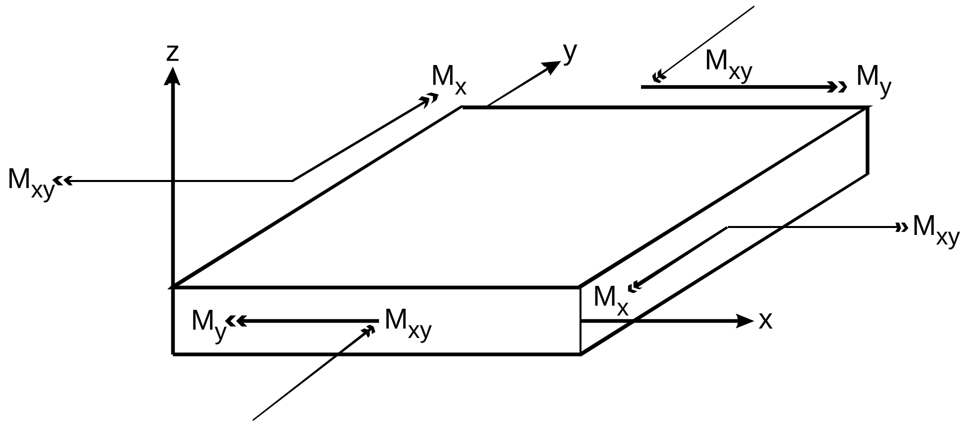
2

$$\tau_{max} = \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \sigma_{xy}^2} \tag{2-13}$$

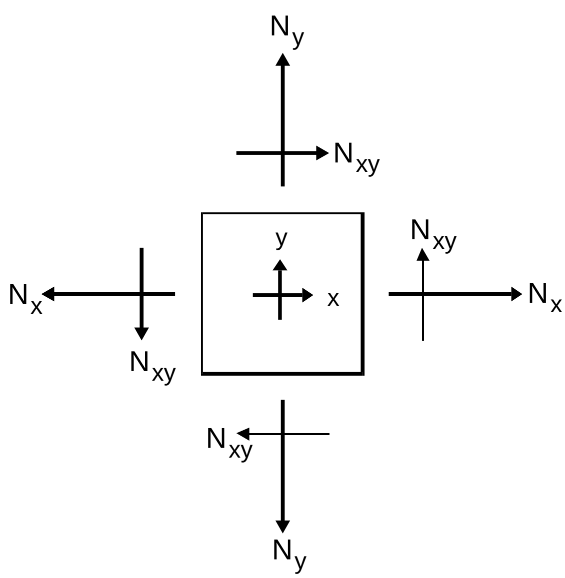
$$\sigma_{vm} = \sqrt{\frac{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2}{2} + 3\sigma_{xy}^2} \tag{2-14}$$

where $\sigma_z = 0$ for plane stress analysis and $\sigma_z = \nu(\sigma_x + \sigma_y) - \alpha\Delta T$ for plane strain analysis.

The figure below defines the sign convention for forces and stresses in the CQUAD4, CTRIA3, CQUAD8 and CTRIA6 elements. Stresses and strains for plate/shell elements have the same sign convention as the element forces.



(a) Plate element forces



(b) Membrane element forces

2.4.6 Composite Elements (CQUAD4, CTRIA3, CQUAD8 and CTRIA6 referencing PCOMP/PCOMPG data)

The **CQUAD4** and **CTRIA3** elements have in-plane (membrane), bending and membrane-bending coupling stiffness. Transverse shear deformation is optionally included in the material it references. The **CQUAD8** and **CTRIA6** elements are general purpose curved shell elements suitable for thick and thin shell analysis. They have membrane, bending and transverse shear stiffness. All four elements can be offset from their grid points.

For composite elements, each individual layer thickness, material, and material orientation is described in the **PCOMP** or **PCOMPG** data. The material orientation for each layer is specified with respect to the material property orientation specified in the CQUAD4, CTRIA3, CQUAD8 or CTRIA6 data. Isotropic (**MAT1**), orthotropic (**MAT8**) or anisotropic (**MAT2**) materials can be used for each layer. In addition, the nonstructural mass per unit area, structural damping coefficient, reference temperature, and laminate failure theory or user supplied stress failure equation (**FINDEX**) or user supplied strain failure equation (**FINDEXN**) are specified in the PCOMP data. The element forces are calculated in the element coordinate system. In addition, the stresses and strains in each layer are calculated in the layer's material coordinate system.

The figure below defines the ordering of the grid numbers associated with the four node quadrilateral composite element. On the CQUAD4 data, the grid point identifiers must be specified in this order. The local material coordinate system is defined by the angle, θ. On the CQUAD4 data, if θ is defined by an integer value, it will refer to the coordinate system with that CID. The layer orientation is defined by the θ_i angle in the PCOMP data statement. To request membrane properties only, the MEM parameter in PCOMP (field 10) must be set to "MEM."

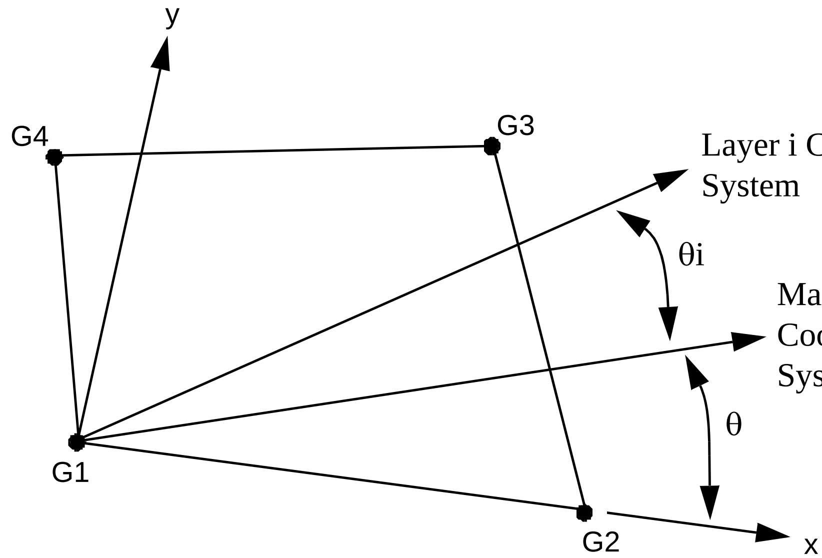


Figure 2-13

The figure below defines the ordering of the grid numbers associated with the three node triangular plate element. On the CTRIA3 data, the grid point identifiers must be specified in this order. The local material coordinate system is defined by the angle, θ. On the CTRIA3 data, if θ is defined by an integer value, it will refer to the coordinate system with that CID. The layer orientations are defined by the θ_i angles in the PCOMP data statements.

To request membrane properties only, the MEM parameter in the PCOMP data statement (field 10) must be set to “MEM.”

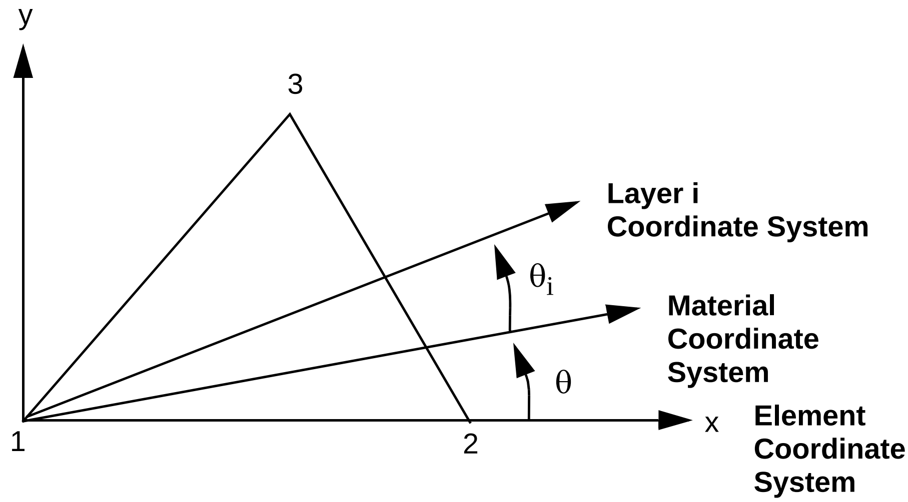


Figure 2-14

The figure below shows the location of the composite in the 3 direction, due to grid offset specified in the CQUAD4/ CTRIA3 data. The figure also shows the definition of z_0 and the convention used to number the layers.

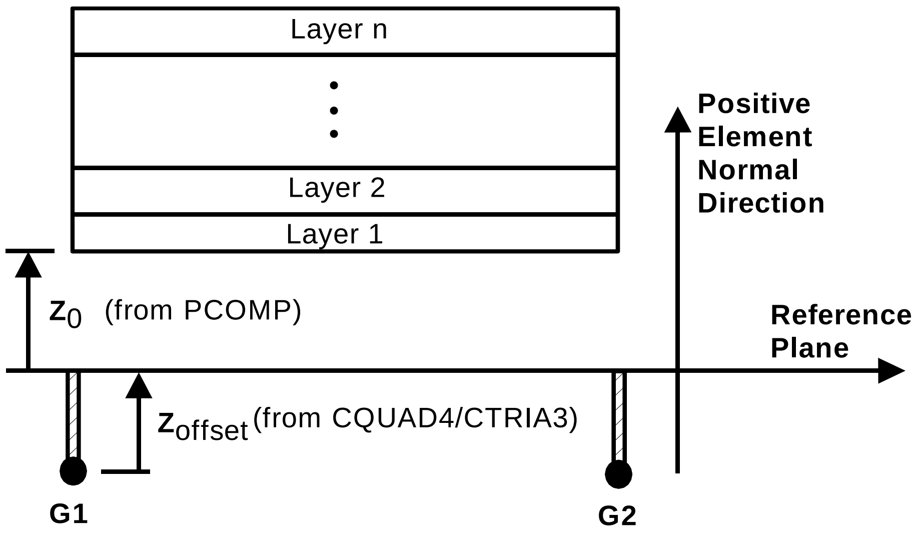


Figure 2-15

For CQUAD8 and CTRIA6 composite elements, the ordering of grids is same at that shown in figures 2.11 and 2.12, but the orientation of the layers are with respect to a material coordinate system which is defined locally at each point (ξ,η).

The user supplied thickness, orientation and material property of each layer is used to generate equivalent PSHELL and MAT2 properties which completely reflect the entire stack of layers. With these properties, the entire stack of layers is modeled by a single plate or shell element. Since CQUAD8 and TRIA6 are general purpose shell elements, the equivalent PSHELL properties must have non-zero shear properties.

For CQUAD4 and CTRIA3 elements, consistent or lumped mass formulation can be used to generate the element mass matrix. The inplane (drilling) rotation is used in the membrane stiffness formulation. For CQUAD8 and CTRIA6 elements, consistent, lumped or coupled mass matrices can be used to generate the element mass matrices.

Geometric (differential) stiffness is calculated for the elements for buckling analysis.

Thermal, centrifugal and gravity loads can be applied to composite elements. Uniform pressure loads over the entire surface can be applied normal to the surface (**PLOAD2**) or in a direction specified in the basic or in a local coordinate system (**PLOAD4**).

Six midplane strains are internally calculated in the element coordinate system, (3 inplane strains: ε_x^0, ε_y^0, γ_{xy}^0 and 3 bending curvatures: κ_x, κ_y and κ_{xy}). From these, the inplane strains on the middle of each layer are calculated using the relationships:

$$\varepsilon_x = \varepsilon_x^0 - z\kappa_x \tag{2-15}$$

$$\varepsilon_y = \varepsilon_y^0 - z\kappa_y \tag{2-16}$$

$$\gamma_{xy} = \gamma_{xy}^0 - z\kappa_{xy} \tag{2-17}$$

where z is the distance from the reference plane to the middle of the layers. The strains are then transformed to the layer coordinate system.

In static analysis, the principal, maximum shear, and von Mises strains are calculated on each layer using the relationships:

$$\varepsilon_1 = \frac{\varepsilon_x + \varepsilon_y}{2} + \sqrt{\frac{(\varepsilon_x - \varepsilon_y)^2}{4} + \frac{\gamma_{xy}^2}{4}} \tag{2-18}$$

$$\varepsilon_2 = \frac{\varepsilon_x + \varepsilon_y}{2} - \sqrt{\frac{(\varepsilon_x - \varepsilon_y)^2}{4} + \frac{\gamma_{xy}^2}{4}} \tag{2-19}$$

$$\gamma_{max} = \sqrt{(\varepsilon_x - \varepsilon_y)^2 + \gamma_{xy}^2} \tag{2-20}$$

$$\gamma_{vm} = \sqrt{\frac{4(\varepsilon_x^2 + \varepsilon_y^2 - \varepsilon_x\varepsilon_y)}{9} + \frac{\gamma_{xy}^2}{3}} \tag{2-21}$$

Six element forces (3 inplane forces: N_x, N_y, and N_{xy} and 3 bending moments: M_x, M_y, and M_{xy}) are recovered in the element coordinate system.

The layer stresses are recovered from the layer strains using the material constants In static analysis, the principal, maximum shear, and von Mises stresses are calculated at the middle of each layer using the relationships:

$$\sigma_1 = \frac{\sigma_x + \sigma_y}{2} + \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \sigma_{xy}^2} \tag{2-22}$$

$$\sigma_2 = \frac{\sigma_x + \sigma_y}{2} - \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \sigma_{xy}^2} \tag{2-23}$$

$$\tau_{max} = \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \sigma_{xy}^2} \tag{2-24}$$

$$\sigma_{vm} = \sqrt{\frac{(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2}{2} + 3\sigma_{xy}^2} \tag{2-25}$$

where $\sigma_z = 0$.

Additionally, the failure index for each layer is also calculated if a failure theory is specified in the PCOMP data and material limits are included in the material data. If the STRN failure theory is specified the failure mode is also calculated (1=failure in the longitudinal direction, 2=failure in the transverse direction, and 12=failure in shear). Only the failure index can be used as a response in design optimization.

The failure indexes are calculated using either built-in equations (STRN,HILL,HOFF or TSAI) or user supplied equations (**FINDEX**,**FINDEXN**).

The built-in equations are:

Maximum Strain (STRN);

$$FI = MAX\left(\frac{\varepsilon_1}{X_1}, \frac{\varepsilon_2}{X_2}, \frac{\gamma_{12}}{X_{12}}\right) \tag{2-26}$$

where

ε_1 = Strain in direction 1 (fiber direction, not principal direction 1).

ε_2 = Strain in direction 2 (transverse to the fiber direction, not principal direction 2).

$\gamma 12$ = In-plane shear strain.

$$X1 \text{ is } \begin{cases} \dfrac{X_T}{E1} & \text{if} \quad \sigma_1 \geq 0 \\ \dfrac{X_C}{E1} & \text{if} \quad \sigma_1 < 0 \end{cases} \tag{2-27}$$

$$X2 \text{ is } \begin{cases} \dfrac{Y_T}{E2} & \text{if} \quad \sigma_2 \geq 0 \\ \dfrac{Y_C}{E2} & \text{if} \quad \sigma_2 < 0 \end{cases} \tag{2-28}$$

$$X_{12} = \frac{S}{G12} \tag{2-29}$$

E1 = Modulus of elasticity in fiber direction 1.

E2 = Modulus of elasticity in fiber direction 2.

G12 = Inplane shear modulus.

Note: On MAT8 entries, strain limits can be entered directly by entering 1.0 in the STRN field. In this case, X_T, X_C, Y_T, Y_C and S are not divided by E1, E2 or G12.

Hill (HILL);

$$FI = \frac{\sigma_I^2}{X_c^2} - \frac{\sigma_I \sigma_{II}}{X_c^2} + \frac{\sigma_{II}^2}{Y_c^2} + \frac{\tau_{I\,II}^2}{S^2} \tag{2-30}$$

Note that the Hill theory is only applicable for orthotropic materials with equal strengths in tension and compression. In other words, $X_C = - X_T$ and $Y_C = - Y_T$.

Hoffman (HOFF);

$$FI = \sigma_I\left(\frac{1}{X_T} + \frac{1}{X_C}\right) + \sigma_{II}\left(\frac{1}{Y_T} + \frac{1}{Y_C}\right) - \frac{\sigma_I^2}{X_T X_C} - \frac{\sigma_{II}^2}{Y_T Y_C} + \frac{\tau_{I\,II}^2}{S^2} + \frac{\sigma_I \sigma_{II}}{X_C X_T} \tag{2-31}$$

Tsai-Wu (TSAI);

$$FI = \sigma_I\left(\frac{1}{X_T} + \frac{1}{X_C}\right) + \sigma_{II}\left(\frac{1}{Y_T} + \frac{1}{Y_C}\right) - \frac{\sigma_I^2}{X_T X_C} - \frac{\sigma_{II}^2}{Y_T Y_C} + \frac{\tau_{I\,II}^2}{S^2} + 2F_{12}\sigma_I \sigma_{II} \tag{2-32}$$

where

X_C is the allowable compressive stress in the I direction (input in MAT8 or MAT1)

X_T is the allowable tensile stress in the I direction (input in MAT8 or MAT1)

Y_C is the allowable compressive stress in the II direction (input in MAT8 or MAT1)

Y_T is the allowable tensile stress in the II direction (input in MAT8 or MAT1)

S is the allowable shear stress in the principal material system

F12 is the interaction term in the tensor polynomial theory of Tsai-Wu (input in MAT8)

σ_I is the stress in the I direction

σ_{II} is the stress in the II direction

Note: I = fiber direction; II = transverse to fiber direction.

2.4.7 Axisymmetric elements (CTRIAX6)

Axisymmetric **CTRIAX6** elements can reference isotropic (**MAT1**) or orthotropic (**MAT3**) materials. Thermal, centrifugal and gravity loads can be applied to the axisymmetric elements. Thermal loads are generated using grid point temperatures, not average element temperatures. Pressure loads can be applied normal to any face or in a direction specified by an angle using the **PLOADX1** input data.

The figure below defines the ordering of the grid numbers associated with the six node axisymmetric element. On this element, the six grid point numbers must be specified in the order shown. The local material coordinate system is defined by the angle θ.

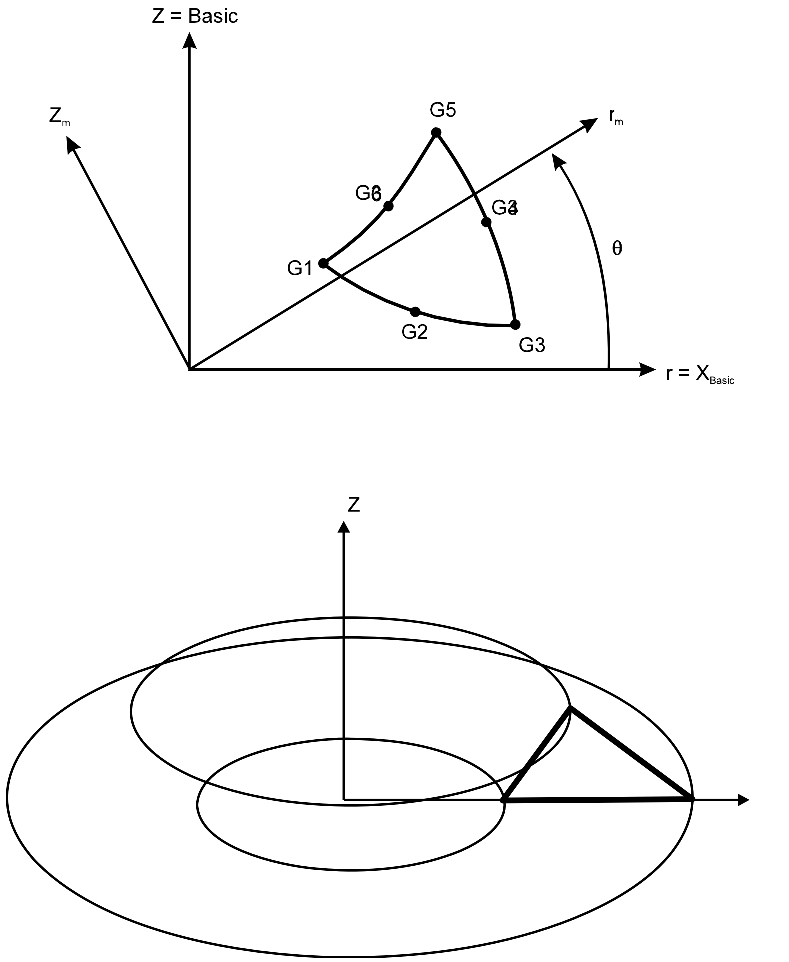


Figure 2-16

The figure below shows the stress convention for a CTRIAX6 axisymmetric element.

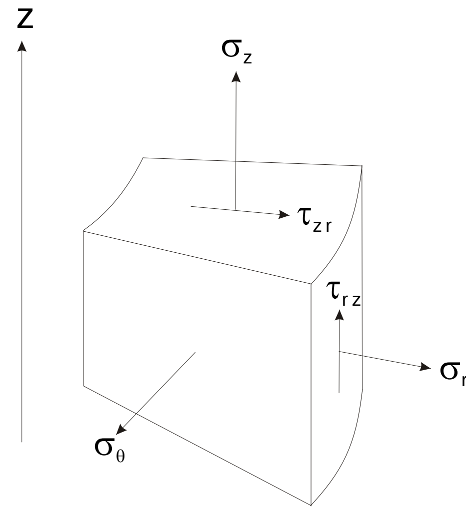


Figure 2-17

Stresses and strains are calculated at the centroid of the element in the material coordinate system which is specified on the MAT3 input data. The material coordinate system can be the basic coordinate system (default). In static analysis, the three principal stresses (or strains) are calculated by solving the 3x3 eigenvalue problem and sorting from maximum to minimum. The von Mises shear stress, Octahedral stress, maximum shear stress, and mean pressure are calculated in static analysis using the relationships:

$$\tau_{oct} = \frac{\sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}}{3} \tag{2-33}$$

$$\tau_{max} = \frac{\sigma_1 - \sigma_3}{2} \tag{2-34}$$

$$\sigma_{VM} = \frac{3\tau_{oct}}{\sqrt{2}} \tag{2-35}$$

$$p_m = -\left(\frac{\sigma_1 + \sigma_2 + \sigma_3}{3}\right) \tag{2-36}$$

In addition, all the stresses described above can be printed at the grid points. However, in this case, they are printed in the basic coordinate system. The von Mises shear strain, Octahedral strain, maximum shear strain, and delta volume are calculated in static analysis using the relationships:

$$\varepsilon_{oct} = \frac{\sqrt{(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_3 - \varepsilon_1)^2}}{3} \quad (2\text{-}37)$$

$$\varepsilon_{max} = \frac{\varepsilon_1 - \varepsilon_3}{2} \quad (2\text{-}38)$$

$$\varepsilon_{vm} = \sqrt{2}\varepsilon_{oct} \quad (2\text{-}39)$$

$$\frac{\Delta V}{V} = \varepsilon_1 + \varepsilon_2 + \varepsilon_3 \quad (2\text{-}40)$$

2.4.8 Solid elements (CHEXA, CPENTA, CTETRA, CHEX20 and CPYRA)

Solid elements (**CHEXA**, **CHEX20**, **CPENTA**, **CTETRA**,**CPYRA**) can reference isotropic (**MAT1**), orthotropic (**MAT11**) or anisotropic (**MAT9**) materials through the **PSOLID** property entry. A consistent or a lumped mass formulation can be used to generate the element mass matrices. Thermal, centrifugal and gravity loads can be applied to the solid elements.

Geometric (differential) stiffness is calculated for these elements for buckling analysis.

Thermal loads are generated using grid point temperatures, not average element temperatures. Pressure loads can be applied normal to any face or in a direction specified by a vector in the basic or any local coordinate system using the **PLOAD4** input data.

The figure below defines the ordering of the grid numbers associated with the six sided, eight node hexahedron element. On the CHEXA data, the grid point identifiers must be specified in this order.

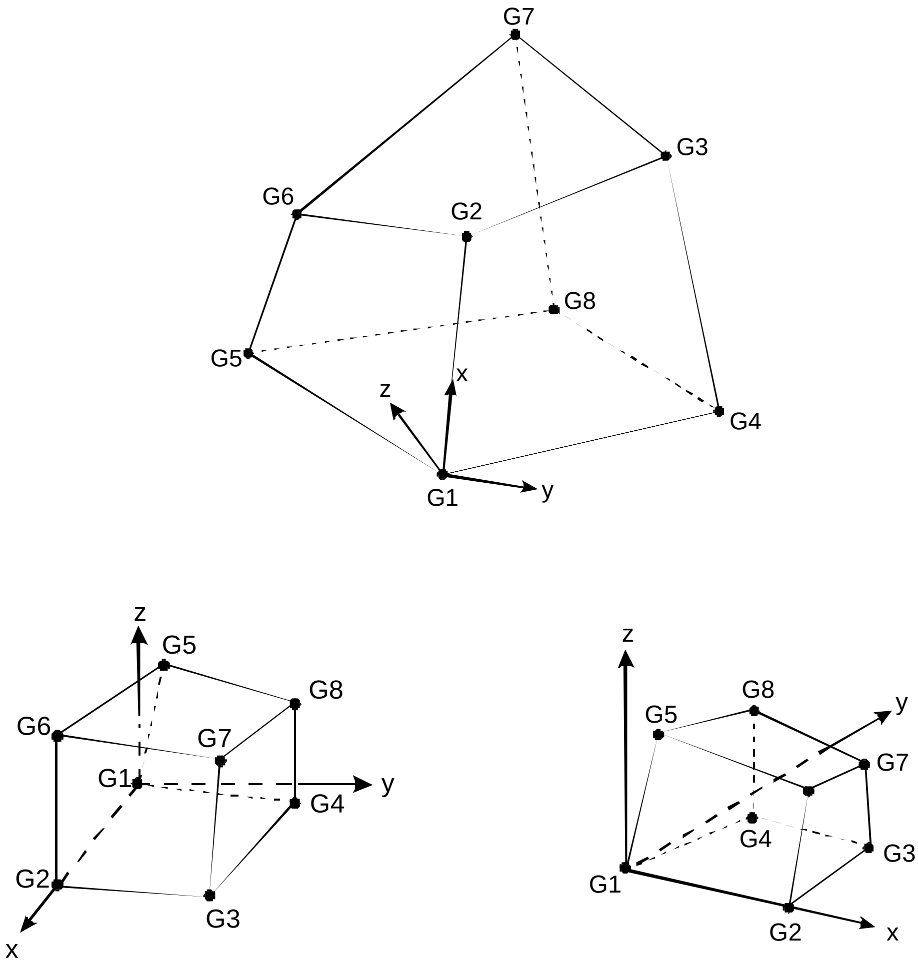


Figure 2-18

The figure below defines the ordering of the grid numbers associated with the six sided hexahedron element. This element may have from eight to twenty one nodes. On the CHEX20 data, the grid point identifiers must be specified in this order. Grids 1 through 8 must be defined. If any of the remaining node identifiers are omitted, the equations for the element are adjusted accordingly.

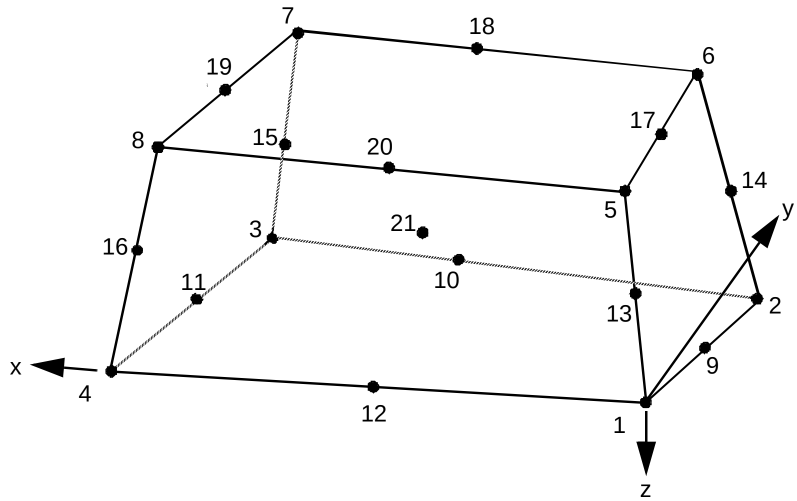


Figure 2-19

The figure below defines the ordering of the grid numbers associated with the five sided, six and fifteen node pentahedral elements. On the CPENTA data, the grid point identifiers must be specified in this order. Additionally, if any of the mid-side nodes are specified, all the mid-side nodes must be specified.

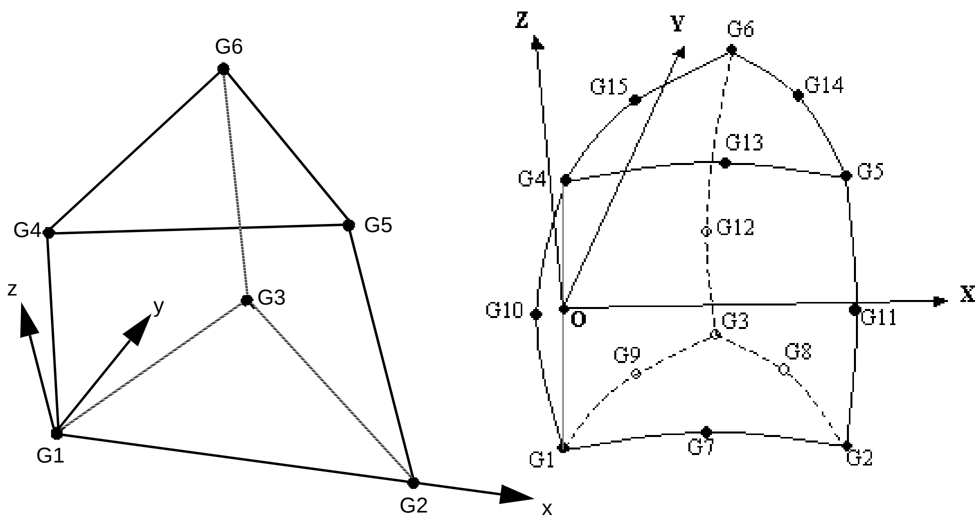


Figure 2-20

The figure below defines the ordering of the grid numbers associated with the four sided, four or ten node tetrahedron element.

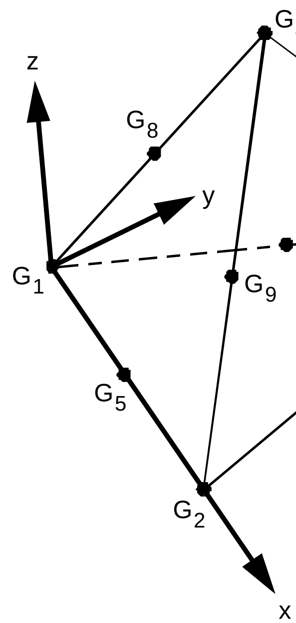


Figure 2-21

The figure below defines the ordering of the grid numbers associated with the five sided, five and thirteen node pyramid elements. On the CPYRA data, the grid point identifiers must be specified in this order. Additionally, if any of the mid-side nodes are specified, all the mid-side nodes must be specified.

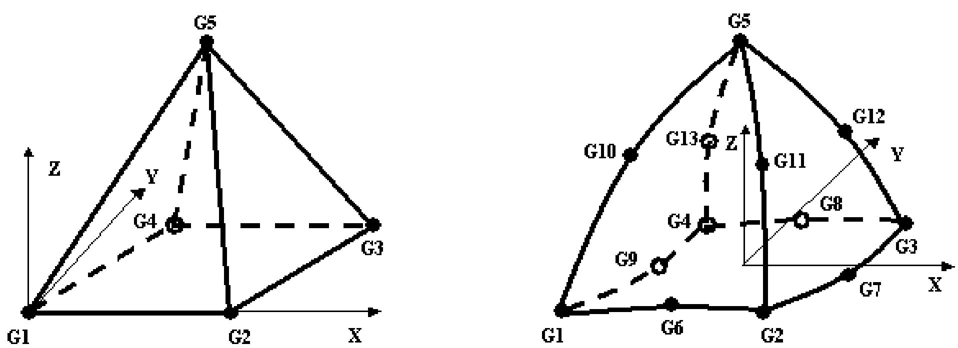


Figure 2-22

Stresses and strains are calculated at the centroid of the element in the material coordinate system which is specified on the PSOLID input data. The material coordinate system can be the basic coordinate system (default), any local coordinate system, or the element coordinate system. In static analysis, the three principal stresses (or strains) are calculated by solving the 3x3 eigenvalue problem and sorting from maximum to minimum. The von Mises shear stress, Octahedral stress, maximum shear stress, and mean pressure are calculated in static analysis using the relationships:

$$\tau_{oct} = \frac{\sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}}{3} \tag{2-41}$$

$$\tau_{max} = \frac{\sigma_1 - \sigma_3}{2} \tag{2-42}$$

$$\sigma_{VM} = \frac{3\tau_{oct}}{\sqrt{2}} \tag{2-43}$$

$$p_m = -\left(\frac{\sigma_1 + \sigma_2 + \sigma_3}{3}\right) \tag{2-44}$$

In addition, all the stresses described above can be printed at the grid points connected to the CHEXA, CHEX20, CPENTA, CTETRA and CPYRA elements. However, in this case, they are printed in the basic coordinate system. The von Mises shear strain, Octahedral strain, maximum shear strain, and delta volume are calculated in static analysis using the relationships:

$$\varepsilon_{oct} = \frac{\sqrt{(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_3 - \varepsilon_1)^2}}{3} \tag{2-45}$$

$$\varepsilon_{max} = \frac{\varepsilon_1 - \varepsilon_3}{2} \tag{2-46}$$

$$\varepsilon_{vm} = \sqrt{2}\varepsilon_{oct} \tag{2-47}$$

$$\frac{\Delta V}{V} = \varepsilon_1 + \varepsilon_2 + \varepsilon_3 \tag{2-48}$$

2.4.9 Bushing Element (CBUSH)

The **CBUSH** element can be used to add generalized stiffness that connects two grid points or one point having the second point grounded. The static element force is recovered by using the following expression:

$$F = K(u_{2elem} - u_{1elem}) \tag{2-49}$$

where

u_{1elem} = displacements at end A rigidly offset to the spring-damper location, and transformed to element coordinates.

u_{2elem} = displacements at end B rigidly offset to the spring-damper location, and transformed to element coordinates.

$$K = \begin{bmatrix} K_1 & & & & & \\ & K_2 & & & & \\ & & K_3 & & & \\ & & & K_4 & & \\ & & & & K_5 & \\ & & & & & K_6 \end{bmatrix}$$

$$F = \begin{Bmatrix} F_x \\ F_y \\ F_z \\ M_x \\ M_y \\ M_z \end{Bmatrix}$$

2

2

Constant values of K_i can be specified by the **PBUSH** data statement. The K_i terms are in the element coordinate system, which is either specified by CID on the CBUSH entry, or is defined by the end grid locations and an orientation vector. See the following figures:

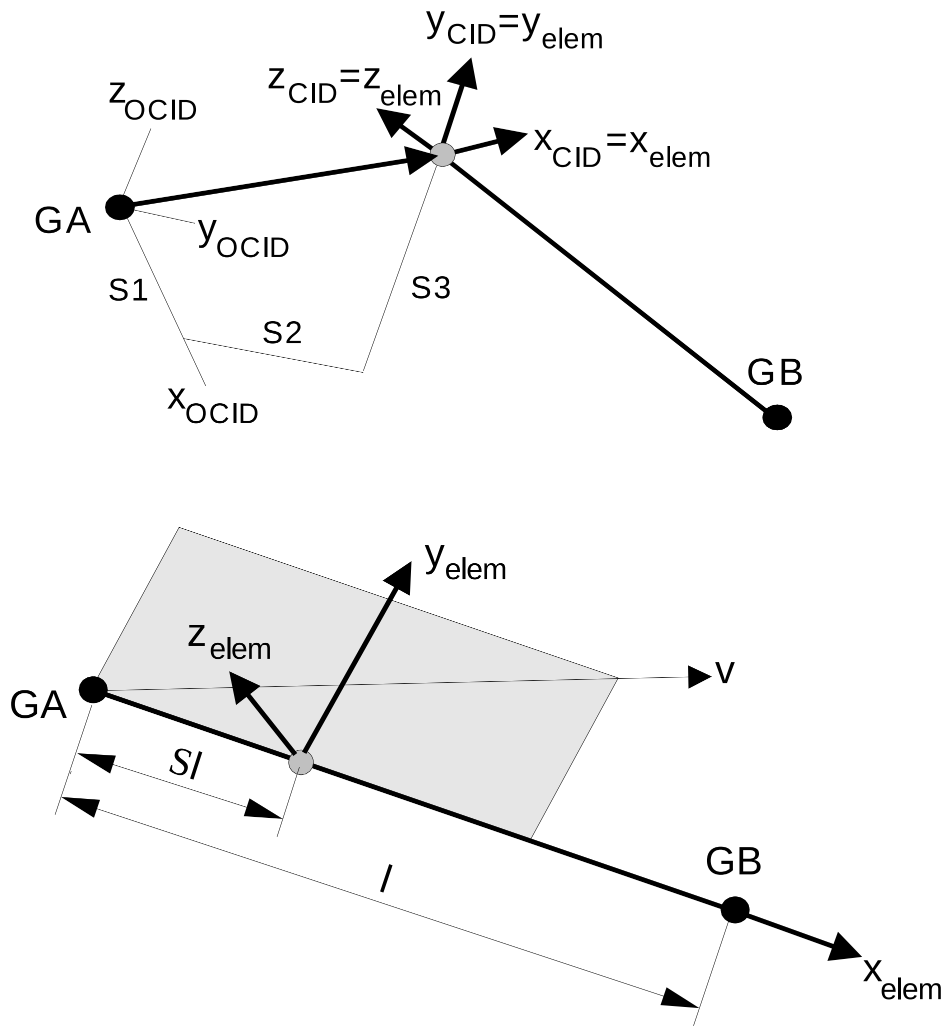


Figure 2-23

Frequency dependent values of K_i can be specified by the **PBUSHT** and **TABLED1**, or **TABLED2**, or **TABLED3**, or **TABLED4** data statements.

Element stresses are calculated by multiplying the recovered element forces by the stress recovery coefficients:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_x \\ \tau_y \\ \tau_z \end{Bmatrix} = \begin{Bmatrix} ST \cdot F_x \\ ST \cdot F_y \\ ST \cdot F_z \\ SR \cdot M_x \\ SR \cdot M_y \\ SR \cdot M_z \end{Bmatrix}$$

Element strains are calculated by multiplying the difference of the displacements by the strain recovery coefficents:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_x \\ \gamma_y \\ \gamma_z \end{Bmatrix} = \begin{bmatrix} ET & & & & & \\ & ET & & & & \\ & & ET & & & \\ & & & ER & & \\ & & & & ER & \\ & & & & & ER \end{bmatrix} (u_{2elem} - u_{1elem})$$

The CBUSH element has no mass and is not affected by thermal, centrifugal or gravity loads. The element is ignored in heat transfer load cases.

2.4.10 Scalar Elastic Element (CELAS1 and CELAS2)

The **CELAS1** and **CELAS2** elastic elements connect two degrees-of-freedom in the general coordinate system, u_1 and u_2, with a stiffness. The degrees-of-freedom are each specified by a grid point and a direction. The element force is recovered using the relationship f = k*(u_1 - u_2), where the element stiffness is specified on the elastic element property data (**PELAS** or in CELAS2). The element stress can also be determined using the stress recovery coefficient from the PELAS or in CELAS2 input data. One end of the elastic element can be grounded (constrained) by replacing the grid ID with 0. The elastic element has no mass properties and is not effected by thermal, centrifugal or gravity loads.

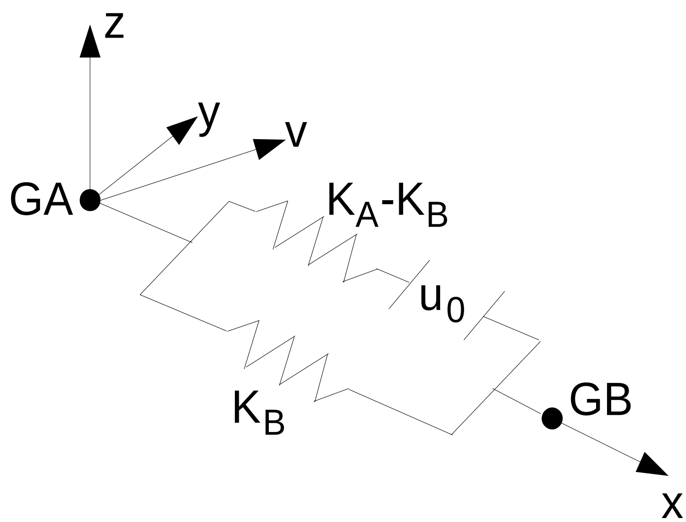
2.4.11 Nonlinear Contact Element (CGAP)

The **CGAP** element can be used to add generalized stiffness that connects two grid points. The properties of the gap elements are specified by the **PGAP** data statement. The stiffness of the element is in the element coordinate system, which is either specified by CID on the CGAP entry, or is defined by the end grid locations and an orientation vector. CGAP elements have two different possible axial stiffnesses, a "closed" stiffness (KA), and an "open" stiffness (KB). Typically, the closed stiffness is very high to simulate a rigid connection, while the open stiffness is very low to simulate disconnection.

In a static loadcase with the **NLPARM** solution control command, the element stiffness is a function of the displacements. To solve the nonlinear system, initial gap states (open/closed) for each gap element will be assumed. The linear static system will be solved, and the displacements will be used to calculate the actual gap states. If the actual gap states match the assumed gap states, then the nonlinear system is solved. If not, then assumed states are updated, and another static iteration begins. The maximum allowed iterations is specified on the NLPARM bulk data entry. If the system is not solved after MAXITER iterations, the program will stop with an error message.

In non-static loadcases, or in static loadcases without the NLPARM solution control command, this element is a linear elastic element with a stiffness determined by U0. If U0 is less than or equal to zero, the gap is closed, and the stiffness is KA. If U0 is greater than zero, the gap is open and the stiffness is KB.

The CGAP element has no mass and is not affected by thermal, centrifugal or gravity loads. The element is ignored in heat transfer load cases.s:



2.4.12 Vector Elastic Element (CVECTOR)

The **CVECTOR** elastic element connects two grid points or one point having the second point grounded. The element force is recovered by using the following expression:

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \begin{bmatrix} K & -K \\ -K & K \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \tag{2-50}$$

where

$$F_1 = \begin{Bmatrix} F_{x1} \\ F_{y1} \\ F_{z1} \\ M_{x1} \\ M_{y1} \\ M_{z1} \end{Bmatrix} \qquad F_2 = \begin{Bmatrix} F_{x2} \\ F_{y2} \\ F_{z2} \\ M_{x2} \\ M_{y2} \\ M_{z2} \end{Bmatrix} = -F_1$$

$$K = \begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16} \\ & K_{22} & K_{23} & K_{24} & K_{25} & K_{26} \\ & & K_{33} & K_{34} & K_{35} & K_{36} \\ & & & K_{44} & K_{45} & K_{46} \\ & \mathrm{SYM} & & & K_{55} & K_{56} \\ & & & & & K_{66} \end{bmatrix}$$

u_1 and u_2 correspond to the displacements at the connection grids.

K_{ij} are specified by the user with the **PVECTOR** data statement. The K_{ij} terms are in the element coordinate system which is defined by the figure below.

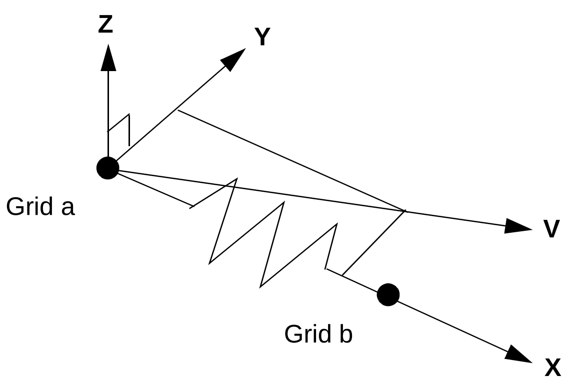


Figure 2-24

Where V is the orientation vector and is defined by the use of the CVECTOR data statement. If the orientation vector is left blank, then the K_{ij} terms are in the output coordinate system of grids.

The vector spring element has no mass and is not affected by thermal, centrifugal or gravity loads. The element is ignored in heat transfer load cases.

This element should be used with care when used with non-coincident grids, because this element make no provisions for maintaining six rigid body modes. The **CBUSH** element should be preferred for most applications.

2.4.13 The General Element

The **GENEL** element is a general element in which either the stiffness or the flexibility matrix is known to the user. If the stiffness matrix is known, the following equation describes the force-displacement relationship;

$$\begin{Bmatrix} F_i \\ F_d \end{Bmatrix} = \begin{bmatrix} K & -KS \\ -S^T K & S^T KS \end{bmatrix} \begin{Bmatrix} u_i \\ u_d \end{Bmatrix} \tag{2-51}$$

If the flexibility matrix is known, the following force-displacement relationship is used;

$$\begin{Bmatrix} u_i \\ f_d \end{Bmatrix} = \begin{bmatrix} Z & S \\ -S^T & 0 \end{bmatrix} \begin{Bmatrix} f_i \\ u_d \end{Bmatrix} \tag{2-52}$$

where

$$u_i = \begin{Bmatrix} u_{i1} \\ u_{i2} \\ \cdot \\ \cdot \\ \cdot \\ u_{im} \end{Bmatrix} \quad \text{are the independent degrees of freedom} \tag{2-53}$$

$$u_i = \begin{Bmatrix} u_{d1} \\ u_{d2} \\ . \\ . \\ . \\ u_{dm} \end{Bmatrix} \quad \text{are the dependent degrees of freedom} \tag{2-54}$$

$$[S] = \begin{bmatrix} S_{11} & S_{12} & . & . & S_{1m} \\ S_{21} & S_{22} & . & . & S_{1m} \\ . & . & . & . & . \\ . & . & . & . & . \\ S_{m1} & S_{m2} & . & . & S_{mm} \end{bmatrix} \quad \text{are the rigid body motion terms} \tag{2-55}$$

The required input are the independent degrees of freedom and either the stiffness or flexibility matrix. Additionally, the rigid body matrix may be input in which case the dependent degrees of freedom have to be input.

If the rigid body matrix is not provided, then *GENESIS* calculates it automatically.

The forces associated with this element are not calculated.

This element is not affected by thermal, centrifugal or gravity loads and it is ignored in heat transfer analysis.

2.4.14 K2UU, K2UU1, M2UU and M2UU1

The user can specify a stiffness matrix and/or a mass matrix to be added to the global stiffness and mass matrices using the **K2UU**, **K2UU1**, **M2UU**, and/or **M2UU1**executive control commands.

K2UU, K2UU1, M2UU and M2UU1 commands simply select files containing the stiffness and mass matrices plus the degrees of freedom associated with them.

K2UU1and M2UU1 are identical to K2UU and M2UU except that K2UU1 and M2UU1 have properties associated to them. The property for K2UU1 is **PK2UU** and the property of M2UU1 is **PM2UU**. The PK2UU and PM2UU property values correspond to scale factors for the stiffness and mass matrix terms, respectively. These properties are designable.

The K2UU/K2UU1 file is a Fortran unformatted sequential access file that has to be written using the following sequence:

```
WRITE(LUN) IDELEM,NEQR,ICODE1,ICODE2
WRITE(LUN) (IDOF(1,J),IDOF(2,J),J=1,NEQR)
ILAST = 0
```

```
   DO 10 J = 1,NEQR
      NROW = NEQR - J + 1
      IFIRST = ILAST + 1
      ILAST = IFIRST + NROW - 1
      WRITE(LUN) (STIFF(I),I=IFIRST,ILAST)
10 CONTINUE
```

where,

LUN	Is the unit number of the unformatted sequential access file.
IDELEM	Currently ignored, use 0.
NEQR	Number of degrees of freedoms.
ICODE1	Currently ignored, use 0.
ICODE2	Currently ignored, use 0.
IDOF(1,J)	Is an integer array that contains the grid numbers (J=1,NEQR).
IDOF(2,J)	Is an integer array that contains the component number (J=1,NEQR).
STIFF(I)	Is a double precision array that contains the lower triangular part of the K2UU stiffness matrix. (I=1,NEQR*(NEQR+1)/2)

For example, if $[K] = \begin{bmatrix} K_{11} & & SYM \\ K_{21} & K_{22} & \\ K_{31} & K_{32} & K_{33} \end{bmatrix}$

and $U = \begin{Bmatrix} u_2 \\ v_3 \\ w_7 \end{Bmatrix}$
displacement 1 at grid 2
displacement 2 at grid 3
displacement 3 at grid 7

Then the K2UU file corresponding to [K] and {U} will contain the following information:

```
0,3,0,0
2,1,3,2,7,3
K11,K21,K31
K22,K32
K33
```

The M2UU/M2UU1 file is a Fortran unformatted sequential access file that has to be written using the following sequence:

```
WRITE(LUN) IDELEM,NEQR,ICODE1,ICODE2
WRITE(LUN) (IDOF(1,J),IDOF(2,J),J=1,NEQR)
ILAST = 0
DO 10 J = 1,NEQR
   NROW = NEQR - J + 1
   IFIRST = ILAST + 1
```

```
      ILAST = IFIRST + NROW - 1
      WRITE(LUN) (MASS(I),I=IFIRST,ILAST)
10 CONTINUE
```

where,

LUN	Is the unit number of the unformatted sequential access file.
IDELEM	Currently ignored, use 0.
NEQR	Number of degrees of freedoms.
ICODE1	Currently ignored, use 0.
ICODE2	Currently ignored, use 0.
IDOF(1,J)	Is an integer array that contains the grid numbers (J=1,NEQR).
IDOF(2,J)	Is an integer array that contains the component number (J=1,NEQR).
MASS(I)	Is a double precision array that contains the lower triangular part of the K2UU stiffness matrix. (I=1,NEQR*(NEQR+1)/2)

For example, if $[M] = \begin{bmatrix} M_{11} & & SYM \\ M_{21} & M_{22} & \\ M_{31} & M_{32} & M_{33} \end{bmatrix}$

and $U = \begin{Bmatrix} u_2 \\ v_3 \\ w_7 \end{Bmatrix}$ displacement 1 at grid 2
displacement 2 at grid 3
displacement 3 at grid 7

Then the M2UU file corresponding to [M] and {U} will contain the following information:

```
0,3,0,0
2,1,3,2,7,3
M11,M21,M31
M22,M32
M33
```


2

2.5 Connector Elements

Connector elements are used to model connections between points, elements, patches or any of their combinations. Here, we briefly define the connector elements available in *GENESIS*.

2.5.1 Weld Element (CWELD)

The **CWELD** element is a general purpose connector element used to model spot weld connections. The connected elements can be of different types and the meshes need not be congruent. With the **CWELD** element and corresponding **PWELD** property entries, the following three connection types can be defined:

- Point-to-Point
- Point-to-Patch
- Patch-to-Patch

For a Point-to-Point connection, upper and lower shell vertex grids GA and GB are connected as shown in **Figure 2-25**. The vector from GA to GB determines the axis and length of the connector.

For a Point-to-Patch connection, a vertex grid point GS of a shell element is connected to a surface patch as shown in **Figure 2-26**. The patch is either defined by an ordered list of grid points, GAi, or a shell element SHIDA. The attachment point GA is obtained by projecting the coordinates of GS onto the patch. The user may also define GA. The vector from GA to GS determines the axis and length of the connector.

For a Patch-to-Patch connection, spot weld grid GS is connected to upper and lower surface patches. The patches can consist of single or multiple elements. For single element patch connections, the lower/upper patches are defined by either grids GAi/GBi or shell elements SHIDA/SHIDB as shown in **Figure 2-27**. For multi-element patch connections, the lower/upper patches are defined by either property identification numbers, PIDA/PIDB, or shell elements SHIDA/SHIDB as shown in **Figure 2-28**. In this case, the elements in patches and connectivity information are automatically determined from GS, PIDA/PIDB or SHIDA/SHIDB. Piercing points GA and GB are determined by projecting GS onto the patches. The user may also define GA and GB. The vector from GA to GB determines the axis and length of the connector. GS can be at an arbitrary location as long as the projected grids GA and GB lie within upper and lower patches. A surface patch must have at least 3 grids and cannot exceed 8 grids. A surface patch need not correspond to an element.

The **SWLDPRM** bulk data entry is provided to override the default parameters used in the **CWELD** connectivity search. Using this, the user can also output connectivity information for debugging purposes.

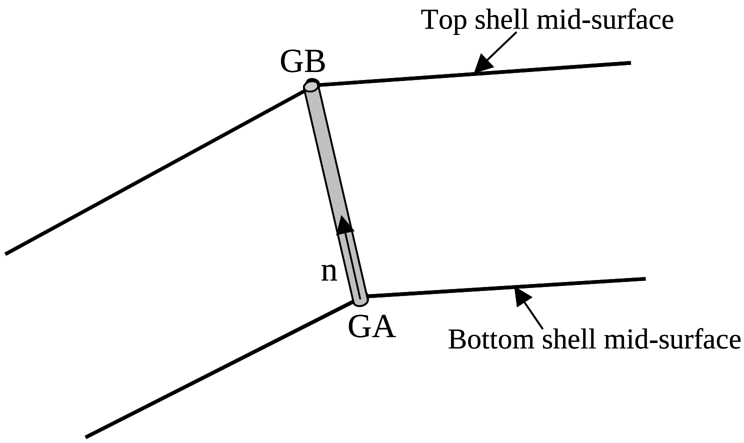


Figure 2-25 Point-to-Point connection

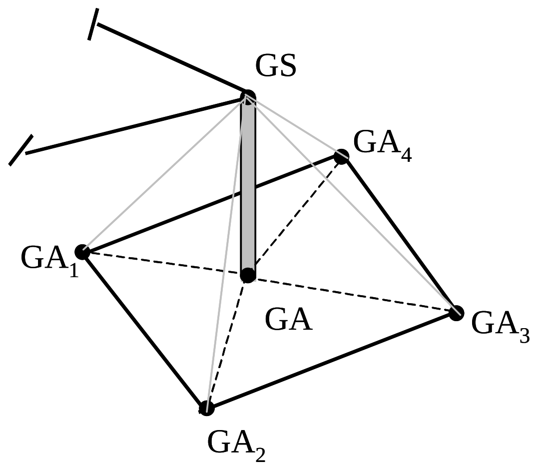


Figure 2-26 Point-to-Patch connection

The **PWELD** property specifies the material identification number and spot weld diameter D. Additionally, the user can define the type of weld to be ‘SPOT’ or general weld and set some advanced parameters.

The finite element model of the **CWELD** element is a shear flexible bar element with two nodes and 12 degrees of freedom. Depending on the type of connection, each node of the bar is connected to either a single node or many nodes belonging to a shell. The six degrees of freedom of each bar node are connected to the three translational degrees of freedom of each shell node with constraints from Kirchoff shell theory.

For Patch-to-Patch connections with multi-element patches, eight auxiliary grids that form an hexahedron are generated using the weld diameter, D. The six degrees of freedom of each beam node are first connected to the three translational degrees of freedom of the four top and the four bottom auxiliary grids using Kirchoff constraints. The translational degrees of freedom of the auxiliary grids are further connected to the translational degrees of freedom of the shell element nodes.

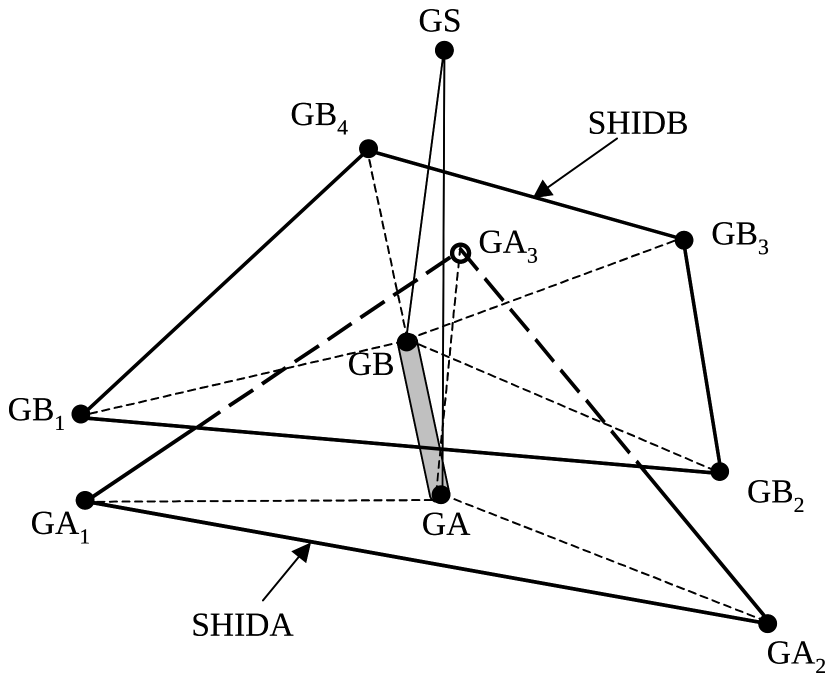


Figure 2-27 Patch-to-Patch connection

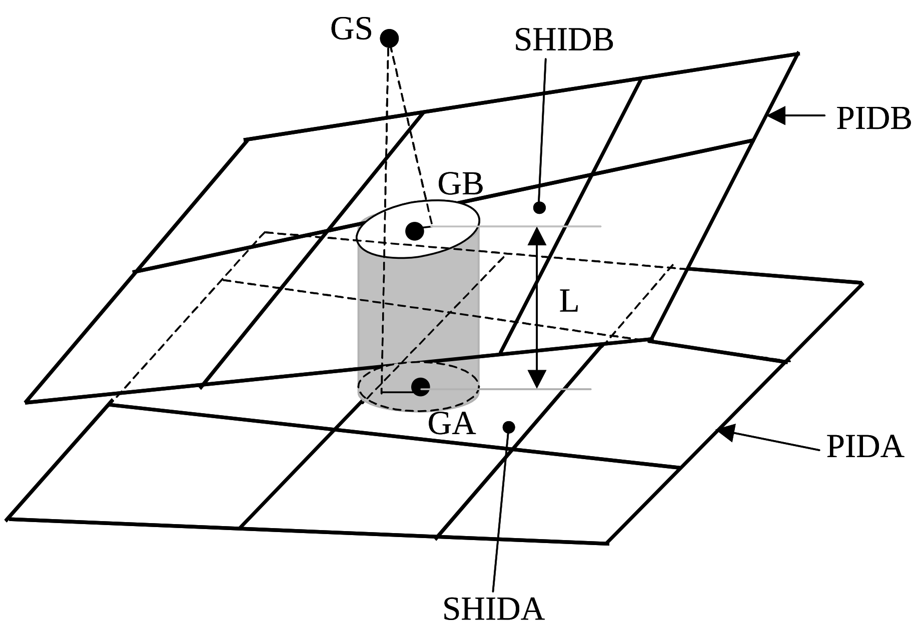


Figure 2-28 Patch-to-Patch connection

2.6 Mass Elements

Two concentrated mass elements and two scalar elements are available. The **CONM2** element has its connection and property data on the same line of input data. The **CONM3** element has its property information defined on the **PCONM3** input data.

The concentrated mass elements have mass and rotational inertia properties. The center of mass can be offset from the grid point by specifying an offset vector. The rotational inertias and offset vector can be defined in the basic or any local coordinate system. Mass elements are not effected by thermal loads.

The scalar mass element, **CMASS1**, can be used to connect any two degrees of freedom of the model or to just connect one degree of freedom to ground. The mass is input using the **PMASS** data statement.

The scalar mass element, **CMASS2**, is the same element as CMASS1. The difference is only on the format. In the CMASS2 data used to specify the degrees of freedom is also used to specify the mass.

The mass elements are ignored in the heat transfer loadcases, but masses are still used for system mass calculations. Mass elements are also used to calculate gravity and centrifugal loads. When the CMASS1 element is connected to scalar points, no gravity or centrifugal loads are created.

2.7 Damping Elements

Two types of damping are available; viscous damping and structural damping. Viscous damping can be specified using the **CDAMP1**, **CDAMP2**, **CVISC** and/or **CBUSH** elements and structural damping can be specified using the elastic elements and the GE coefficients in their materials, or specifying the global damping using the analysis parameter, **G**. In modal dynamic response analysis, it is also possible to specify viscous or structural modal damping using the analysis data statement **TABDMP1** and the analysis parameter **KDAMP**.

2.7.1 Damp Element (CDAMP1 and CDAMP2)

The viscous damping elements **CDAMP1** and **CDAMP2** connect two degrees of freedom in the general coordinate system, u_1 and u_2, with a viscous damping coefficient, b. Each degree of freedom is specified by a grid point and a direction. The complex element force is recovered using the relationship $f = ib\,(u_1 - u_2)$, where the element damping coefficient is specified on the element property data, **PDAMP** for CDAMP1, or in CDAMP2, and i is the imaginary number ($\sqrt{-1}$). The force is printed using the **FORCE** output request in the solution control. One end of the element can be grounded (constrained) by replacing the grid ID with 0.

The element has no elastic or mass properties and is not affected by thermal, centrifugal or gravity loads. This element is only used for dynamic analysis.

2.7.2 Viscous Element (CVISC)

The viscous damping element, **CVISC**, can connect any two grids of the structure. It has two viscous coefficients to represent extensional and/or torsional damping and they are specified using the **PWELD** data statement. Complex element forces and torques can be recovered using the **FORCE** output request of the solution control.

The element has no elastic or mass properties and is not affected by thermal, centrifugal or gravity loads. This element is only used for dynamic analysis.

2.7.3 Bushing Element (CBUSH)

The **CBUSH** element can be used to add generalized viscous damping that connects two grid points or one point having the second point grounded. The dynamic element force is recovered by using the following expression:

$$F = K(u_{2elem} - u_{1elem}) + B(\dot{u}_{2elem} - \dot{u}_{1elem}) \tag{2-56}$$

where

u_{1elem} = displacements at end A rigidly offset to the spring-damper location, and transformed to element coordinates.

u_{2elem} = displacements at end B rigidly offset to the spring-damper location, and transformed to element coordinates.

$\dot{u}_{1elem}$ = velocities at end A rigidly offset to the spring-damper location, and transformed to element coordinates.

$\dot{u}_{2elem}$ = velocities at end B rigidly offset to the spring-damper location, and transformed to element coordinates.

$$K = \begin{bmatrix} K_1 & & & & & \\ & K_2 & & & & \\ & & K_3 & & & \\ & & & K_4 & & \\ & & & & K_5 & \\ & & & & & K_6 \end{bmatrix} ; \quad B = \begin{bmatrix} B_1 & & & & & \\ & B_2 & & & & \\ & & B_3 & & & \\ & & & B_4 & & \\ & & & & B_5 & \\ & & & & & B_6 \end{bmatrix}$$

$$F = \begin{Bmatrix} F_x \\ F_y \\ F_z \\ M_x \\ M_y \\ M_z \end{Bmatrix}$$

K_i and B_i are specified by the **PBUSH** data statement.

2.7.4 Structural Damping Elements

Any of the elastic elements: **CELAS1**, **CELAS2**, **CBUSH**, **CVECTOR**, **CROD**, **CBAR**, **CBEAM**, **CQUAD4**, **CQUAD8**, **CTRIA3**, **CTRIA6**, **CSHEAR**, **CTRIAX6**, **CTETRA**, **CPENTA**, **CPYRA**, **CHEXA** or **CHEX20** can be used to add structural damping to the structure. The damping is added by specifying the corresponding GE coefficient in the appropriate material (**MAT1**, **MAT2**, **MAT3**, **MAT8**, **MAT9** or **MAT11**), or in the appropriate property (**PELAS**, **CELAS2**, **PBUSH**, **PVECTOR**, **PCOMP** or **PCOMPG**).

2.8 Rigid and Interpolation Elements

The rigid and interpolation elements are used as a convenient method of generating multi-point constraints (MPCs). They are applied to all load cases.

The rigid rod (**RROD**) has five independent and one dependent translational degree-of-freedom.

The rigid bar (**RBAR**) has six independent and six dependent degrees-of-freedom.

The general rigid element (**RBE1**) is connected to several grid points with a total of six independent degree-of-freedom and to any number of grids, each with one to six dependent degrees-of-freedom.

The general rigid element (**RBE2**) is connected to one grid point with all six independent degrees-of-freedom and to any number of other grid points with from one to six dependent degrees-of-freedom per grid.

Geometric (differential) stiffness is calculated for the rigid elements RROD, RBAR, and RBE2 for buckling analysis.

The interpolation element **RBE3** connects degrees of freedom so the motion at a reference grid is a weighted average of the motion of the rest of the grid points it references. This element is useful to distribute loads and masses from one point to multiple points. A more detailed explanation is provided in Section **2.8.1**.

The interpolation element **RSPLINE** connects grid points using beam equations. This element is useful to create transition meshes (see Section **2.8.2**).

The interpolation element **BOLT** connects top and bottom portions of a bolt model to a bolt control grid. This combines with the **BOLTSUB** solution control command to facilitate nonlinear contact analysis using bolt preloading. See Section **2.8.3**.

Geometric (differential) stiffness is not calculated for the interpolation elements RBE3, RSPLINE or BOLT or the rigid element RBE1.

Ordinarily, MPC and rigid/interplolation element entries specify which of the connected degrees of freedom are to be made dependent. Degrees of freedom may not be specified as dependent on more than one entry and may not be constrained by SPC or specified on ASET or SUPORT1. If the analysis parameter **AUTOMSET** is set to YES, then the program will automatically choose a set of degrees of freedom to be made dependent that satisfies the above requirements. In this case, the constraint equations generated by the MPC/rigid/interpolation element entries will be preserved, but the dependent d.o.f. specifications will be discarded. Consequently it is allowed for multiple entries to specify the same degree of freedom as dependent, or to constrain specified degrees of freedom with SPC. Note that in this case, it is possible to create a conflict between MPC/rigid/interpolation elements and non-zero SPC/SPCD enforcements such that there is no solution. To resolve this conflict, the incompatible MPC/rigid/interpolation element constraints will be discarded.

The rigid and interpolation elements are not effected by the thermal, centrifugal or gravity loads.

The rigid and interpolation elements are ignored in heat transfer loadcases. However, if a temperature link is needed between two or more grids, **MPC** can be used.

2.8.1 RBE3 Element

The **RBE3** element is an interpolation element designed to redistribute loads and masses in a statically equivalent manner. The dependent degrees of freedom are set to a weighted average of the independent degrees of freedom. The weighting coefficients are computed based on user input weights and the requirement for static equivalency.

The RBE3 creates constraint equations of the following form:

$$u_d = [A]u_i \tag{2-57}$$

where

u_d = the dependent degrees of freedom

u_i = the independent degrees of freedom

A = the matrix of weighting coefficients

This system of constraint equations has the effect of redistributing loads

from the dependent dof to the independent dof in the following manner:

$$F_i = [A]^T F_d \tag{2-58}$$

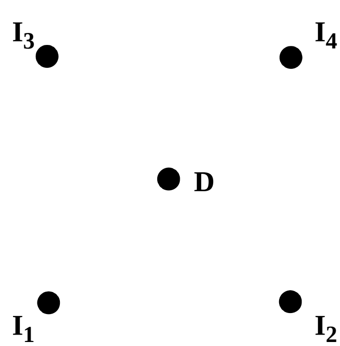
where

F_d = the applied loads at the dependent dof

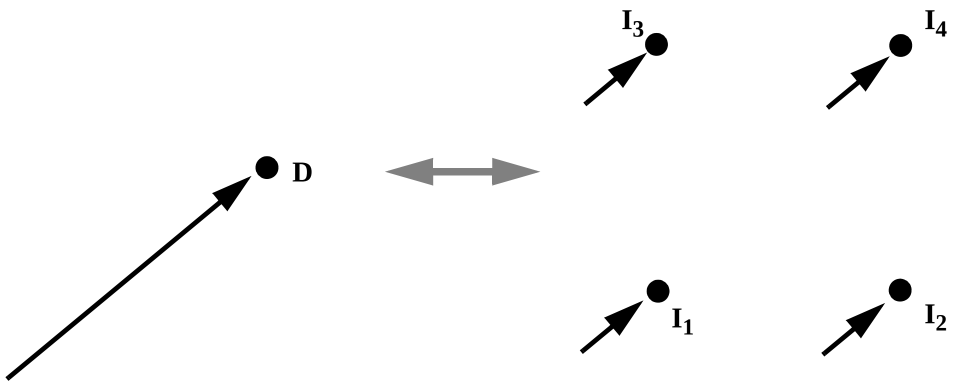
F_i = the effective applied loads at the independent dof

The coefficients in the matrix A are calculated such that F_i and F_d are statically equivalent.

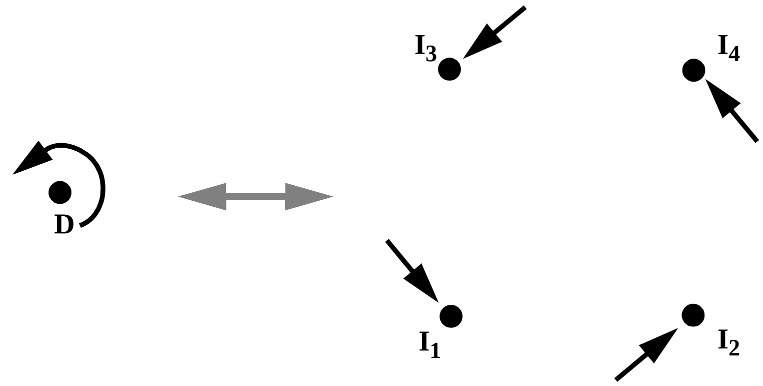
Consider the following example: an RBE3 is used to connect the degrees of freedom at grid D (the dependent or reference dof) with the translational dof at grids I_j (the independent dof) with equal user weighting.



A force applied at D will result in equivalent forces at I_j:



A moment normal to the plane applied at D will result in equivalent forces at I_j:



Note that if the RBE3 had also connected the rotational dof at the independent grids, then the applied moment would result in equivalent forces and moments applied at the independent grids. However, it is difficult to determine the proper mix of these (i.e., select the user weight factors), and therefore it is not recommended to include rotational dof at independent grids. If any rotational dofs are included on the independent grids, then the weight factors for those rotational dofs are scaled by the square of the average of the distances from the reference grid to the independent grids. This has the property of making the weight factors independent of the units used for the model.

If the reference grid is not connected to any other element besides the RBE3, then the RBE3 only redistributes loads and masses, and adds no stiffness to the model.

2.8.2 RSPLINE element

The **RSPLINE** element is an interpolation element that uses the equations of an elastic beam to generate constraint equations.

The RSPLINE creates constraint equations of the following form:

$$u_d = [A]u_i \tag{2-59}$$

where

u_d = the dependent degrees of freedom
u_i = the independent degrees of freedom
A = the matrix of weighting coefficients

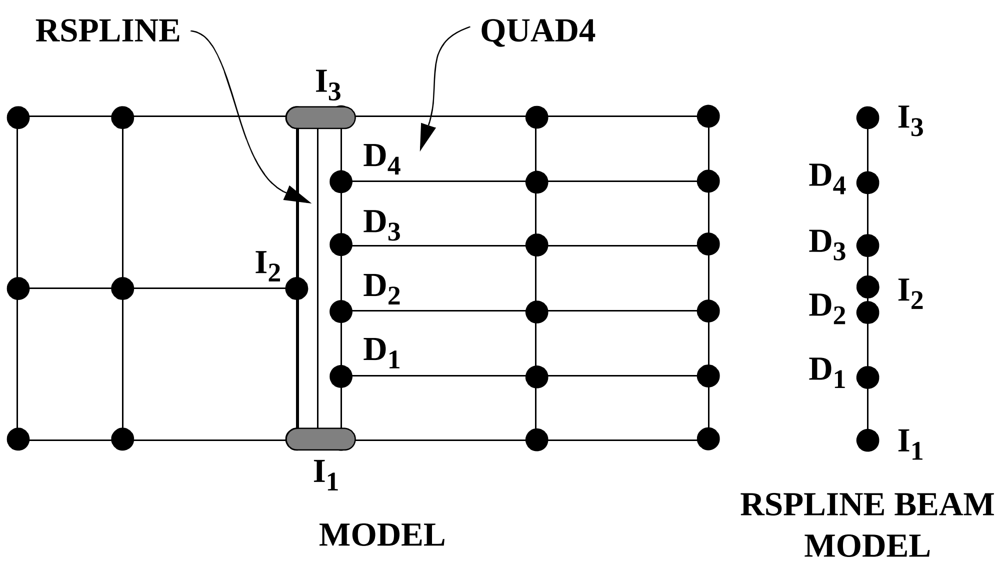
The coefficients in the matrix A are calculated using the interpolation functions of an imaginary elastic beam passing through the grid points. A typical use of the RSPLINE element is to change the mesh density in a plate/shell model.

2

Consider the following example: an RSPLINE is used to connect the degrees of freedom at grids I_j and D_k along the line I_1 - I_3. Without the RSPLINE, the model is incorrect, because the displacements across I_1 - I_3 are not conforming.

With the RSPLINE, the displacements at D_k (the dependent grids) are interpolated using the values of the displacements at I_j (the independent grids). The interpolation provides the same values at the dependent dof as an auxiliary model consisting of beam elements connecting the grids and using enforced displacements at the independent grids.

The RSPLINE forces the displacements to be conforming across the line I_1 - I_3.



Note that the RSPLINE does add stiffness to the model to enforce its interpolations.

If the grids in the RSPLINE do not lie along a straight line, then the extensional/torsional dof will be coupled with the bending dof in the interpolation equations. The field D/L on the RSPLINE bulk data entry controls the strength of this coupling.

2.8.3 BOLT Element

The **BOLT** element is an interpolation element designed to connect separated top and bottom portions of a bolt model with a bolt control grid. When the bolt control grid is constrained with SPC, then the bolt can carry arbitrary loading. When the bolt control grid is left free, whatever externally applied loading acting on the control grid defines the total load in the bolt (at the top/bottom split). A loading normal to the top/bottom split directed from the bottom to the top will apply a tension to the bolt. Any non-zero enforced displacement applied to the bolt control grid will effectively apply a pre-strain loading to the bolt model.

The BOLT element identifies the bolt control grid and lists top/bottom pairs of grids. For the bolt element to have the intended effect, each bolt control grid should not be connected to any other element or MPC. In addition, each top/bottom grid pair should be coincident. Nothing other than the BOLT element should connect the top grids to the bottom grids. It is recommended that each bolt control grid uses an output coordinate system which has one of the axes normal to the top/bottom split, and directed from bottom to top.

The BOLT creates constraint equations of the following form:

$$
\begin{aligned}
u_{bx} - u_{tx} - u_{cx} &= 0 \\
u_{by} - u_{ty} - u_{cy} &= 0 \\
u_{bz} - u_{tz} - u_{cz} &= 0
\end{aligned}
\qquad (2\text{-}60)
$$

where

u_b = the degrees of freedom of a bottom grid

u_t = the degrees of freedom of the corresponding top grid

u_c = the degrees of freedom of the control grid

The degrees of freedom of the bottom grids are set as dependent.

The **BOLTSUB** solution control command facilitates nonlinear contact analysis using bolt preloading. The procedure to use bolt preloading is as follows: First, define a static loadcase with the bolt control grids left free, and with the only applied loadings being forces on bolt control grids. These forces define the bolt preloadings. Define a second static loadcase that uses the desired external loading, and add the BOLTSUB command to refer to the first loadcase. The effect of the BOLTSUB command is to take the calculated displacements of the bolt control grids in the referenced loadcase and apply those as enforced displacements on the bolt control grids in the loadcase that contains the BOLTSUB command. This will effectively pre-strain the bolt model to the desired loading level set by the first loadcase. Finally, the calculated **SPCFORCE** of each bolt control grid reveals the total load acting in the bolt (at the top/bottom split).

2.9 Fluid Elements

GENESIS can perform frequency response analysis on a coupled structure-fluid system. In this case, the structure is modeled as previously described. The fluid is modeled with solid elements (**CHEXA**, **CHEX20**, **CPENTA**, **CTETRA**,**CPYRA**) that refer to a **PSOLID** with the word "PFLUID" in the FCTN field. The fluid material properties are defined on a **MAT10** entry. **GRID** entries referenced by fluid elements must have -1 in the CD field.

The figure below defines the ordering of the grid numbers associated with the six sided, eight node hexahedron element. On the CHEXA data, the grid point identifiers must be specified in this order.

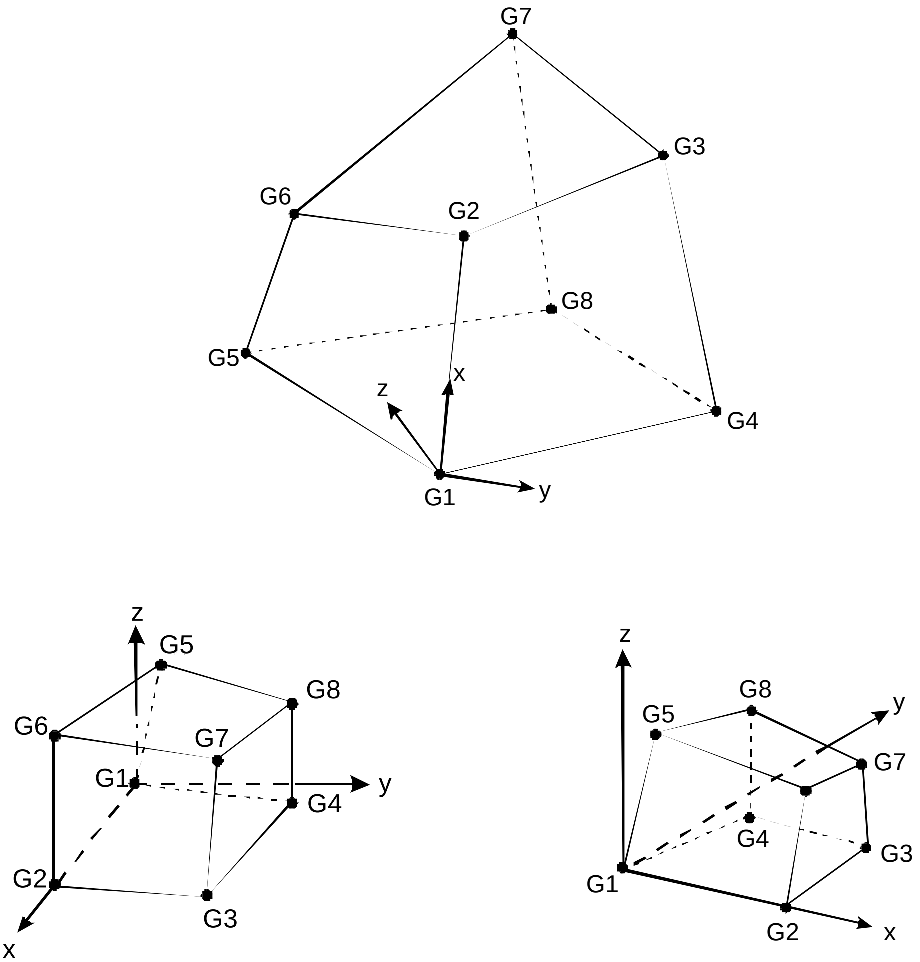


Figure 2-29

2

The figure below defines the ordering of the grid numbers associated with the six sided hexahedron element. This element may have from eight to twenty one nodes. On the CHEXA data, the grid point identifiers must be specified in this order. Grids 1 through 8 must be defined. If any of the remaining node identifiers are omitted, the equations for the element are adjusted accordingly.

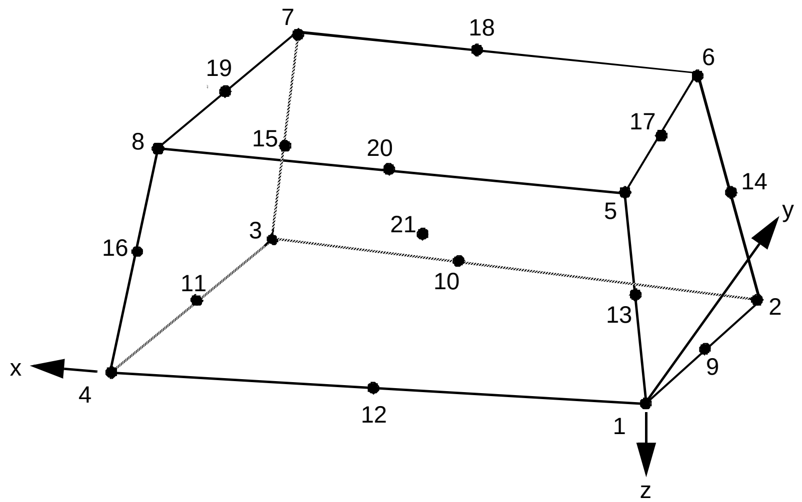


Figure 2-30

The figure below defines the ordering of the grid numbers associated with the five sided, six and fifteen node pentahedral elements. On the CPENTA data, the grid point identifiers must be specified in this order. Additionally, if any of the mid-side nodes are specified, all the mid-side nodes must be specified.

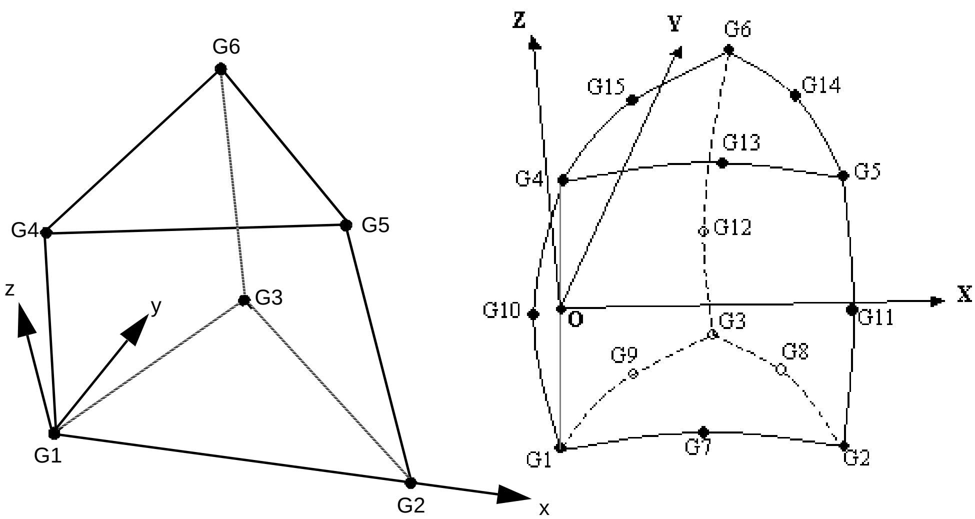


Figure 2-31

The figure below defines the ordering of the grid numbers associated with the four sided, four or ten node tetrahedron element.

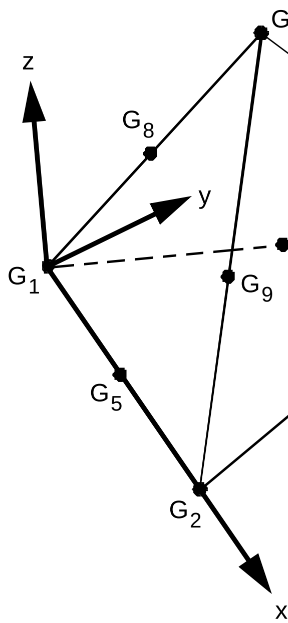


Figure 2-32

The figure below defines the ordering of the grid numbers associated with the five sided, five and thirteen node pyramid elements. On the CPYRA data, the grid point identifiers must be specified in this order. Additionally, if any of the mid-side nodes are specified, all the mid-side nodes must be specified.

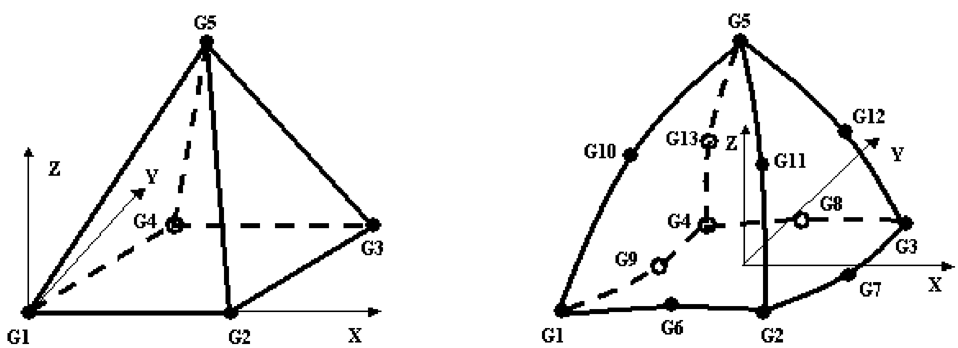


Figure 2-33

2.10 Structural Loads

There are five types of load sets used by *GENESIS* in structural analysis. Concentrated loads at grid points (forces and moments), pressure, and distributed tractions and moments and enforced displacements comprise the first type of loads (external loads). These loads are activated by the **LOAD**=SID command in the solution control section of the input data for static analysis and with **RLOAD1** or **RLOAD2** data statements in the bulk data section for dynamic analysis. The second type of load is the gravity load and is activated by the **GRAVITY**=SID command in the solution control section of the input data for static analysis. For dynamic analysis, RLOAD1 or RLOAD2 data statements in the bulk data section specify the gravity loads. The third type of load is the thermal load, used only in static analysis, and is activated by the **TEMPERATURE**=SID command in the solution control section of the input data. The temperature load can be the solution of a heat transfer analysis. The fourth type of load available results from initial deformation on axial elements (CROD, CBAR and CBEAM), deform loads are be activated by the **DEFORM**=SID command in the solution control section of the input data. Deform loads can only be used in static load cases. Finally, the fifth type is the centrifugal load which can be activated with **CENTRIFUGAL**=CID in the solution control section of the input data. Centrifugal loads can only be used in static load cases.

Concentrated Loads

Concentrated forces (**FORCE**) and moments (**MOMENT**) can be applied to the grid points by specifying a magnitude and direction vector. The direction vector can be specified in any coordinate system. Alternatively, the direction can be specified as the direction from one grid point to another using the **FORCE1** and **MOMENT1** input data.

Distributed Loads

Linearly varying distributed tractions and moments can be applied to CBAR or CBEAM elements in the basic or element coordinate system using the **PLOAD1** input data or in any local coordinate system using the **PLOADA** input data. PLOAD1 can also be used to apply constant loads at an interior point of the beam or to apply a partial pressure load. Uniform pressure loads over the entire surface can be applied to plate/shell and shear panel elements normal to the surface using the **PLOAD2** input data or in a direction specified in the basic or a local coordinate system using the **PLOAD4** input data. Linearly varying pressure loads can be applied to any side in any direction on the r-z plane of axisymmetric elements using the **PLOADX1** input data. Pressure loads can be applied to solid elements normal to any face or in a direction specified by a vector in the basic or any local coordinate system using the **PLOAD4** input data.

Load Combinations

Concentrated and distributed load sets may be linearly combined to create a new load set using the **LOAD** bulk data statement. For a more general way to create load combinations, the user may use the **LOADCOM** and **LOADSEQ** solution control command.

Gravity Loads

Gravity loads (**GRAV**) can be applied to the structure by specifying a magnitude and direction vector. The direction vector can be in the basic or any local rectangular coordinate system.

Thermal Loads

Temperatures can be specified at the grid points using the **TEMP** data statement. In order to decrease input data preparation, default temperatures can be assigned to all grid points using the **TEMPD** data statement. Thermal loads can also be defined as the solution of a heat transfer analysis. Element temperatures are calculated by a simple averaging of grid point temperatures, except for the CHEXA, CPENTA, CTETRA and CPYRA elements, where the element nodal temperatures are used. Thermal loads are calculated using the element reference temperature and coefficient of thermal expansion from the material data. Note that if a load case combination (LOADCOM) is used then no more than one of the referenced load cases may contain a thermal load.

Deform Loads

Non-elastic initial deformations (force fit) can be specified for selected CROD, CBAR and CBEAM elements using the **DEFORM** data statement.

Centrifugal Loads

Centrifugal loads can be applied to the structure by specifying a rotation vector and magnitudes for the angular velocity and acceleration using the **RFORCE** data statement. The direction vector can be in the basic or any local rectangular coordinate system.

Frequency Dependent Loads

Cyclic loads with their spatial distributions being a function of the loading frequency can be selected with the **DLOAD**=SID command of the solution control. Three bulk data statements can be used to specify the required data: **RLOAD1**, **RLOAD2** or **RLOAD3**. The spatial distribution for the three RLOADs are defined as;

$$p(f) = A[C(f) + iD(f)]e^{i\langle\theta - 2\pi f\tau\rangle} \text{ for RLOAD1} \tag{2-61}$$

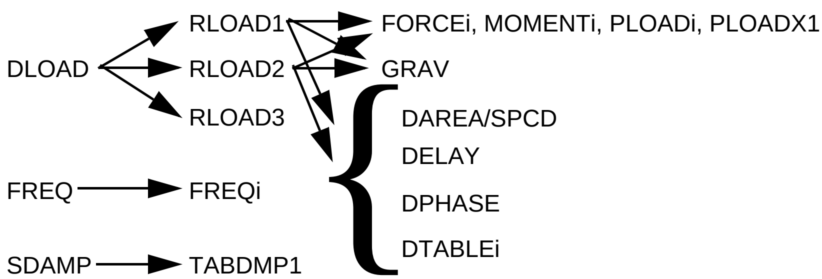
$$p(f) = AB(f)e^{i\langle \phi(f) + \theta - 2\pi f\tau \rangle} \text{ for RLOAD2} \tag{2-62}$$

$$p(f) = A \text{ for RLOAD3} \tag{2-63}$$

where A is defined either with **DAREA** data or through a load set ID (concentrated load, pressure load or gravity load), C(f), D(f), B(f) and $\phi(f)$ are defined on TABLEDi entries, θ is defined on a **DPHASE** entry, τ is defined on a **DELAY** entry and f is the loading frequency. The coefficients that specify A, τ, and θ can be different for each degree of freedom. Finally, the loads are applied in the output (general) coordinate system.

2

A single solution control command, DLOAD can reference any combination of RLOAD1, RLOAD2 and RLOAD3 data. Multiple RLOADs of any type are also allowed. Combinations of loads generated by DAREA, point, pressure and gravity loads are also allowed. The data reference tree for dynamic loads is;



The solution control command **DLOAD** can also reference the bulk data command **DLOAD**. The bulk data command DLOAD itself can reference any combination of RLOAD1, RLOAD2 and RLOAD3 data. The equation used to combine the RLOAD data is:

$$F = S\sum_{i} S_i F_{SIDi}$$

2.11 System Inertia

GENESIS calculates basic inertia properties (i.e., the location of the center of gravity and the second moments of inertia) of the finite element model if the analysis parameter **GRDPNT** is greater than -1 and there are structural (i.e., non-heat transfer) loadcases. Inertia property calculations do not include any fluid elements. Boundary conditions are not considered for these system inertia calculations. Six rigid mode displacement vectors are constructed using the six degrees of freedom at the grid specified by GRDPNT (or the origin of the basic coordinate system if GRDPNT is zero). These rigid mode vectors are used to reduce the global mass matrix to a 6x6 rigid body mass matrix:

$$[M_r] = [G]^T[M][G] \tag{2-64}$$

where

[M] = the global mass matrix

[G] = the 6 rigid mode vectors

Note that if the non-consistent (lumped) global mass matrix is selected (with the **COUPMASS** analysis parameter) this rigid body mass matrix can contain significant error if the finite element mesh is not sufficiently refined.

The rigid body mass matrix is partitioned into 3x3 matrices as follows:

$$[M_r] = \begin{bmatrix} M_{tt} & M_{tr} \\ M_{rt} & I_{gp} \end{bmatrix} \tag{2-65}$$

where

M_{tt} = the 3x3 matrix of scalar masses.

$M_{tr} = M_{rt}^T$ = the 3x3 matrix of first mass moments with respect to the GRDPNT grid.

I_{gp} = the 3x3 matrix of second moments of inertia wrt the GRDPNT grid.

$$[I_{gp}] = \begin{bmatrix} I_{11} & I_{12} & I_{13} \\ & I_{22} & I_{23} \\ SYM & & I_{33} \end{bmatrix} \tag{2-66}$$

where

$$I_{11} = \int \rho[(y - y_{gp})^2 + (z - z_{gp})^2]dV$$

$$I_{12} = -\int \rho[(x - x_{gp})(y - y_{gp})]dV$$

$$I_{13} = -\int\rho[(x-x_{gp})(z-z_{gp})]dV$$

$$I_{22} = \int\rho[(x-x_{gp})^2 + (z-z_{gp})^2]dV$$

$$I_{23} = -\int\rho[(y-y_{gp})(z-z_{gp})]dV$$

$$I_{33} = \int\rho[(x-x_{gp})^2 + (y-y_{gp})^2]dV$$

and

ρ = mass density

x_{gp}, y_{gp}, z_{gp} = coordinates of the GRDPNT grid

2

M_{tt} identifies three scalar masses (one for each basic coordinate system axis). These scalar masses should be the same unless special modeling techniques (e.g., CMASS1) are used.

$$M_x = M_{tt}(1, 1) \tag{2-67}$$

$$M_y = M_{tt}(2, 2) \tag{2-68}$$

$$M_z = M_{tt}(3, 3) \tag{2-69}$$

The location of the center of gravity of the model is determined from M_{tt} and M_{tr}. For each basic coordinate system axis, the center of gravity is determined in the basic system relative to the GRDPNT grid as follows:

$$X_x = \frac{M_{tr}(1, 1)}{M_x} \qquad Y_x = \frac{-M_{tr}(1, 3)}{M_x} \qquad Z_x = \frac{M_{tr}(1, 2)}{M_x} \tag{2-70}$$

$$X_y = \frac{M_{tr}(2, 3)}{M_y} \qquad Y_y = \frac{M_{tr}(2, 2)}{M_y} \qquad Z_y = \frac{-M_{tr}(2, 1)}{M_y} \tag{2-71}$$

$$X_z = \frac{-M_{tr}(3, 2)}{M_z} \qquad Y_z = \frac{M_{tr}(3, 1)}{M_z} \qquad Z_z = \frac{M_{tr}(3, 3)}{M_z} \tag{2-72}$$

I_{gp} contains elements of the inertia tensor. (E.g., the off-diagonal terms should be negated for use in a **CONM2**). The matrix of second moments of inertia with respect to the center of gravity, I_{cg}, is constructed using the parallel axes theorem. For I_{cg}, the off diagonal terms are the negative of the elements of the inertia tensor.

$$I_{cg}(1, 1) = I_{gp}(1, 1) - M_y Z_y^2 - M_z Y_z^2 \tag{2-73}$$

$$I_{cg}(2, 1) = -I_{gp}(2, 1) - M_z X_z Y_z \tag{2-74}$$

$$I_{cg}(3, 1) = -I_{gp}(3, 1) - M_y X_y Z_y \tag{2-75}$$

$$I_{cg}(2, 2) = I_{gp}(2, 2) - M_z X_z^2 - M_x Z_x^2 \tag{2-76}$$

$$I_{cg}(3, 2) = -I_{gp}(3, 2) - M_x Y_x Z_x \tag{2-77}$$

$$I_{cg}(3, 3) = I_{gp}(3, 3) - M_x Y_x^2 - M_y X_y^2 \tag{2-78}$$

Note: I_{cg} is symmetric.

Principal second moments of inertia with respect to the GRDPNT grid and with respect to the center of gravity are obtained by solving for the eigenvalues of the corresponding inertia tensors. Principal directions are printed in the basic coordinate system.

2.12 Static Analysis Calculation Control

In static analysis the individual finite element stiffness matrices and load vectors are combined to form a system stiffness matrix and system load vector. Point forces and moments are then added to the system load vector. Since there are six degrees of freedom for each grid the total number of degrees of freedom is six times the number of grid points.

Constrained degrees of freedom, specified by SPC data, are removed from the system stiffness matrix and load vector. MPC's and rigid elements constraint equations are added to the system stiffness matrix and load vector. In *GENESIS*, the rotational degrees of freedom for grids that are only connected to solid and/or rod elements and not referenced by MPC data are automatically removed from the system stiffness matrix and load vector.

At this point the system matrix should not be singular and the grid point displacements can be found by solving the linear equation:

$$[K]\{u\} = \{F\} \tag{2-79}$$

where [K] is the system stiffness matrix, {F} is the system load vector, and {u} is the displacement vector to be calculated.

The system stiffness matrix may still be singular if the user did not use SPC data to constrain all degrees of freedom that do not have stiffness associated with them. These degrees of freedom can be constrained automatically using the analysis parameter **AUTOSPC**. The input data command to use this feature is:

```
PARAM,AUTOSPC,YES
```

The analysis parameter **PRGPST** controls whether or not to print a table listing all degrees of freedom constrained by AUTOSPC. If the AUTOSPC option is used then degrees of freedom with stiffness less than **EPZERO** multiplied by the average value of the diagonal elements are automatically constrained. The default value of EPZERO is 1.0E-8. The value of EPZERO can be changed with PARAM data. For example, to change the value of EPZERO to 1.0E-6 use the input data command:

```
PARAM,EPZERO,1.0E-6
```

To solve for the displacements, the system stiffness matrix must be triangularized. The stiffness matrix is checked for singularities by comparing the ratio of the diagonal elements before and after triangularization. If the ratio is greater than **MAXRATIO** (default value = 1.0E7) then the matrix is considered singular. The value of MAXRATIO can be changed with PARAM data. For example, to change the value of MAXRATIO to 1.0E12 use the input data command:

```
PARAM,MAXRATIO,1.0E12
```

If the analysis parameter **BAILOUT** is set to NO (the default), then the matrix triangularization process does not stop when the ratio is greater than MAXRATIO. If the analysis parameter BAILOUT is set to YES with the input data command:

```
PARAM,BAILOUT,YES
```

then the triangularization process stops when the ratio is greater than MAXRATIO.

Performance Issues

When the individual finite element matrices and load vectors are formed, they are stored in disk files. When the system stiffness matrix and load vector are assembled, the individual finite element matrices and load vectors are read from the disk files.

On some computer systems the large amount disk file I/O for the individual finite element stiffness matrices can lead to long run times. In this case it is better to calculate the individual finite element matrices as they are needed when the system stiffness matrix is being assembled. Whether the individual finite element stiffness matrices and load vectors are calculated once and stored in disk files or calculated as they are needed during the assembly of system matrix is controlled by the analysis parameter **EOF**. If EOF is NO then the individual finite element stiffness matrices and load vectors are calculated once and stored in disk files. If EOF is YES then the individual finite element stiffness matrices and load vectors are calculated as they are needed during the assembly of system matrix. The default value of EOF is installation dependent. Note that EOF can only be set to YES when the analysis parameter SOLVER is 1 (see below). In general EOF should be YES on computers that have CPU's that are fast relative to the disk I/O speed. To set EOF to YES use the input data command:

```
PARAM,EOF,YES
```

There are two solvers available in *GENESIS* used to triangularize the system stiffness matrix. One is a block solver and the other is a sparse matrix solver. In general the sparse matrix solver is much faster and uses less disk space. However, it may require a large amount of computer core memory. The block solver can solve very large problems with a small amount of computer core memory. The default is to use the sparse matrix solver. The choice of solver is set using the analysis parameter **SOLVER**. If SOLVER is 1 then the sparse matrix solver is used. If solver is set to 2 with the input data command:

```
PARAM,SOLVER,2
```

then the block solver is used. Note that at some installations only the block solver is available.

No matter which solver is used, *GENESIS* will run large problems much faster when more computer core memory is used. The amount of computer core memory used by *GENESIS* is determined by the Executive Control command **LENVEC**. LENVEC determines the number of integer words used for internal storage. Since on most systems an integer uses four bytes of memory, four times LENVEC is the number of bytes used by *GENESIS* for internal storage.

To run very large problems with the sparse matrix solver the amount of internal storage needed by *GENESIS* may be larger than the amount of computer core memory. The value of LENVEC can be set to use more memory than the computer core has on machines that use virtual memory. In this case the limitation on the amount of internal storage used by *GENESIS* is the amount of virtual memory on your system.

GENESIS will run problems faster when multiple cores are used in parallel. The number of threads used by *GENESIS* is determined by the Executive Control command **THREADS**.

2.12.1 Inertia Relief

Normal static analysis requires that a body be in equilibrium. Usually equilibrium is maintained by requiring the reaction forces at the supports be such that they exactly balance the applied loading. Of course, this requires that the body be constrained. However, it is sometimes desirable to analyze the effects of loading on a free body.

An unconstrained body with applied loads will, in general, not be in equilibrium, so a special analysis technique is needed. If transient dynamic effects are ignored, then the result of the applied loading will be elastic deformation superimposed on top of rigid body acceleration. This rigid body acceleration can be used to create inertial “loads”. These inertial loads exactly balance the applied loads, putting the body is in a state of quasi-equilibrium.

The governing equations for a finite element model can be written as:

$$[M]\{\ddot{u}\} + [K]\{u\} = \{F\} \tag{2-80}$$

If the structure is unconstrained (or insufficiently constrained), then the stiffness matrix, [K] will be singular. There are two methods available to remove the singularities: automatic and manual. In the automatic method, the system is augmented with 6 constraint equations as follows:

$$\begin{bmatrix} K & MR_a \\ R_a^T M & 0 \end{bmatrix} \begin{Bmatrix} u \\ \ddot{u}_a \end{Bmatrix} = \begin{Bmatrix} F \\ 0 \end{Bmatrix} \tag{2-81}$$

where the columns of R_a are the 6 rigid body modes defined by the 6 degrees of freedom at the grid defined by the **GRDPNT** parameter (or the origin of the basic coordinate system if GRDPNT is zero or not defined) and $\ddot{u}_a$ are the rigid body accelerations.

In the manual method, a set of degrees of freedom, $\{u_r\}$, (called the support set) just sufficient to define the rigid body modes of the structure must be specified, and the rigid motion displacement vector is expressed as a function of those dof:

$$\{u\} = [R]\{u_r\} \tag{2-82}$$

The governing equations are reduced to the support (rigid body) dof as follows:

$$[M_{rr}]\{\ddot{u}_r\} + [K_{rr}]\{u_r\} = \{F_r\} \tag{2-83}$$

where

$$[M_{rr}] = [R]^T[M][R] \tag{2-84}$$

$$[K_{rr}] = [R]^T[K][R] \tag{2-85}$$

$$\{F_r\} = [R]^T\{F\} \tag{2-86}$$

2

Now, $[K_{rr}]$ = [zero] because [R] consists of rigid body (zero strain energy) modes. Therefore, the rigid body accelerations can be determined:

$$\{\ddot{u}_r\} = [M_{rr}]^{-1}\{F_r\} \tag{2-87}$$

These are put back in the original governing equations and the support set dof are constrained to zero to obtain the following quasi-equilibrium problem:

$$[K]\{u\} = F - [M][R]\{\ddot{u}_r\} = \{F_{effective}\} \tag{2-88}$$

Inertia relief is requested in *GENESIS* with the **SUPORT** solution control command or with PARAM, **INREL**,-2.

The automatic method is selected with the AUTO keyword:

```
SUPORT = AUTO
```

The manual method is selected by referencing a support set id defined by **SUPORT1** bulk data statements:

```
SUPORT = sid
```

PARAM,INREL,-2 may be used to set the default to be SUPORT=AUTO for all static loadcases.

In the manual method, *GENESIS* calculates the rigid modes, [R], based on a factorization of the stiffness matrix with the support dof constrained to zero. Therefore, all of the support dof must be elastically connected to the structure (e.g., rotational dof of a grid connected only to solid elements may not be included in the support set). *GENESIS* checks the validity of the rigid modes by comparing the magnitude of the diagonal elements of $[K_{rr}]$ to the values in the original stiffness matrix. If this ratio is too high, it indicates a potential problem with the chosen support set. The tolerance for this check can be changed with the analysis parameter **IRTOL**. For example:

```
PARAM,IRTOL,1.0E-5
```

The residuals for inertia relief loadcases should also be checked carefully. If the support set contains insufficient dof, erroneous results may be generated due to numerical round-off in the solution process. In this case, the residual will be large.

2.13 Frequency Calculation Control

There are presently four methods available for frequency calculation in *GENESIS*; the subspace iteration method, the Lanczos method, the SMS approximation method and the Guyan reduction method. *GENESIS* also has the ability to load an external shared object (DLL) implementing a user-supplied eigenvalue solution method. Mode shapes can be normalized using four types of norms: MAX, MAX0, MASS and POINT norms. Frequency calculations are controlled by the **EIGR** or **EIGRL** input data. The solution to the eigenvalue problem is activated with **METHOD** = SID in a load case in the solution control. The mass matrix can be chosen as a consistent mass matrix or as a diagonal (lumped) mass matrix by setting the analysis parameter, **COUPMASS**, using the PARAM data statement (the default is to use the consistent mass matrix). If weight density is used in the material data, it must be converted to mass density by dividing by the acceleration of gravity (g). This can be done automatically using the analysis parameter **WTMASS**. All material densities are multiplied by WTMASS (Default value = 1.0). If weight density is used, then WTMASS should be set to 1/g. For example, if inches are used for the length dimension, then g=386 in/sec^2 and WTMASS is set to 0.00259 using the input command:

```
PARAM, WTMASS, 0.00259
```

Note that if the WTMASS parameter is not 1.0, it is assumed that <u>all</u> mass-related values (including densities, non-structural masses and CONM2 masses) are in weight units and are scaled by WTMASS.

Subspace Iteration Method

Subspace iteration can be used on any size of problem. The lowest ND (default =1) frequencies, including rigid body modes, will be calculated, where ND is specified on the EIGR entry in the input data. Note that the stiffness matrix must be positive semidefinite (i.e., no negative frequencies) if this method is used.

The analysis parameter **EPSEIG** is the convergence factor for all of the eigenvalues found by subspace iteration. If the maximum relative change in any eigenvalue is less than EPSEIG from one iteration to the next, then the process stops. The default value for EPSEIG is 1.0E-6. The maximum number of iterations is determined by the analysis parameter **ITMXSS**. The default is 50. This can be changed, say to 100, using the input data command:

```
PARAM, ITMXSS, 100
```

Lanczos Method

The Lanczos method can be used on any size problem. The calculated frequencies will be searched according to the following table:

V1	V2	ND	Modes Calculated
V1	V2	ND	At most ND modes between V1 and V2
V1	V2	Blank	All modes between V1 and V2
V1	Blank	ND	First ND modes greater than or equal to V1
V1	Blank	Blank	First one mode greater than or equal to V1
Blank	V2	ND	At most ND modes less than or equal to V2
Blank	V2	Blank	All modes less than or equal to V2
Blank	Blank	ND	First ND modes
Blank	Blank	Blank	First one mode

Where, V1 and V2 are the lower and upper bounds, respectively, on the frequency range of interest. ND is the number of desired frequencies including rigid body modes. V1, V2 and ND are specified on the **EIGR** or **EIGRL** entry.

The Lanczos method can only be used in conjunction with the sparse matrix solver. Therefore, the analysis parameter **SOLVER** should be set to 1. Note that not all installations have the sparse matrix solver.

SMS Approximation Method

The SMS approximation method can be used on any size of problem. The calculated frequencies will be searched according to the following table:

V1	V2	ND	Modes Calculated
V1	V2	ND	At most ND modes between V1 and V2
V1	V2	Blank	All modes between V1 and V2
Blank	V2	ND	At most ND modes less than or equal to V2
Blank	V2	Blank	All modes less than or equal to V2

Where, V1 and V2 are the lower and upper bounds, respectively, on the frequency range of interest. ND is the number of desired frequencies including rigid body modes. V1, V2 and ND are specified on the EIGR entry.

The SMS approximation method builds a reduced approximation of the full finite element model. Therefore, frequencies calculated by this method are only close approximations of the frequencies of the input structure. This method is designed to calculate large numbers of frequencies very quickly, and is especially well suited to use with modal dynamic reduction. For large problems (i.e., 500,000+ dof) where a reasonable upper bound cutoff frequency is known, this method is the fastest available in *GENESIS*.

The SMS approximation method works by internally calculating all frequencies below V2, even if V1 is given. The performance of the method therefore depends on the number of modes below V2, not on the number of modes between V1 and V2.

The SMS method can only be used in conjunction with the sparse matrix solver. Therefore, the analysis PARAMeter SOLVER should be set to 1. Note that not all installations have the sparse matrix solver.

Guyan Reduction Method

The Guyan reduction technique solves an approximate problem using selected degrees of freedom specified by the ASET command. This technique is explained in detail in Section **2.13.1**.

Normalization of Eigenvectors

There are four types of normalization available with the EIGR data statement. These are MAX, MAX0, MASS and POINT.

MAX is used to normalize the eigenvector so its maximum component is one at every design cycle.

MAX0 is used to normalize the eigenvector so its maximum component is one at the zeroth design cycle (first analysis). This normalizing component is then used in the rest of the optimization to normalize the eigenvector. This norm has the advantage over the MAX norm of making the eigenvectors continuous functions of the design variables.

MASS is used to normalize the eigenvector so $\Phi^T M \Phi$ is one.

POINT is used to select a particular grid and component (degree of freedom) to normalize the eigenvector.

2.13.1 Guyan Reduction

The eigenvalue problem may be reduced to a user-specified set of degrees of freedom using Guyan reduction. The Guyan reduction solution method is requested with the **ASET** = sid command in the solution control. The A-SET degrees of freedom are specified on **ASET2** and/or **ASET3** bulk data statements. Note that the A-SET degrees of freedom must be mutually exclusive with respect to the dependent degrees of freedom in rigid elements as well as the constrained degrees of freedom (SPC dof and dependent dof on MPC) in the A-SET loadcase.

All of the degrees of freedom of the problem are divided into the ‘a’ set (as specified on ASET2 and ASET3 bulk data statements) and the ‘o’ set (all of the remaining degrees of freedom). The global stiffness matrix, K, is partitioned accordingly.

$$\begin{bmatrix} K_{aa} & K_{ao} \\ K_{oa} & K_{oo} \end{bmatrix} \tag{2-89}$$

A static condensation technique is used to approximate the ‘o’ set dof in terms of the ‘a’ set dof:

$$u = \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = Gu_a = \begin{bmatrix} G_a \\ G_o \end{bmatrix} u_a \tag{2-90}$$

2

where G_a = I (identity) and $G_o = -K_{oo}^{-1}K_{oa}$

Note that this is only an approximation that assumes that the external loads (in the eigenvalue case, inertial loads) on the ‘o’ set dof are negligibly small.

The reduced stiffness, mass and damping matrices are computed as:

$$\bar{K} = G^T K G \tag{2-91}$$

$$\bar{M} = G^T M G \tag{2-92}$$

$$\overline{K_4} = G^T K_4 G \tag{2-93}$$

Note that, in general, the reduced stiffness, mass and damping matrices are full matrices, even if the global stiffness and mass matrices are sparse. Therefore, the memory and disk space requirements for this method grow very rapidly as the number of degrees of freedom in the ‘a’ set is increased.

The reduced eigenvalue problem, $[\bar{K} - \lambda\bar{M}]\phi_a = 0$, is solved using Given’s method, regardless of what is specified on the EIGR statement. Note that the reduced mass matrix must be positive definite for this method. Therefore, massless degrees of freedom may not be included in the ‘a’ set. The lowest ND frequencies (of the reduced problem -- which are only approximations of the frequencies of the full system) are returned. The optional data V1 and V2 may be used to return the lowest ND frequencies in the range V1 to V2.

The reduced stiffness and/or mass and/or damping matrices may be output using the solution control statements:

KAA = `DMIG or POST`

MAA = `DMIG or POST`

K4AA = `DMIG`

Reduced mass matrices may optionally be calculated by an external user program and input into *GENESIS* using the following solution control statement:

```
MAAUSER = YES
```

This will attempt to call the GNMASS routine to calculate the reduced mass matrix. The values of the design variables, the loadcase numbers and the A-SET degrees of freedom list are passed to the user routine GNMASS at the start of each design cycle. This user routine should calculate (or execute an external program that calculates) the reduced mass matrices for all A-SET loadcases that have MAAUSER = YES and return these matrices.

The eigenvector (mode shape) of the non reduced problem, i.e. the eigenvector containing the ‘a’ and ‘o’ degrees of freedom, can be printed using the same command as for regular frequency load cases. In other words, by using the SVECTOR command.

Guyan Reduction with Craig-Bampton Modes

The Guyan reduction is a static condensation, and is only correct when there is no loading on the omitted (non-ASET) degrees of freedom. The quality of the reduced matrices can be improved by adding Craig-Bampton basis vectors to partially account for inertial loading on the omitted degrees of freedom. In this case, the reduction basis matrix, G, is augmented with additional vectors known a fixed-interface modes, C, as in Eq. **(2-94)**. These additional vectors are the mass-normalized eigenvectors of the system with all ASET degrees of freedom (the interface) constrained.

$$G_{CB} = \begin{bmatrix} G & C \end{bmatrix} \tag{2-94}$$

Each fixed-interface mode is associated to a generalized degree of freedom. The collection of generalized degrees of freedom, u_q is then concatenated to the ASET degrees of freedom to make the new set of reduced degrees of freedom:

$$u = \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = \begin{bmatrix} G & C \end{bmatrix} \begin{Bmatrix} u_a \\ u_q \end{Bmatrix} = \begin{bmatrix} G_a & C_a \\ G_o & C_o \end{bmatrix} \begin{Bmatrix} u_a \\ u_q \end{Bmatrix} \tag{2-95}$$

Note that $C_a = 0$ (because C is calculated with the ASET dofs fixed).

Now, the reduced stiffness and mass matrices are computed as:

$$\bar{K}_{CB} = G_{CB}^T K G_{CB} = \begin{bmatrix} \bar{K} & 0 \\ 0 & \lambda_{CB} \end{bmatrix} \tag{2-96}$$

$$\bar{M}_{CB} = G_{CB}^T M G_{CB} = \begin{bmatrix} \bar{M} & G^T M C \\ C^T M G & I \end{bmatrix} \tag{2-97}$$

Where λ_{CB} is a diagonal matrix of the fixed-interface eigenvalues.

To add Craig-Bampton modes in a Guyan reduction loadcase, two extra solution control commands must be added. **CBMETHOD** points to an **EIGR** or **EIGRL** bulk data entry, and is used to control how the fixed-interface modes are calculated. **QSET** selects a set defined by **QSET2** and/or **QSET3** bulk data entries, and specifies grid/component degrees of freedom that will act as generalized degrees of freedom for the fixed-interface modes. The number of Craig-Bampton vectors used will be the minimum of the number of modes calculated by CBMETHOD and the number of degrees of freedom specified by QSET.

2.13.2 Using User Supplied Mass Matrix in Guyan Reduction Load Cases

Users may replace the *GENESIS* reduced mass matrix with their own reduced mass matrix in any Guyan reduced load cases. The user must write a subroutine that, given the arguments, calculates the mass matrix. This function must be included in a shared object (DLL) that is specified by the **GNMASS** executive control command. The Fortran name of the subroutine is GNMASS, and is declared as follows:

```
SUBROUTINE GNMASS(UDV,IASET,NDVT,NEQR,NEQRL,IUSERL,
     *                   RMASS,IERROR)
INTEGER NDVT, NEQR, NEQRL, IUSERL, IERROR
DOUBLE PRECISION UDV(NDVT),  RMASS(NEQRL)
INTEGER  IASET(2,NEQR)
C
C     INPUT: UDV(NDVT)    - THE DESIGN VARIABLE VALUES IN USER
C                           DEFINED ORDER.
C           IASET(1,NEQR) - GRIDS ASSOCIATED TO ASET DEGREE OF FREEDOM
C                 (2,NEQR) - COMPONENT ASSOCIATED TO ASET DEGREE OF
C                            FREEDOM. OPTIONALLY, THIS ARRAY MAY BE
C                            CHANGED BY THE USER TO CHANGE ORDER OF
C                            GRID AND COMPONENT.
C             NDVT   - THE TOTAL NUMBER OF DESIGN VARIABLES.
C             NEQR   - THE NUMBER OF ASET DEGREE OF FREEDOM.
C             NEQRL  - DIMENSION OF RMASS [=NEQR*(NEQR+1)/2)]
C             IUSERL - LOADCASE NUMBER
C
C     OUTPUT: RMASS(NEQRL) - USER CALCULATED REDUCED MASS  MATRIX
C                            IN PACKED FORM (COLUMNS OF THE LOWER
C                            TRIANGLE)
C
C              IERROR - ERROR FLAG
C                     =0 FOR NO ERROR.
C                     =1 FOR ERROR.
```

In the GNMASS subroutine, the user has to fill in the RMASS array for each Guyan reduction loadcase. The RMASS array has to be filled in by columns considering only the lower triangular part.

For example, if $[M] = \begin{bmatrix} M_{11} & & SYM \\ M_{21} & M_{22} & \\ M_{31} & M_{32} & M_{33} \end{bmatrix}$ then it is stored as

RMASS(1) = M_{11}

RMASS(2) = M_{21}

RMASS(3) = M_{31}

RMASS(4) = M_{22}

RMASS(5) = M_{32}

RMASS(6) = M_{33}

To select the use of the user supplied subroutine, the command **MAAUSER** = YES has to be specified in the Guyan reduction loadcase.

If a language other than Fortran is used to create the shared object, care must be taken to ensure that the interface function has the correct name and arguments. The actual required function names are system dependent. For example, using the C language, the DRESP3 interface function should have the following prototype:

Microsoft Windows	`__declspec( dllexport ) void GNMASS(double udv[], int iaset[][2], int *ndvt, int *neqr, int *neqrl, int *iuserl, double rmass[], int *ierror);`
Solaris, Linux, IRIX, OSF1, HP-UX	`void gnmass_(double udv[], int iaset[][2], int *ndvt, int *neqr, int *neqrl, int *iuserl, double rmass[], int *ierror);`
AIX	`void gnmass(double udv[], int iaset[][2], int *ndvt, int *neqr, int *neqrl, int *iuserl, double rmass[], int *ierror);`

2.13.3 Using External Eigenvalue Solver Methods

Users may supply their own eigenvalue solution methods, for example, to off-load the computation to a remote massively parallel processing machine. The user must supply a shared object (DLL) that implements the desired solution method. The **EIGMETHOD** executive control command is used to load a shared object, and assign it a method identification number (*mthid*). For an external eigenvalue solver to be used for a particular loadcase, *mthid* must be placed in the METHOD field of the activated **EIGR** bulk data entry.

2

An EIGMETHOD shared object must export seven interface functions. In the C language, the functions should have the following prototypes, noting these system dependent defines:

Microsoft Windows	**#define SUBROUTINE __declspec(dllexport) void**
Linux	**#define SUBROUTINE void** **#define EIGCAPS eigcaps_** **#define EIGINIT eiginit_** **#define EIGINPUT eiginput_** **#define EIGSOLVE eigsolve_** **#define EIGPAIRS eigpairs_** **#define EIGMDAMP eigmdamp_** **#define EIGFINIS eigfinis_**

```
SUBROUTINE EIGCAPS (int *mthid, int *icap,
                    int *ires);
SUBROUTINE EIGINIT (int *mthid, int *neq, int *nwant,
               int *ilflag, double *eleft, int *irflag, double *eright,
                    int *nrescv, int *ndamps,
                    ll_type *nctx, ll_type *ictx, int *ierr);
SUBROUTINE EIGINPUT (int *mthid, int *iwhat, int *nrows,
                     int *icol, int irows[], double values[],
                     ll_type *nctx, ll_type *ictx, int *ierr);
SUBROUTINE EIGSOLVE (int *mthid,
                ll_type *nctx, ll_type *ictx, int *nfound, int *ierr);
SUBROUTINE EIGPAIRS (int *mthid, int *nvs, int *levecs, int *mstart,
      int *mend,
                     ll_type *nctx, ll_type *ictx, double evals[],
      double evecs[], int *ierr);
SUBROUTINE EIGMDAMP (int *mthid, int *idamp, int *imod,
               ll_type *nctx, ll_type *ictx, double damp[], int *ierr);
SUBROUTINE EIGFINIS (int *mthid,
                     ll_type *nctx, ll_type *ictx, int *ierr);
```

The type **ll_type** is system dependent, and is an integral type at least as large as a **(void *)**. On most systems, the following should be used:

```
#define ll_type long long
```

GENESIS will call `EIGCAPS` several times after loading the shared object to query the special capabilities supported by the external library. Note that if an external library chooses to support damping matrix reduction, then it must also claim to support residual vector calculation for *GENESIS* to be able to take advantage of external damping reduction.

Each time *GENESIS* needs to solve an eigensystem (which could be many times during a program run), it will call `EIGINIT` once to initialize the solution process. It will then call `EIGINPUT` many times to pass the stiffness and mass matrices and possibly load vectors and damping matrices into the external library, one column at a time. It will then call `EIGSOLVE` once to indicate that all matrix data has been input and that the external library should solve the system. It will then call `EIGPAIRS` possibly several times to retrieve the calculated eigenvalues and eigenvectors in blocks of mode number ranges. If any damping matrix data was passed to `EIGINPUT`, then the program will next call `EIGMDAMP` many times to retrieve the reduced damping matrix or matrices, one column at a time. Finally, the program will call `EIGFINIS` to finalize the solution process, giving the external library a chance to free up any resources that it may have allocated during the solution.

Detailed descriptions of each function's parameters follows:

```
C*********************************************************************
C     SUBROUTINE NAME: EIGCAPS
C
C     PURPOSE: GET CAPABILITIES OF EXTERNAL EIGENVALUE SOLVER
C
C     INPUT: MTHID - EIGMETHOD ID
C            ICAP - THE CAPABILITY TO BE INQUIRED
C                   = 1 : ABLE TO CALCULATE RESIDUAL VECTORS
C                   = 2 : ABLE TO CALCULATE MODAL DAMPING MATRICES
C
C     OUTPUT:
C            IRES   - THE RESULT OF THE INQUIRY
C                    =-1 FOR NO USER SUPPLIED ROUTINE.
C                    = 0 : DOES NOT HAVE CAPABILITY
C                    > 0 : HAS CAPABILITY
C

C*********************************************************************
C     SUBROUTINE NAME: EIGINIT
C
C     PURPOSE: INITIALIZE AN EXTERNAL EIGENVALUE SOLVER CONTEXT
C
C     INPUT: MTHID - EIGMETHOD ID
C            NEQ    - THE DIMENSION OF THE SYSTEM (NUMBER OF DOF)
C          NWANT  - THE DESIRED NUMBER OF EIGENPAIRS (0 = all in range)
C            ILFLAG - WHETHER THERE IS A LEFT END TO THE RANGE
C            ELEFT  - LEFT END OF RANGE
C            IRFLAG - WHETHER THERE IS A RIGHT END TO THE RANGE
C            ERIGHT - RIGHT END OF RANGE
C           NRESCV - NUMBER OF POTENTIAL RESIDUAL VECTORS (NUMBER OF
     LOADS)
C            NDAMPS - NUMBER OF DAMPING MATRICES TO REDUCE [0,2]
C            NCTX   - SIZE OF CONTEXT MEMORY (IN 8-BYTE LONGS)
C
C     OUTPUT:
C            ICTX(NCTX) - EIGMETHOD CONTEXT MEMORY
C            IERR   - THE ERROR SWITCH.
C                    =-1 FOR NO USER SUPPLIED ROUTINE.
C                    =0 FOR NO ERROR.
C                    >0 FOR ERROR.
C
C      NOTES:  ILFLAG == 0 implies left end is -infinity
C              IRFLAG == 0 implies right end is +infinity
```

```
C*********************************************************************
C     SUBROUTINE NAME: EIGINPUT
C
C     PURPOSE: INPUT MATRIX DATA INTO AN EXTERNAL EIGENSOLVER
C
C     INPUT: MTHID - EIGMETHOD ID
C            IWHAT   - FLAG FOR WHAT DATA IS BEING INPUT
C                      = 1 : Stiffness
C                      = 2 : Mass
C                      = 3 : Load vector
C                      = 4 : Damp1
C                      = 5 : Damp2
C            NROWS   - THE DIMENSION IF IROWS/VALUES [1,NEQ]
C            ICOL    - THE COLUMN NUMBER [1,NEQ]
C            IROWS   - ARRAY OF ROW NUMBERS
C            VALUES  - ARRAY OF MATRIX VALUES
C            NCTX   - SIZE OF CONTEXT MEMORY (IN 8-BYTE LONGS)
C
C     IN/OUT:
C            ICTX(NCTX) - EIGMETHOD CONTEXT MEMORY
C
C     OUTPUT:
C            IERR   - THE ERROR SWITCH.
C                    =-1 FOR NO USER SUPPLIED ROUTINE.
C                    =0 FOR NO ERROR.
C                    >0 FOR ERROR.
C
C     NOTES:  IROWS is in the Fortran (1-based) convention.
C             This is scratch memory, so can be altered in-place to
C             0-based, if needed.
C
C             Stiffness, Mass and Dampx are assumed to be symmetric
C            matrices.  Only data in the lower triangle will be input
C             (i.e., row >= col).
C             For Load vectors, ICOL will be in range [1,NRESCV], and
C             rows will be in range [1,NEQ].
```

Load vectors will only be input if the `EIGCAPS` routine indicates that the external solver has the capability to calculate residual vectors, and residual vectors are requested by the user. Damping matrices will only be input if the `EIGCAPS` routine indicates that the external solver has the capability to calculate modal reduced damping, and the problem has viscous and/or elemental structural damping matrices.

2

```
C*********************************************************************
C      SUBROUTINE NAME: EIGSOLVE
C
C      PURPOSE: TELL EXTERNAL EIGENSOLVER TO SOLVE
C
C      INPUT: MTHID - EIGMETHOD ID
C             NCTX   - SIZE OF CONTEXT MEMORY (IN 8-BYTE LONGS)
C
C      IN/OUT:
C             ICTX(NCTX) - EIGMETHOD CONTEXT MEMORY
C
C      OUTPUT:
C             NFOUND - THE NUMBER OF EIGENPAIRS ACTUALLY FOUND
C             IERR   - THE ERROR SWITCH.
C                     =-1 FOR NO USER SUPPLIED ROUTINE.
C                     =0 FOR NO ERROR.
C                     >0 FOR ERROR.
C
C      NOTE: NFOUND should include all calculated eigenpairs plus the
C            number of calculated residual vectors (if any).  Any
C            returned residual vector should have a pseudo-eigenvalue
C            greater than ERIGHT (if IRFLAG)

C*********************************************************************
C      SUBROUTINE NAME: EIGPAIRS
C
C      PURPOSE: RETRIEVE EIGENVALUES AND EIGENVECTORS
C
C      INPUT: MTHID - EIGMETHOD ID
C             NVS    - SIZE  OF EVALS and EVECS
C             LEVECS - LEADING DIMENSION OF EVECS
C             MSTART - MODE NUMBER OF FIRST PAIR TO RETRIEVE [1,MEND]
C           MEND   - MODE NUMBER OF LAST PAIR TO RETRIEVE [MSTART,NFOUND]
C             NCTX   - SIZE OF CONTEXT MEMORY (IN 8-BYTE LONGS)
C
C      IN/OUT:
C             ICTX(NCTX) - EIGMETHOD CONTEXT MEMORY
C
C      OUTPUT:
C             EVALS(NVS) - EIGENVALUES
C             EVECS(LEVECS,NVS) - EIGENVECTORS (MASS NORMALIZED)
C             IERR   - THE ERROR SWITCH.
C                     =-1 FOR NO USER SUPPLIED ROUTINE.
C                     =0 FOR NO ERROR.
C                     >0 FOR ERROR.
```

```
C*******************************************************************
C      SUBROUTINE NAME: EIGMDAMP
C
C      PURPOSE: RETRIEVE PORTION OF MODAL REDUCED DAMPING MATRIX
C
C      INPUT: MTHID - EIGMETHOD ID
C             IDAMP  - WHICH DAMPING MATRIX [1,2]
C             IMOD   - COLUMN NUMBER OF MODAL DAMPING MATRIX [1,NFOUND]
C             NCTX   - SIZE OF CONTEXT MEMORY (IN 8-BYTE LONGS)
C
C      IN/OUT:
C             ICTX(NCTX) - EIGMETHOD CONTEXT MEMORY
C
C      OUTPUT:
C             DAMP(IMOD) - PORTION OF COLUMN IMOD IN UPPER TRIANGLE
C                          OF MODAL DAMPING MATRIX IDAMP
C             IERR   - THE ERROR SWITCH.
C                     =-1 FOR NO USER SUPPLIED ROUTINE.
C                     =0 FOR NO ERROR.
C                     >0 FOR ERROR.
```

If the `EIGCAPS` routine indicates that the external solver does not have the capability to calculate modal reduced damping, then the `EIGMDAMP` function does not need to be defined.

```
C*******************************************************************
C      SUBROUTINE NAME: EIGFINIS
C
C      PURPOSE: FINALIZE AN EXTERNAL EIGENVALUE SOLVER CONTEXT
C
C      INPUT: MTHID - EIGMETHOD ID
C             NCTX   - SIZE OF CONTEXT MEMORY (IN 8-BYTE LONGS)
C             ICTX(NCTX) - EIGMETHOD CONTEXT MEMORY
C
C      OUTPUT:
C             IERR   - THE ERROR SWITCH.
C                     =-1 FOR NO USER SUPPLIED ROUTINE.
C                     =0 FOR NO ERROR.
C                     >0 FOR ERROR.
```

The context memory, ICTX, is passed unaltered to all API routines to allow these routines to share a state. NCTX is guaranteed to be at least 1, so that ICTX will be at least large enough to hold a (void *). In most cases, NCTX will be substantially larger.

Any returned non-zero error code will be reported to the user, and all calculations will be aborted. It is recommended to use a different error code value for every checked error condition to facilitate locating the source of errors.

2.14 Superelement Reduction

The stiffness, mass and loading may be reduced to a user-specified set of degrees of freedom using static condensation. This technique can be useful for reducing the analysis time of a design problem by grouping the non-designed portion into a superelement, and reducing this superelement to its interface degrees of freedom. Superelement reduction is requested with the **REDUCE** executive control command and the **BOUNDARY** = sid solution control command. The boundary (interface) degrees of freedom are specified on **ASET2** and/or **ASET3** bulk data statements. Note that the boundary degrees of freedom must be mutually exclusive with respect to the dependent degrees of freedom in rigid elements as well as the constrained degrees of freedom (SPC dof and dependent dof on MPC).

All of the degrees of freedom of the problem are divided into the ‘a’ set (as specified on ASET2 and ASET3 bulk data statements) and the ‘o’ set (all of the remaining degrees of freedom). The global stiffness matrix, K, is partitioned accordingly.

$$\begin{bmatrix} K_{aa} & K_{ao} \\ K_{oa} & K_{oo} \end{bmatrix} \qquad (2\text{-}98)$$

A static condensation technique is used to approximate the ‘o’ set dof in terms of the ‘a’ set dof:

$$u = \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = Gu_a = \begin{bmatrix} G_a \\ G_o \end{bmatrix} u_a \qquad (2\text{-}99)$$

where G_a = I (identity) and $G_o = -K_{oo}^{-1}K_{oa}$

The reduced stiffness, mass and damping matrices are computed as:

$$\bar{K} = G^T K G \qquad (2\text{-}100)$$

$$\bar{M} = G^T M G \qquad (2\text{-}101)$$

$$\overline{K_4} = G^T K_4 G \qquad (2\text{-}102)$$

The reduced load matrix is computed as:

$$\bar{P} = G^T P \qquad (2\text{-}103)$$

The reduced matrices may be output in the **DMIG** bulk data format using the **ALOAD**, **KAA**, **MAA** and **K4AA** solution control commands. The DMIG file may, in turn, be included into a residual model to account for the eliminated superelement.

The analysis parameter **SEMPC** can be used in conjunction with a **DISPLACEMENT** or **SVECTOR** output request to output a portion of the G_o matrix in **MPC** format. This allows the residual problem to calculate results for degrees of freedom omitted by the reduction process.

Note that, in general, the reduced stiffness, mass and damping matrices are full matrices, even if the global stiffness and mass matrices are sparse. Therefore, the memory and disk space requirements for this method can grow very rapidly as the number of degrees of freedom in the ‘a’ set is increased.

2.15 Buckling Analysis

Buckling analysis is used to study the stability of structures subject to static load cases. In buckling analysis, buckling load factors and buckling mode shapes are calculated by solving an eigenvalue problem.

2.15.1 Buckling Elements

Geometric (differential) stiffness matrices are available for the elastic elements: CROD, CBAR, CBEAM, CSHEAR, CTRIA3, CQUAD4, CTRIA6, CQUAD8, CTETRA, CPENTA, CHEXA, CHEX20 and CPYRA; and the rigid elements: RROD, RBAR and RBE2.

The rest of the elements such as CELAS1, CBUSH, CGAP, RSPLINE or GENEL do not have geometric stiffness matrices but they can be used in buckling analysis. They only contribute to the stiffness matrix and will not buckle individually.

2.15.2 Boundary Conditions

The boundary conditions used in a buckling loadcases are not specified in the buckling loadcase itself. They are specified in the static loadcase referenced by the buckling loadcase with the solution control command **STATSUB**.

2.15.3 Buckling Loads

The static loads used in buckling analysis are not specified on the buckling loadcase itself. They are specified in the static loadcase referenced by the buckling loadcase with the solution control command **STATSUB**.

2.15.4 Buckling Analysis Control

Buckling analysis by the finite element method parallels in many aspects natural frequency analysis. Instead of a mass matrix, a geometric (differential) stiffness matrix is used. In buckling analysis, the individual geometric stiffness matrices are assembled to create the global geometric stiffness matrix.

The governing equations for bifurcation buckling analysis using the finite element method can be written as:

$$[K - \lambda Kg]\phi = 0 \tag{2-104}$$

where [K] is the system stiffness matrix and [Kg] is the system geometric stiffness matrix. $\{\phi\}$ and λ are the bucking mode shape and the buckling load factor to be calculated.

There are presently two methods available for buckling calculations in *GENESIS*: subspace iteration and Lanczos. Mode shapes can be output using three types of norms: MAX, STIFF and POINT.

The buckling calculations are controlled by the **EIGR** or **EIGRL** input data. The solution to the eigenvalue problem is activated with the solution control command **METHOD**. The static analysis load case is selected with the solution control command **STATSUB**.

Subspace Iteration

Subspace iteration can be used on any size of problem. The lowest ND number of eigenvalues (in magnitude) will be calculated, where ND is specified on the EIGR input data.

The analysis parameter **EPSEIG** is the convergence factor for all of the eigenvalues found by Subspace Iteration. If the maximum relative change in any eigenvalue is less than EPSEIG from one iteration to the next, then the process stops. The default value for EPSEIG is 1.0E-6. The maximum number of iterations is determined by the analysis parameter **ITMXSS**. The default is 50. This can be changed, say to 100, using the input data command:

```
PARAM, ITMXSS, 100
```

Lanczos Method

The Lanczos method can be used on any size problem. The lowest ND eigenvalues (in magnitude), are calculated. The optional data, V1 and V2, can be used to define the buckling load factor range of interest. If V1 and V2 are present, the lowest ND buckling load factors (in magnitude) in this range are calculated. If only V1 is present, the ND lowest buckling load factors above V1 are calculated. If only V2 is present, the ND lowest buckling load factors below V2 are calculated.

The Lanczos method can only be used in conjunction with the sparse matrix solver. Therefore, the analysis parameter SOLVER should be set to 1. Note that not all installations have the sparse matrix solver.

Normalization of Eigenvectors

There are three types of normalizations available with the EIGR data statement for buckling loadcases. These are MAX, STIFF and POINT.

MAX is used to normalize the eigenvector so its maximum component is one at every design cycle.

STIFF is used to normalize the eigenvector so $\Phi^T K \Phi$ is one.

POINT is used to select a particular grid and component (degree of freedom) to normalize the eigenvector.

The buckling mode shapes can be printed using the same command as for frequency load cases (i.e., the **SVECTOR** command). Also, they can be printed using the **DISPLACEMENT** command. The referenced SET for both cases corresponds to mode numbers, not grid numbers.

2.16 Dynamic Analysis Calculation Control

In dynamic analysis the individual finite element matrices and element dynamic pressure and gravity load vectors are combined to form system stiffness [K], mass [M], viscous damping [B], and structural damping [K_s] matrices and system load vectors. The system structural damping is the sum of the global structural damping, G, multiplied by the system stiffness matrix and the elemental structural damping matrix, [K_4], which is the assembly of individual element stiffness matrices multiplied by the structural damping coefficient specified in their material data:

$$\left[K_4\right] = \sum_i GE_i \left[k_i\right] \tag{2-105}$$

$$\left[K_s\right] = G\left[K\right] + \left[K_4\right] \tag{2-106}$$

The global structural damping coefficient is specified with the analysis parameter **G**. For example, for 2% structural damping the command would be:

```
PARAM,G,0.02
```

Dynamic point forces and moments are then added to the system load vectors. Since there are six degrees of freedom for each grid the total number of degrees of freedom is six times the number of grid points. Each degree of freedom has both real and imaginary components.

MPC's and rigid elements constraint equations are added to the system stiffness matrix and load vectors.

Constrained degrees of freedom, specified by **SPC** data, are removed from the system matrices and load vectors. In *GENESIS* the rotational degrees of freedom for grids that are only connected to solid and/or rod elements and not referenced by MPC data are automatically removed from the system matrices and load vectors.

The system stiffness matrix may be singular if the user did not use SPC data to constrain all degrees of freedom that do not have stiffness associated with them. These degrees of freedom can automatically removed from the system matrices and load vectors using the analysis parameter **AUTOSPC**. The input data command to use this feature is:

```
PARAM,AUTOSPC,YES
```

If the AUTOSPC option is used then degrees of freedom with stiffness less than EPZERO multiplied by the average value of the diagonal elements are automatically constrained. The default value of **EPZERO** is 1.0E-8. The value of EPZERO can be changed with PARAM data. For example, to change the value of EPZERO to 1.0E-7 use the input data command:

```
PARAM,EPZERO,1.0E-7
```

At this point the linear system of matrix equations should not be singular and the grid point displacements can be found by solving the linear equation:

$$[M]\{\ddot{U}\} + [B]\{\dot{U}\} + [K]\{U\} + i[K_s]\{U\} = \{F\} \quad (2\text{-}107)$$

where {F} is a system load vector, and {U} is the displacement vector to be calculated.

The load and displacement vectors are, in general, complex. The system of equations must be solved for each loading frequency OMEGA. For each loading frequency the load vector is of the form $\{P\}e^{i\Omega t}$ and the displacements of the form $\{u\}e^{i\Omega t}$. Substituting into **(2-107)** gives

$$(-\Omega^2[M] + i\Omega[B] + [K] + i[K_s])\{u\} = \{P\} \quad (2\text{-}108)$$

2

Direct Dynamic Analysis

(2-108) can be written as

$$[C]\{u\} = \{P\} \quad (2\text{-}109)$$

where [C] is called the system complex matrix.

To solve for the displacements the system complex matrix must be triangularized. If the analysis parameter **SOLVER** is 2 (see Section **2.12**) then the system complex matrix is checked for singularities by comparing the ratio of the absolute value diagonal elements before and after triangularization. If the ratio is greater than **MAXRATIO** (default value = 1.0E7) then the matrix is considered singular. The value of MAXRATIO can be changed with PARAM data. For example, to change the value of MAXRATIO to 1.0E12 use the input data command:

```
PARAM,MAXRAT,1.0E12
```

If the analysis parameter **BAILOUT** is set to -1 (the default) and the analysis parameter SOLVER is 2, then the matrix triangularization process does not stop when the ratio is greater than MAXRATIO. If the analysis parameter BAILOUT is set to 1 with the input data command:

```
PARAM,BAILOUT,1
```

then the triangularization process stops when the ratio is greater than MAXRATIO.

Modal Dynamic Analysis

In modal dynamic analysis the displacements are calculated as a linear combination of mode shapes:

$$\{u\} = \sum_{i=1}^{N} \{\phi_i\} z_i = [\Phi]\{z\} \quad (2\text{-}110)$$

These mode shapes come from a frequency calculation load case and from residual vectors. Ordinarily, all modes calculated in the frequence calculation load case are used in the modal basis. The **MODESELECT** solution control command or the analysis parameters **LFREQ** and **HFREQ** can be used to select a subset of the calculated modes to be used in the modal basis. The residual vectors are calculated from a set of target vectors. Target vectors are determined from the static solution of unit loads applied to each degree of freedom in the **USET** named U6. If no USET named U6 is defined, then the static solutions of the real parts of the load applied at the first loading frequency of every modal dynamic load case are used as target vectors. The use of residual vectors leads to more accurate results. The creation of residual vectors can be controlled by the analysis parameter **RESVEC** (default=YES). To not use residual vectors, use the command:


```
PARAM,RESVEC,NO
```

The number of mode shapes used determines the accuracy of the results. In general, it is recommended that two times the number of modes that exist below the highest loading frequency be used. Substituting **(2-110)** into **(2-108)** and premultiplying by $[\Phi]^T$ leads to:

$$\begin{aligned}(-\Omega^2[\Phi]^T[M][\Phi] + i\Omega[\Phi]^T[B][\Phi]\\ +[\Phi]^T[K][\Phi] + i[\Phi]^T[K_s][\Phi])\{z\} = [\Phi]^T\{P\}\end{aligned} \tag{2-111}$$

which can be written as:

$$(-\Omega^2[m] + i\Omega[b] + [k] + i[k_s])\{z\} = [c]\{z\} = \{p\} \tag{2-112}$$

where

$$[m] = [\Phi]^T[M][\Phi] \tag{2-113}$$

$$[b] = [\Phi]^T[B][\Phi] \tag{2-114}$$

$$[k] = [\Phi]^T[K][\Phi] \tag{2-115}$$

$$[k_s] = [\Phi]^T[K_s][\Phi] \tag{2-116}$$

$$[p] = [\Phi]^T[P] \tag{2-117}$$

[m], [b], [k], and $[k_s]$ are called the modal matrices. Modal damping can be added to the system with **TABDMP1** data which is selected by the Solution Control command SDAMPING. If the analysis parameter **KDAMP** = 1 (default) the modal damping is added as viscous damping which leads to

$$[b] = [\Phi]^T[B][\Phi] + 2\pi f_i g(f_i) \tag{2-118}$$

where f_i is the frequency of the ith mode shape and $g(f_i)$ is the modal damping at frequency f_i from the modal damping table TABDMP1. If KDAMP = -1 the modal damping is added to the structural damping which leads to

$$[k_s] = [\Phi]^T[K_s][\Phi] + 4\pi_i^2 f_i^2 g(f_i) \tag{2-119}$$

Modal damping is not added to the residual vector modes.

In general, modal dynamic analysis is less costly than direct dynamic analysis, especially if there are many loading frequencies. Modal dynamic analysis may not be as accurate as direct dynamic analysis if not enough modes are used. Before an optimization run it is a good idea to check the accuracy of the modal dynamic analysis by comparing it to a direct dynamic analysis of the structure.

Guyan Reduction in Dynamics

The eigenvector calculated in a frequency loadcase can be used in a modal dynamic loadcase, although this is not recommended because those eigenvectors are only approximations of the real eigenvectors.

Performance Issues

See Section **Static Analysis Calculation Control** (p. 76).

2.16.1 User Function of Frequency Response Results

Some physical responses, for example, the acoustic response at a measurement point, can be well approximated by a linear function of the dynamic displacements, velocities or accelerations. The linear function can be written as:

$$UFDISP_i = \{C(\Omega)\}_i^T\{u\} \tag{2-120}$$

$$UFVELO_i = \{C(\Omega)\}_i^T\{v\} \tag{2-121}$$

$$UFACCE_i = \{C(\Omega)\}_i^T\{a\} \tag{2-122}$$

2

where: i represents the field point (measurement point), C is the vector of complex coefficients for all the degrees of freedom, and u, v, a are the complex dynamic displacements, velocities and accelerations, respectively, which are implicit functions of the loading frequency, Ω.

While, for optimization, *GENESIS* provides a synthetic response capability that could be used to construct such linear functions, it would not be efficient to do so, since the linear functions typically involve many degrees of freedom, and the linear coefficients vary over the loading frequency range. To efficiently calculate such linear functions, *GENESIS* provides a capability to load the linear coefficients from a special file.

The **UFDATA** executive control command allows the user to specify a file containing data to create linear user functions of dynamic displacements, velocities and accelerations. The results of the linear combinations can be printed out using the **UFDISP**, **UFVELO** and/or **UFACCE** solution control commands.

The UFDATA file is a formatted text file that resembles bulk data. Unlike bulk data, the order of the items in the file is important. It contains three sections:

1. Header line
2. Grid list
3. Coefficient data

Header Line

The first line of the file must be the header line, and has the following format::

1	2	3	4	5	6	7	8	9	10
$GENESIS UFDATA			NLF		NFP		NG		

Field	Information	Description
1-2	File magic	Must be the characters “$GENESIS UFDATA “
4	NLF	The number of distinct loading frequencies for which coefficients are defined (Integer > 0).
6	NFP	The number of field points for which coefficients are defined (Integer > 0).
8	NG	The number of different grids that have non-zero coefficients for any field point at any loading frequency (Integer > 0).

Grid List

After the header line, the grids that have non-zero coefficients for any field point at any loading frequency must be identified:

1	2	3	4	5	6	7	8	9	10
UFGRID	G1	G2	G3	G4	G5	G6	G7	G8	
+	G9	G10	…						

2

Field	Information	Description
2-9	Gi	Grid identification number of a **GRID** entry (Integer > 0). There must be exactly NG grid IDs listed. No grid identification number may be repeated.

Coefficient Data

After the grid list, the coefficient data is entered, for every field point and every loading frequency in the following sequence.

SEQ = 0

DO I = 1,NLF

    DO J = 1, NFP

        SEQ = SEQ + 1

        Write data for field point J at loading frequency I

    END DO

END DO

The data for a field point at a loading frequency is entered as follows:

1	2	3	4	5	6	7	8	9	10
UFDATA	SEQ	I	FREQ	FPID					
*	G1		CXR1		CYR1		CZR1		
*			CXI1		CYI1		CZI1		
*	G2		CXR2		CYR2		CZR2		
*			CXI2		CYI2		CZI2		
*	Gi		CXRi		CYRi		CZRi		
*			CXIi		CYIi		CZIi		

Field	Information	Description
2	SEQ	UFDATA sequence number (Integer > 0). This must count up sequentially from 1 to the total number of UFDATA entries (=NLF*NFP). The entries must be in order.
3	I	The index of the loading frequency (Integer > 0).
4	FREQ	The loading frequency in Hz (Real > 0.0).
5	FPID	The field point identification number (Integer > 0).
2-3	Gi	Grid identification number of a **GRID** entry (Integer > 0).
4-5	CXRi	Real part of the coefficient for the basic x-translation component of grid Gi (Real).
6-7	CYRi	Real part of the coefficient for the basic y-translation component of grid Gi (Real).
8-9	CZRi	Real part of the coefficient for the basic z-translation component of grid Gi (Real).

4-5	CXIi	Imaginary part of the coefficient for the basic x-translation component of grid Gi (Real).
6-7	CYIi	Imaginary part of the coefficient for the basic y-translation component of grid Gi (Real).
8-9	CZIi	Imaginary part of the coefficient for the basic z-translation component of grid Gi (Real).

Remarks:

1. There can up up to 2*NG continuation lines. If all coefficients for a particular Gi are zero for a field point / loading frequency, then the two continuation lines for that Gi can be omitted.
2. Grid Gi must have been listed on the UFGRID entry.
3. The coefficients are only defined for the translation components of the grid displacement and must be defined in the basic coordinate system, even if the grid specifies a non-basic output coordinate system.
4. The FREQ values must be the same for all field points.
5. The FREQ values do not have to match the loading frequencies specified by the **FREQUENCY** set of the frequency response loadcase(s). To evaluate the user function at a frequencies different from any given FREQ value, an interpolation of the coefficients with the closest FREQ values will be used.

2.16.2 Equivalent Radiated Power

In NVH applications, it is desirable to calculate the noise radiated into an acoustic fluid from vibration of the surrounding structure. The sound power radiating into an acoustic fluid is determined using a coupled structural-acoustic problem to calculate both the velocities and pressures at the structure-fluid interface. The equivalent radiated power calculation makes the assumption that the pressure at the structure-fluid interface is equal to the normal velocity times the fluid density times the speed of sound in the fluid. This makes it possible to estimate the sound power with an uncoupled structural problem. For a given structural panel, the equivalent radiated power is calculated as:

$$\mathrm{ERP} = \frac{1}{2}\rho c \int\int v_n^* v_n \, dA \tag{2-123}$$

where: ρ is the fluid density, c is speed of sound in the fluid, v_n is the component of the velocity normal to the structural panel and A is the panel area. * is the complex conjugate operator. Note that ERP is an implicit functions of the loading frequency, Ω.

The **ERPPNL** bulk data is used to define panels for ERP calculations. ERPPNL associates a panel name with an element set that contains the elements of the panel. Note that a panel can only be comprised of shell or composite CQUAD4, CTRIA3, CQUAD8 and/or CTRIA6 elements. For each panel, *GENESIS* calculates ERP as follows:

$$\mathrm{ERP} = \frac{1}{2}\mathrm{ERPRLF}\ \mathrm{ERPRHO}\ \mathrm{ERPC}\sum v_n^* v_n A_e \tag{2-124}$$

where **ERPRLF** is the radiation loss factor, **ERPRHO** is the fluid density, **ERPC** is the speed of sound of the fluid, v_n is the normal component of the velocity at each grid of the panel, and A_e is the effective panel area of each grid. ERPRLF, ERPRHO and ERPC are defined by **PARAM** bulk data entries.

It is often convenient to consider sound pressure values using a decibel scale. *GENESIS* calculates ERP on a decibel scale as follows:

$$\mathrm{ERP(dB)} = 10\ \log\left(\frac{\mathrm{RHOCP}}{\mathrm{ERPREFDB}}\mathrm{ERP}\right) \tag{2-125}$$

where **RHOCP** is the decibel scale factor and **ERPREFDB** is the decibel reference value, both defined by PARAM bulk data entries.

The calculated ERP and ERP(dB) values can be printed out using the **ERP** solution control command.

2.16.3 Modal Contribution

In modal dynamic analysis the displacements are calculated as a linear combination of mode shapes:

$$u_j = \sum_{i=1}^{N} \phi_{ji} z_i \tag{2-126}$$

The individual modal components, $\phi_{ji} z_i$, are complex numbers that sum together to equal the total complex displacement u_j. To understand how important the individual modes are to the total displacement, the contribution of each mode can be calculated. The modal contribution of mode i to degree of freedom j is defined as:

$$MC_{ji} = Re\left(\frac{\phi_{ji} z_i}{u_j}\right) \tag{2-127}$$

That is, the modal contribution is the real part of the complex division of the modal component by the total displacement. If complex numbers are imagined as two-dimensional vectors, then the modal contribution is equivalent to the dot product of the modal component with the total displacement divided by the magnitude of the total displacement.

Modal contribution tables can be printed with the **MCONTRIB** solution control command.

2.16.4 Coupled Fluid-Structure Models

If the model contains any fluid elements, the frequency response loadcases will perform a coupled fluid-structure analysis. The fluid portion of the model is governed by the linearized wave equation, with the state variable being p, the pressure change from the static pressure level. A fluid “mass” matrix, $[M_f]$, and “stiffness” matrix, $[K_f]$ are assembled. Note that while these matrices are analogous to their structure counterparts in the governing equations, then do not actually have units of mass or stiffness. A coupling matrix, [A], is assembled to connect the fluid pressures on the outer fluid boundary with the structural accelerations normal to the surface of shell elements close to that surface. The total governing system equation for the frequency response problem at loading frequency Ω is:

$$\begin{bmatrix} -\Omega^2[M] + i\Omega[B] + [K] + i[K_s] & -[A]^T \\ -[A] & -[M_f] + \frac{1}{\Omega^2}[K_f] \end{bmatrix} \begin{Bmatrix} u \\ p \end{Bmatrix} = \begin{Bmatrix} P \\ P_f \end{Bmatrix} \qquad (2\text{-}128)$$

“Loading” on the fluid system, $\{P_f\}$, is normally zero, but if desired, values can be assigned using **RLOAD1** and/or **RLOAD2** in combination with **DAREA**/**DELAY**/**DPHASE**. Enforced pressure levels can be set using **RLOAD1**/**RLOAD2** in combination with **SPCD**.

By default, areas of the fluid boundary that have no close shell elements will be considered to be bounded by a rigid structure. “Open” boundaries can be simulated by using **SPC** to enforce fluid pressures to zero.

A structural damping analog can be added into the fluid system by defining the analysis parameter **GFL**. This will make the $[K_f]$ matrix complex, with the imaginary part being equal to GFL times the real part.

The complex govering system can be solved directly, as in direct dynamic analysis. The system can also be solved using a modal approximation. In this case, the structure system eigenvalue problem will be solved independently of the fluid system eigenvalue problem. The eigenvalue calculation methods for the structure system the the fluid system can be the same or can be different.

DISPLACEMENT output requests for fluid grids will print the fluid pressure in component T1.

2.17 Random Response Analysis Calculation Control

Random response analysis is available as a postprocessing analysis of one or more existing frequency response loadcases.

The program can perform the following random response analysis:

1. Power spectral density functions (PSDF) , $S_j(\omega)$
2. Autocorrelation function (ATOC), Rj(t)
3. root mean square (RMS), u_j
4. Cumulative root mean square (CRMS),$u_j(\omega)$
5. number of zero crossings, N0

The above analyses are available for the following user requested responses:

1. Displacements
2. Velocities
3. Accelerations
4. Forces
5. Stresses
6. Strains

The random responses are calculated using frequency response results under the assumption that the system is linear and the excitations are stationary with respect to the time.

Two key functions are associated to a physical variable $u_j(t)$ (displacement, velocity, acceleration, stress, strain or force): the power spectral density function and the autocorrelation function.

Power Spectral Density Function Definition

The power spectral density function and is defined by:

$$S_j(\omega) = \lim_{T \to \infty} \frac{2}{T}\left[\int_0^T e^{-i\omega t} u_j(t)dt\right]^2 \tag{2-129}$$

Autocorrelation Function Definition

The auto correlation function that is defined as follows:

$$R_j(\tau) = \lim_{T \to \infty} \frac{1}{T}\int_0^T u_j(t)u_j(t-\tau)dt \tag{2-130}$$

Relationship Between Autocorrelation Function and the Power Spectral Density Function

The two defined functions are not independent, and it can be shown that the autocorrelation function is the Fourier transform of the power spectral density function. Therefore, Eq. **(2-130)** can be written as:

$$R_j(\tau) = \frac{1}{2\pi}\int_0^{\infty} S_j(\omega)\cos(\omega\tau)d\omega \tag{2-131}$$

The above equation allows the autocorrelation function to be evaluated for a given time lag (specified in RANDT1) once the corresponding power spectral function S_j is known.

Fourier analysis

The transfer function theory states that if $H_{ja}(\omega)$ is the frequency response of a physical variable u_j due to an excitation source $Q_a(\omega)$

$$u_j(\omega) = H_{ja}(\omega) \times Q_a(\omega) \tag{2-132}$$

where $u_j(\omega)$ is the Fourier transform of the u_j and $Q_a(\omega)$ is the Fourier transform of Q_a. The power spectral density of the response $S_j(\omega)$, is related to the power spectral density of the loading source $S_a(\omega)$ by the following expression:

$$S_j(\omega) = |H_{ja}(\omega)|^2 \times S_a(\omega) \tag{2-133}$$

If there are multiple sources and they are statistically independent, then the power spectral density of the total response can be calculated using the following expression:

$$S_j(\omega) = \sum_a S_{ja}(\omega) = \sum_a |H_{ja}(\omega)|^2 \times S_a(\omega) \tag{2-134}$$

$S_a(\omega)$ in the expression above corresponds to the power spectral density function values of the loading provided through **RANDPS** (using K=0) and **TABRND1**.

If there are multiple sources and they are statically correlated, then the auto spectral density of the response is evaluated using the following expression:

$$S_j(\omega) = \sum_a \sum_b H_{ja}(\omega) \times \overset{\circ}{H}_{jb}(\omega) \times S_{ab}(\omega) \tag{2-135}$$

$S_{ab}(\omega)$ in the expression above corresponds to the power spectral density function values of the loading. These values are specified using **RANDPS** and **TABRND1** entries.

Equations (2-134) and (2-135) are used to calculate the desired spectral density functions using the frequency responses results. Once the spectral density function is known, RMS, CRMS and N0 can be calculated.

The Root Mean Square (RMS) Response:

The root mean square value is calculated using the following expression:

$$\bar{u}_j^2 = \frac{1}{2\pi}\int_0^{\infty} S_j(\omega)d\omega = R_j(0) \tag{2-136}$$

The Cumulative Root Mean Square (CRMS) Response:

The cumulative root mean square values are evaluated using the following expression:

$$\bar{u}_j(\omega)^2 = \frac{1}{2\pi}\int_0^{\omega} S_j(\omega)d\omega \tag{2-137}$$

The Number of Zero Crossing (N0):

The expected value of the number of zero crossings with positive slope per unit time is a statistical quantity of interest. Its value can be found using the power spectral density functions as follows:

$$N0 = \sqrt{\left[\frac{\int_0^{\infty}\left(\frac{\omega}{2\pi}\right)^2 S_j(\omega)d\omega}{\int_0^{\infty} S_j(\omega)d\omega}\right]} \tag{2-138}$$

From its definition, it can be seen that this quantity is a mean frequency where the power spectral density function is used as weighting factor. This quantity is sometimes referred to as the apparent frequency and is a quantity of interest for fatigue analysis.

Output

Output for the responses described above can be requested using the following solution control commands:

DISPLACEMENT, **VELOCITY**, **ACCELERATION**, **STRESS**, **STRAIN** or **FORCE**

The key options for each of the above commands are: PSDF, ATOC and RMS.

The PSDF option is used to request output for the power spectral density functions evaluated a each frequency value in the random loop. The ATOC option is used to request output for autocorrelation function values evaluated at each time lag. The RMS option is used to request a) root mean square, b) cumulative root mean square function values evaluated at each frequency value in the random loop, and c) the number of zero crossings.

The results from frequency response can be printed to the output file and/or to the punch post processing file. The options RPRINT, RPUNCH control this on the above solution control commands.

2.18 Heat Transfer Analysis

Heat transfer analysis is used to determine the static steady state temperature distribution in the structure. This temperature distribution can then be used as a thermal load in a static structural analysis.

2.18.1 Conduction Elements

The scaler element CELAS1, line elements CROD, CBAR and CBEAM, plane elements CQUAD4 and CTRIA3, general curved elements CQUAD8 and CTRIA6, the axisymmetric element CTRIAX6, and solid elements CHEXA, CPENTA, CTETRA, CHEX20 and CPYRA are all used as conduction elements in the heat transfer analysis. Rigid elements: RROD, RBAR, RBE1 and RBE2; interpolation elements: RBE3, RSPLINE and BOLT; mass elements: CONM2, CONM3 and CMASS1, damping elements: CVISC and CDAMP1, CDAMP2; CBUSH, CGAP and CSHEAR elements are ignored in heat transfer analysis. The isotropic thermal conductivity for the elements is specified with the **MAT4** data. In addition, the plane and solid elements may have anisotropic thermal conductivities that are specified on **MAT5** data.

Grid point offsets for CBAR, CBEAM, CQUAD4, CTRIA3, CQUAD8 and CTRIA6 elements are treated as perfect conductors and do not effect the heat transfer analysis. For the plane elements (CQUAD4/CTRIA3) and curved shell elements (CQUAD8/CTRIA6), heat conduction is only considered in the membrane direction. Heat conduction is not considered through the thickness.

2.18.2 Boundary Conditions

Grid point temperatures can be specified using **SPC** or **SPCD** data. Temperatures of zero degrees can be specified using **SPC1** data. If a boundary grid point has no loads and is not constrained an adiabatic boundary condition is used. MPC's can be used to specify a linear relationship between a set of grid point temperatures.

2.18.3 Heat Transfer Loads

Element volume internal heat generation is specified using the **QVOL** data. All conduction elements, except the CELAS1 which has no volume, may be specified with this data. Heat flux into a set of grid points can be specified using the **QHBDY** data statement.

2.18.4 Heat Transfer Boundary Load Element (CHBDY)

The **CHBDY** element is used to specify heat flux, thermal vector flux, and/or convection into a set of grid points. This element references from one to four grid points. The **PHBDY** property data for this element is used to specify the material and absorptivity of the element. This element references **MAT4** data for the convective film coefficient for convection loads.

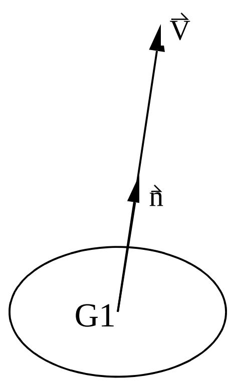
QBDY1 data is used to specify uniform heat flux into CHBDY elements while **QBDY2** data is used to specify grid point heat flux into CHBDY elements. Thermal vector flux can be specified using the **QVECT** data.

There are five types of CHBDY elements: POINT, LINE, ELCYL, AREA3 and AREA4. The figures below defines the grid points associated with each type. Also, the normal vector definition is shown for each type.

2

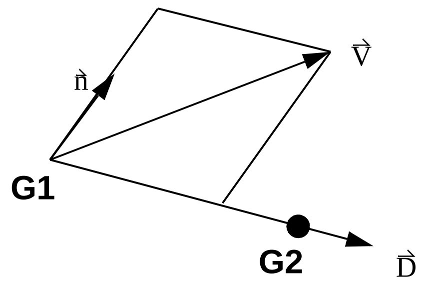
CHBDY type - POINT

The unit normal vector is internally calculated by $\vec{n} = \frac{\vec{V}}{|\vec{V}|}$ where $\vec{V}$ is given in the basic coordinate system.



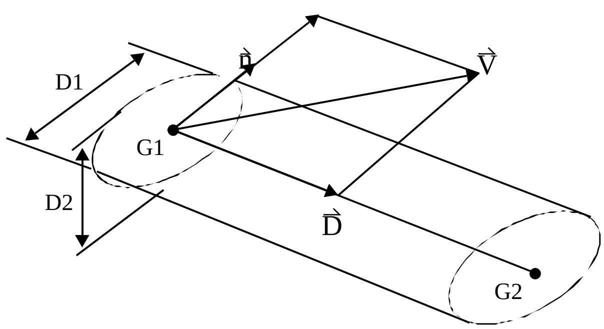
CHBDY type - LINE

The unit normal vector is defined to be perpendicular to the vector connecting grids 1 and 2 and the line in the plane defined by $\vec{V}$ and $\vec{D}$.



CHBDY type - ELCYL

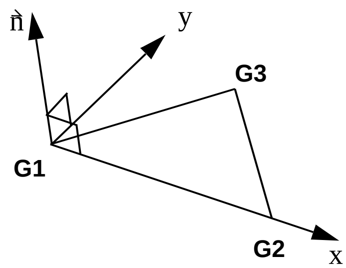
The unit normal vector is defined to be perpendicular to $\vec{D}$ and to be in the plane defined by $\vec{V}$ and $\vec{D}$. The elliptical diameters D1 and D2 are measured parallel and perpendicular to the unit vector $\vec{n}$.



2

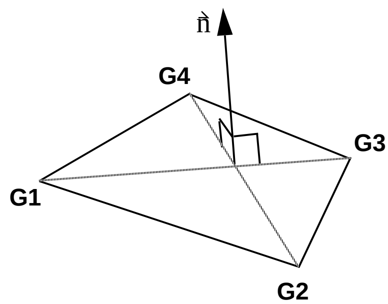
CHBDY type - AREA3

The unit normal vector is perpendicular to the plane defined by grids 1, 2 and 3.



CHBDY type - AREA4

The unit normal vector is defined to be perpendicular to the plane defined by lines connecting grids 1, 3 and grids 2 and 4.



2.18.5 Heat Transfer Calculation Control

Heat transfer analysis by the finite element method parallels static analysis. Instead of a stiffness matrix, a conduction matrix is used. Point heat flux is used instead of point forces and distributed surface fluxes are used instead of pressure loads. Volumetric heat generation can be thought of as a body force similar to gravity. Enforced temperatures are used instead of enforced displacements. The convection boundary condition can be thought of as an elastic foundation boundary condition.

In heat transfer analysis the individual finite element conduction matrices and flux load vectors are combined to form a system conduction matrix and system flux load vector. Grid point fluxes are then added to the system load vector. Since there is only one degree of freedom for each grid, the temperature, the total number of degrees of freedom is equal to the number of grid points.

Constrained temperatures, specified by SPC data, are removed from the system conduction matrix and flux load vector. MPC constraint equations are added to the system conduction matrix and flux load vector. In *GENESIS* the grids that are not connected to any elements or referenced by MPC data are automatically removed from the system conduction matrix and flux load vector.

At this point the system matrix should not be singular and the grid point temperatures can be found by solving the linear equation:

$$[k]\{T\} = \{F\}$$

where [k] is the system conduction matrix, {F} is the system flux load vector, and {T} is the temperature vector to be calculated.

The system conduction matrix may still be singular if the user did not use SPC data to constrain all degrees of freedom that do not have conduction associated with them. These degrees of freedom can by constrained automatically using the analysis parameter **AUTOSPC**. The input data command to use this feature is:

```
PARAM,AUTOSPC,YES
```

If the AUTOSPC option is used then degrees of freedom with conduction less than **EPZERO** multiplied by the average value of the diagonal elements are automatically constrained. The default value of EPZERO is 1.0E-8. The value of EPZERO can be changed with PARAM data. For example, to change the value of EPZERO to 1.0E-6 use the input data command:

```
PARAM,EPZERO,1.0E-6
```

To solve for the temperatures the conduction matrix must be triangularized. The conduction matrix is checked for singularities by comparing the ratio of the diagonal elements before and after triangularization. If the ratio is greater than **MAXRATIO** (default value = 1.0E7) then the matrix is considered singular. The value of MAXRATIO can be changed with PARAM data. For example, to change the value of MAXRATIO to 1.0E12 use the input data command:

```
PARAM,MAXRAT,1.0E12
```

If the analysis parameter **BAILOUT** is set to -1 (the default), then the matrix triangularization process does not stop when the ratio is greater than MAXRATIO. If the analysis parameter BAILOUT is set to 1 with the input data command:

```
PARAM,BAILOUT,1
```

then the triangularization process stops when the ratio is greater than MAXRATIO.

If a diagonal element is exactly zero after triangularization then the matrix is singular and the process stops, even if BAILOUT is set to -1.

2

2.19 Units

GENESIS itself does not have any built-in requirements for the units of input data. It is the user's responsibility to ensure that all of the input data is in consistent units. The user's choice for the units of the input numbers also dictates how any output numbers must be interpreted.

Structural analysis generally requires units for physical quantities of length, time, mass, force, temperature, weight, etc.

Let the following symbols represent the user's chosen units for the respective quantities:

L = Length (distance)

V = Velocity

A = Acceleration

t = Time

f = Frequency

M = Mass

F = Force

W = Weight

P = Pressure

ρ = Density

T = Temperature

Consistency of units implies relationships like the following:

$F = M\,L / t^2$

$W = M$ g (g = acceleration due to gravity in units of L / t^2)

$V = L / t$

$A = L / t^2$

f = cycles / t

$P = F / L^2$

$\rho = M / L^3$

etc.

In structural analysis, it is almost always desirable to measure frequencies in Hz (cycles/second). This dictates that the unit used for time (t) must be seconds. The user must select two other basic units among (L, M, F, P, W) and the remaining units will be derived from that selection. Units for temperature must also be selected. This choice will almost certainly be Fahrenheit if the basic units are among the English system or Celsius if the basic units are among the metric system. The thermal expansion coefficients must be consistent with the unit for temperature.

Example 1

Let

t = sec (seconds)
L = mm (millimeters)
F = N (Newtons)
T = degree Celsius

Now, we can derive the following:

$M = F\,t^2 / L$ = N sec^2/mm = kg m/mm = 1000 kg = Mg (megagrams)

$P = F / L^2$ = N/mm^2 = 1000000 N/m^2 = MPa (megaPascals)

$\rho = M / L^3$ = Mg/mm^3 = 10^9 g/cm^3

g = 9810 mm/sec^2 (approx.)

$W = M$ g = 9810 Mg mm /sec^2 = 9810 N

f = Hz

Numbers in the input data should be entered using these units. For example, density should be input in 10^9 g/cm^3. This means that a material having a density of 7.8 g/cm^3 should have the number 7.8E-9 entered as the density on MATi data. Output numbers should likewise be interpreted using these units. For example, an output mass of 0.325 means 0.325 Mg = 325 kg.

Using this system of units, material data should have numbers similar to the following:

Material	E (in MPa)	ρ (in Mg/mm^3)
Steel	207.0E3	7.8E-9
Aluminum	72.0E3	2.8E-9
Rubber	2.0	1.1E-9

2

2

Example 2

Let

t = sec (seconds)
L = in (inches)
F = lb (pounds)
T = degree Fahrenheit

Now, we can derive the following:

$M = F\,t^2 / L$ = lb sec^2/in = slug ft/in = 12 slug = 386.22 lbm (386.22 pound mass)

$P = F / L^2$ = lb/in^2 (psi)

$\rho = M / L^3$ = 386.22 lbm/in^3

g = 386.22 in/sec^2 (approx.)

$W = M$ g = 12 slug 386.22 in/sec^2 = 386.22 lb

f = Hz

A mass of 12 slugs (or 386.22 lbm) is sometimes called a slinch. A force of one pound will accelerate one slinch at one inch per second squared (1 lb = 1 slinch in / sec^2).

Using this system of units, material data should have numbers similar to the following:

Material	E (in psi)	ρ (in 386.22 lbm/in^3)
Steel	30.0E6	7.3E-4
Aluminum	10.0E6	2.6E-4
Rubber	290.0	1.0E-4

Example 3

Let

t = sec (seconds)
L = cm (centimeters)
F = N (Newtons)
T = degree Celsius

Now, we can derive the following:

$M = F\,t^2 / L$ = N sec^2/cm = kg m/cm = 100 kg
$P = F / L^2$ = N/cm^2 = 10000 N/m^2 = 10 kPa
$\rho = M / L^3$ = 100 kg/cm^3 = 10^5 g/cm^3
g = 981 cm/sec^2 (approx.)
$W = M$ g = 98100 kg cm/sec^2 = 981 N
f = Hz

Example 4

Let

t = sec (seconds)
L = m (meters)
M = kg (kilograms)
T = degree Celsius

Now, we can derive the following:

$F = M\,L / t^2$ = kg m/sec^2 = N (Newtons)
$P = F / L^2$ = N/m^2 = Pa (Pascals)
$\rho = M / L^3$ = kg/m^3 = 10^{-3} g/cm^3
g = 9.81 m/sec^2 (approx.)
$W = M$ g = 9.81 kg m/sec^2 = 9.81 N
f = Hz

2.20 Element Verification

GENESIS internally performs several tests to measure the distortion of the shape of the CSHEAR, CTRIA3, CQUAD4, CTRIA6, CQUAD8,CTRIAX6, CTETRA, CPENTA, CHEXA, CHEX20 and CPYRA elements of the user's finite element model. The results of these tests are compared against two limit ranges.

GENESIS first checks if each element is acceptable or not. If the element is acceptable then *GENESIS* checks if the element is in a recommended range. If the element is between the acceptable but not recommended range, a warning message will be given. The message will include the element number with the test parameter value and the recommended range. On the other hand, if the element is in the unacceptable range *GENESIS* will stop with an error message that includes the element number, the test parameter values, and the acceptable range.

2

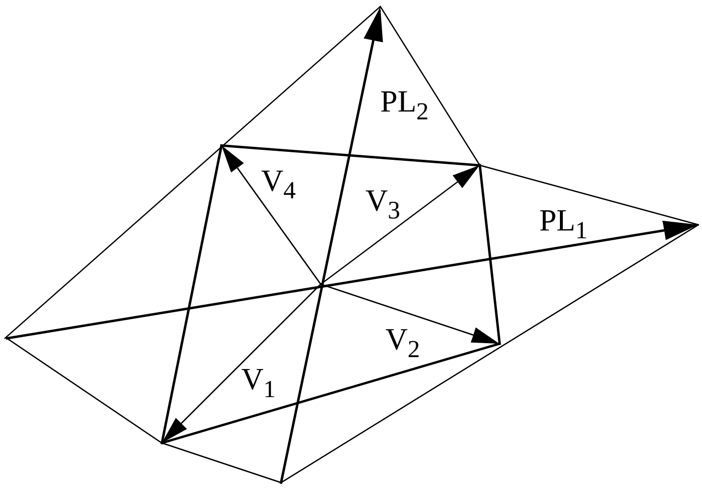
The results of all the tests can be printed in the output file by using the **SHAPECK** PARAMeter. A value of SHAPECK=5 will print all the shape characteristics of all the checked elements.

It is important to note that there is no a unique way of defining each distortion parameter. Therefore, the results given by the program may or may be the same as results given by other codes or pre or post-processors.

The accuracy of the finite element analysis results are, in general, related to the quality of the finite element mesh. However this is not always true. For example, a rectangular membrane assembled with a distorted mesh subject to a constant in plane loading will give exact results. For this reason and because sometimes it is not possible to have all elements in a mesh reasonably undistorted, the limits for issuing warning and error messages can be changed by the user. The **DISTOR** data statement allows those changes. In addition, the SHAPECK analysis parameter allows conversion of error messages to warning messages, skipping the checking, skipping the printing of warning messages, etc. (see the PARAM bulk data command (p. 560) for a detailed explanation of SHAPECK).

In general, the meshes should be formed with undistorted elements and it is not recommended to increase the distortion limits or to skip the checking. Skipping the checking should be only done when a problem is rerun and it is known that the mesh provides good results.

The definition of the distortion parameters requires the definition of two auxiliary vectors. These auxiliary vectors are called the PLANE VECTORS. The plane vectors allow the definition of a unique plane that represents a warped face of a solid element or a QUAD4.



The plane vectors are:

$$PL_1=(V_3+V_2)-(V_1+V_4) \tag{2-139}$$

$$PL_2=(V_3+V_4)-(V_1+V_2) \tag{2-140}$$

where V_1, V_2, V_3 and V_4 are the vectors that connect the four nodes with the center of gravity of the element.

2.20.1 CTRIA3 Shape Verifications

The following characteristics of the shape of the triangular element CTRIA3 are calculated and checked:

- CTRIA3 Aspect Ratio

- CTRIA3 Skew Angle

In addition, the program checks that there are no collapsed nodes i.e. that the three nodes of the CTRIA3 have different coordinates.

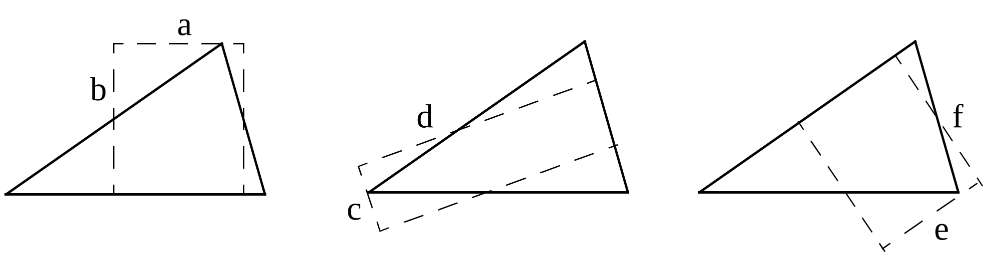
CTRIA3 Aspect Ratio

The aspect ratio of the CTRIA3 is defined as follows:

For each of the three edges of the triangle a rectangle is constructed which has one of its side on the edge, the parallel side passes through the grid which is opposite to the edge and the perpendicular sides of the rectangle pass through the mid points of the other two edges. Then the maximum aspect ratio of the three rectangles is the aspect ratio of the triangle:

$$AR = MAX\ (AR_1, AR_2, AR_3\ ) \tag{2-141}$$

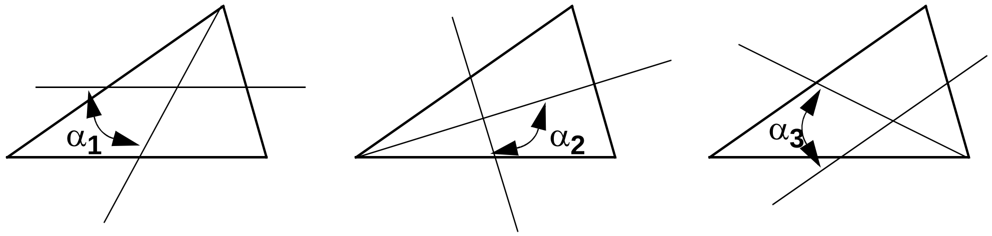
where AR_1 = MAX { a/b, b/a }, AR_2 = MAX { c/d, d/c }, AR_3 = MAX { e/f, f/e }



CTRIA3 Skew Angle

The skew angle of a CTRIA3 is calculate by:

1. calculating the mid-points of the edges
2. calculating the minimum angle between a line that connects two mid-points and another line that connects the third mid-point with the grid that is opposite to it. In total there are three of these angles (See the figure below).



3. The difference between 90 degrees and the minimum of the three angles calculated in 2) is the skew angle, i.e.:

$$\mathrm{SKEW} = 90 - \mathrm{MIN}\ (\alpha_1, \alpha_2, a_3) \quad (2\text{-}142)$$

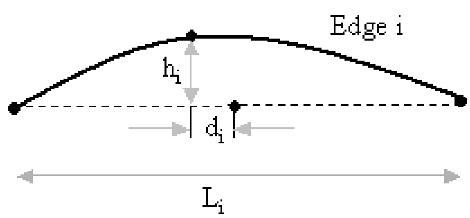
2.20.2 CTRIA6 Shape Verifications

The following characteristics of the shape of the triangular element CTRIA6 are calculated and checked:

- CTRIA6 Aspect Ratio
- CTRIA6 Skew Angle
- CTRIA6 Hoe normal offset
- CTRIA6 Hoe tangent offset

In addition, the program checks that there are no collapsed nodes i.e. that the six nodes of the CTRIA6 have different coordinates. The Aspect Ratio and Skew Angle tests are same as that for CTRIA3, i.e., only corner nodes are used.

CTRIA6 Hoe Normal and Hoe Tangent Offset



The Hoe Normal Offset is the ratio of the maximum normal offset of edge mid-side node to the distance between the edge corner nodes.

HOE NORMAL OFFSET = hi/Li

The Hoe Tangent Offset is the ratio of the distance between the real and ideal mid-side node and the distance between the edge corner nodes. If the mid-side node does not lie on the line connecting the corner nodes (i.e. Hoe Normal is non-zero), then its projection onto the line connecting the corner nodes is used for the calculation.

HOE TANGENT OFFSET = di/Li

The offsets are calculated for all edges of the element.

2.20.3 CQUAD4 Shape Verifications

The following characteristics of the quadrilateral element CQUAD4 are calculated and checked:

- CQUAD4 Aspect Ratio

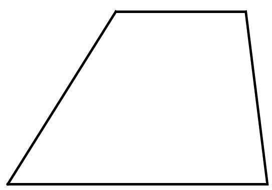
- CQUAD4 Skew Angle

- CQUAD4 Taper

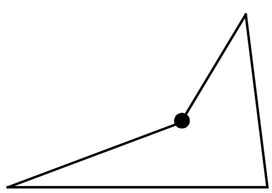
- CQUAD4 Warp Angle

- The Four Interior Angles of the CQUAD4

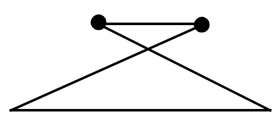
In addition the program checks the convexity of the CQUAD4 and its topology. The convexity test checks that there are no re-entrant angles in the CQUAD4 and the topology test checks that there are no collapsed nodes, i.e. that the four nodes of the CQUAD4 have different coordinates. The figure bellow shows the concept of re-entrant angles.



No Re-entrant Angle



Single Re-entrant Angle



Double Re-entrant Angle

CQUAD4 Aspect Ratio

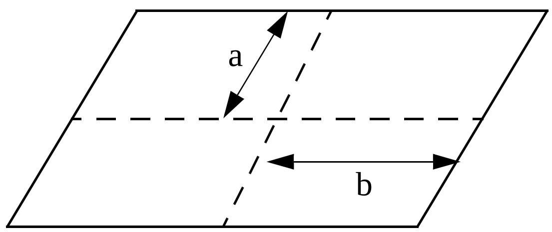
The aspect ratio of a CQUAD4 is defined as follows:

$$AR = MAX\left[\frac{|PL1|}{|PL2|}, \frac{|PL2|}{|PL1|}\right] \tag{2-143}$$

where $|PLi|$ represents the length of the plane vector i.

For the special case of a non-warped CQUAD4 with a parallelogram shape the expression for the A.R. represents the following:

$$AR = MAX\left[\frac{a}{b}, \frac{b}{a}\right] \tag{2-144}$$



CQUAD4 Aspect Ratio of Parallelogram

CQUAD4 Skew Angle

The skew angle of a CQUAD4 is defined as the difference between 90 degrees and the angle between the plane vectors.

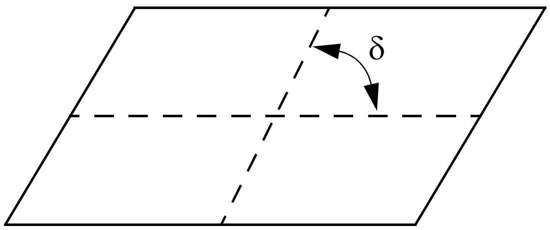
i.e.

$$SA = 90 - \mathrm{ANGLE}(PL_1, PL_2) \quad (2\text{-}145)$$

where PL_i is the plane vector i.

When the quad4 is not warped and has a parallelogram shape the expression of the S.A. reduces to:

$$SA = 90 - \delta \quad (2\text{-}146)$$



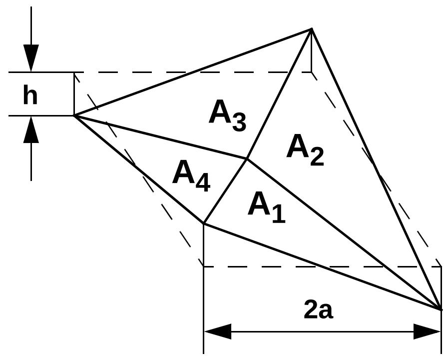
CQUAD4 Skew Angle of Parallelogram

CQUAD4 Taper

The CQUAD4 TAPER is defined as four times the minimum of the interior areas of the CQUAD4 divided by the sum of the four interior areas. An interior area of the CQUAD4 is defined as the area between two consecutive nodes and the centroid of the element.

i.e.

$$\mathrm{TAPER} = \frac{4\mathrm{MIN}(A_1, A_2, A_3, A_4)}{A_1 + A_2 + A_3 + A_4} \tag{2-147}$$



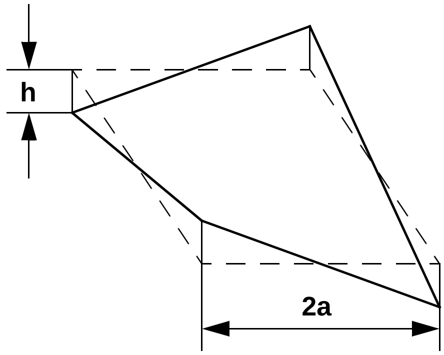
Note that with this definition, a rectangle will have a taper of one, while a collapsed triangle will have a taper equal to zero.

CQUAD4 Warping Angle

The warping angle of the CQUAD4 is defined as:

$$\mathrm{WARP} = \operatorname{atan}\left(\frac{h}{a}\right) \tag{2-148}$$

where h and a are defined in the figure below:



CQUAD4 Interior Angles

The interior angles are simply the angles between consecutive edges of the CQUAD4. For warped CQUAD4 the sum of the interior angles is, in general, not equal to 360 degrees. A negative value of an interior angle represent a re-entrant angle.

2.20.4 CQUAD8 Shape Verifications

The following characteristics of the quadrilateral element CQUAD8 are calculated and checked:

- CQUAD8 Aspect Ratio

- CQUAD8 Skew Angle

- CQUAD8 Taper

- CQUAD8 Warp Angle

- The Four Interior Angles of the CQUAD8

- CQUAD8 Hoe Normal Offset

- CQUAD8 Hoe Tangent Offset

In addition the program checks the convexity of the CQUAD8 and its topology. The convexity test checks that there are no re-entrant angles in the CQUAD8 and the topology test checks that there are no collapsed nodes, i.e. that the eight nodes of the CQUAD8 have different coordinates.

Aspect Ratio, Skew Angle, Taper, Warp Angle and Interior Angle checks are same as that of CQUAD4, i.e., using only the 4 corner nodes. Hoe Normal and Hoe Tangent offsets are calculated for each of the 8 edges as described for CTRIA6.

2.20.5 Shear Panel Shape Verifications

The following characteristics of the quadrilateral element CSHEAR are calculated and checked:

- CSHEAR Aspect Ratio

- CSHEAR Skew Angle

- CSHEAR Taper

- CSHEAR Warp Angle

- The Four Interior Angles of the CSHEAR

In addition the convexity and topology of the CSHEAR are checked. The convexity test checks that there are no re-entrant angles in the CSHEAR and the topology test checks that there are no collapsed nodes, i.e. that the four nodes of the CSHEAR have different coordinates.

The definitions of the CSHEAR distortion parameters corresponds to the definitions of the CQUAD4. See **CQUAD4 Shape Verifications** (p. 129).

2.20.6 CTRIAX6 Shape Verifications

The following characteristics of the shape of the element CTRIAX6 are calculated and checked:

- CTRIAX6 Aspect Ratio
- CTRIAX6 Skew Angle
- CTRIAX6 Hoe Normal Offset
- CTRIAX6 Hoe Tangent Offset

The first 2 parameters: aspect ratio and skew angle are calculated using only the corner nodes. Their definitions are the same as those given for **CTRIA3 Shape Verifications** (p. 126). The hoe normal offset and the hoe tangent offset are defined next:

CTRIAX6 Hoe Normal Offset (HNO)

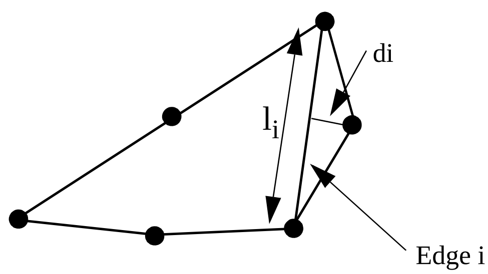
The hoe normal offset of an edge of the CTRIAX6 is defined as the ratio between the midside node's perpendicular offset distance and the distance between the corner nodes of the edge. The hoe normal offset of the CTRIAX6 is defined as the maximum of the hoe normal of its three edges.

i.e.

$$HNO = \underset{i}{MAX}(HNE_i) \qquad i = 1, 2, 3 \tag{2-149}$$

where HNE_i = HOE normal offset of edge i

$$HNE_i = \frac{d_i}{l_i} \tag{2-150}$$



Normal Offset of an edge of CTRIAX6

CTRIAX6 Hoe Tangent Offset (HTO)

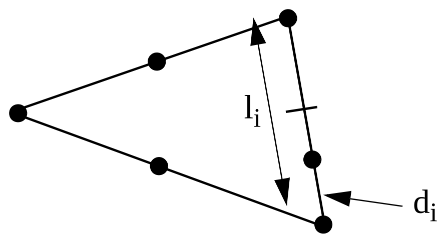
The hoe tangent offset of an edge of the CTRIAX6 is defined as the ratio of the distance between the real and ideal midside node location and the distance between the corner nodes of the edge. If the midside node has a normal offset different than zero, then its projection on the line defined by the two corner nodes is used in the calculations. The hoe tangent offset of the CTRIAX6 is defined as the maximum of the hoe tangent offset of its three edges.

i.e.

$$HTO = \underset{i}{MAX}(HTE_i) \qquad i = 1, 2, 3 \tag{2-151}$$

where HTE_i = HOE tangent offset of edge i

$$HTE_i = \frac{0.5L_i - d_i}{L_i} \tag{2-152}$$



Tangent Offset of an edge of TRIAX6

2

2.20.7 CTETRA Shape Verifications

The following characteristics of the shape of the CTETRA element are calculated and checked by *GENESIS*:

- CTETRA Aspect Ratio
- FACE Skew Angle of CTETRA
- CTETRA Collapse
- CTETRA Edge Angle

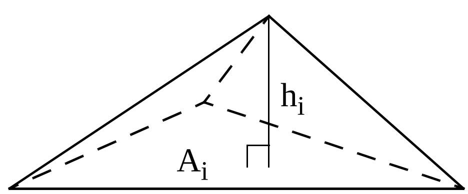
If the CTETRA has 10 nodes, two additional checks are performed:

- CTETRA Hoe Normal Offset
- CTETRA Hoe Tangent Offset

CTETRA Aspect Ratio (AR)

The aspect ratio of a CTETRA element is calculated as the maximum of four values. Each value corresponds to the ratio between the vertical height of a vertex and the square root of the opposing face area. With this definition the A.R. of an equilateral CTETRA is 1.0.

$$AR = \underset{i}{MAX}\left(\frac{h_i}{\sqrt{A_i}}\right) \qquad i = 1, 2, 3, 4 \tag{2-153}$$



Face Skew Angle of CTETRA (FSA)

The face skew angle of a CTETRA is defined as the maximum skew angle among the four triangular faces of the CTETRA. Each face is treated as a CTRIA3 elements.

i.e.

$$FSA = \underset{i}{MAX}(SAF_i) \qquad i = 1, 2, 3, 4, 5, 6 \tag{2-154}$$

where SAF_i = is the SKEW ANGLE of face i, with face i treated as a CTRIA3.

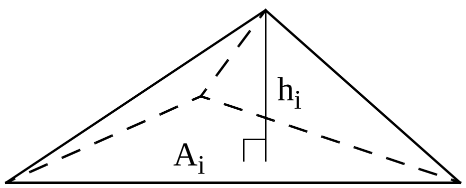
CTETRA Collapse (C)

The collapse parameter of a CTETRA element is calculated as the minimum of four values. Each value corresponds to the ratio between the vertical height of a vertex and the square root of the opposing face area. With this definition the collapse of an equilateral CTETRA is 1.0, and the collapse value would be 0.0 if the CTETRA is collapsed to a triangle.

i.e.

$$C = \underset{i}{\mathrm{MIN}}\left(\frac{h_i}{\sqrt{A_i}}\right) \qquad i = 1, 2, 3, 4 \tag{2-155}$$

(2-156)



CTETRA Edge Angle (EA)

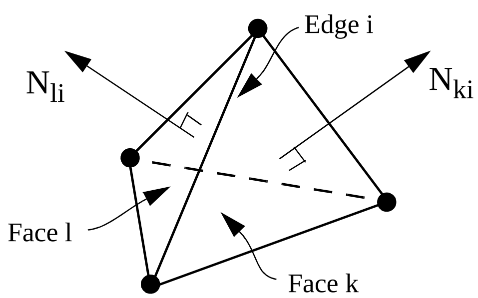
An edge angle is the absolute value of the angle between two faces meeting at an edge subtracted from 90 degrees. The tetra edge angle is defined as the maximum of the six edge angles.

i.e.

$$EA = \left|90 - \mathrm{MAX}(EA_i)\right| \qquad i = 1, 2, 3, 4, 5, 6 \tag{2-157}$$

where EA_i = ANGLE (Nk_i, Nl_i)

Nk_i and Nl_i are the normal vectors of face k and l that have a common edge i.



CTETRA Hoe Normal Offset (HNO)

The hoe normal offset of an edge of the 10 node CTETRA is defined as the ratio between the midside node's perpendicular offset distance and the distance between the corner nodes of the edge. The hoe normal offset of the CTETRA is defined as the maximum of the hoe normal of its six edges.

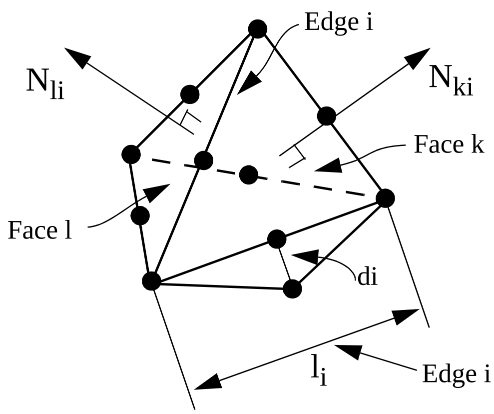
i.e.

$$HNO = \underset{i}{MAX}(HNE_i) \quad i = 1, 2, ..., 6 \tag{2-158}$$

where HNE_i = HOE normal offset of edge i

$$HNE_i = \frac{d_i}{l_i} \tag{2-159}$$

.



CTETRA Hoe Tangent Offset (HTO)

The hoe tangent offset of an edge of the CTETRA is defined as the ratio of the distance between the real and ideal midside node location and the distance between the corner nodes of the edge. If the midside node has a normal offset different than zero, then its projection on the line defined by the two corner nodes is used in the calculations. The hoe tangent offset of the CTETRA is defined as the maximum of the hoe tangent offset of its six edges.

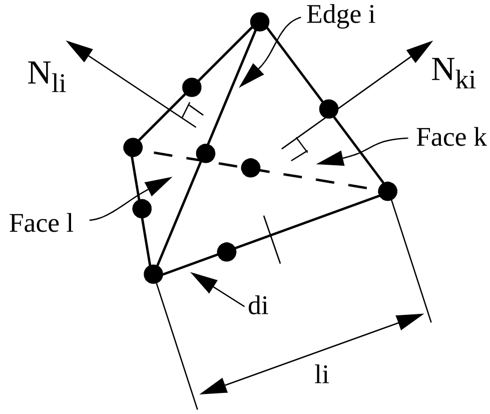
i.e.

$$HTO = \underset{i}{MAX}(HTE_i) \quad i = 1, ..., 6 \tag{2-160}$$

where HTE_i = HOE tangent offset of edge i

$$HTE_i = \frac{0.5L_i - d_i}{L_i} \tag{2-161}$$

.



2

2.20.8 CPENTA Shape Verifications

The following characteristics of the shape of the CPENTA element are calculated and checked:

- CPENTA Aspect Ratio
- Face Skew Angle of CPENTA
- Face Taper of CPENTA
- Face Warp Angle of CPENTA
- CPENTA Twist Angle
- CPENTA Edge Angle

If the CPENTA has 15 nodes, two additional checks are performed:

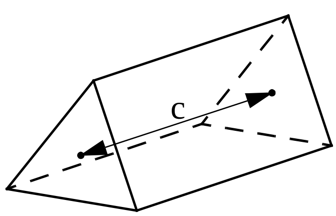
- CPENTA Hoe Normal Offset
- CPENTA Hoe Tangent Offset

In addition, *GENESIS* checks the convexity of the quadrilateral faces and their topology. The convexity test checks that there are no re-entrant angles in any of the three quadrilateral faces of the CPENTA and the topology test checks that there are no collapsed nodes, i.e. that the six nodes of the CPENTA have different coordinates.

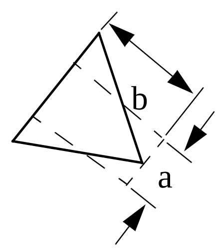
CPENTA Aspect Ratio

The calculation of the aspect ratio of a CPENTA requires several steps: First, calculate an average triangle from the two triangular faces of the CPENTA. Second, considering the aspect ratio of the average triangle as a CTRIA3 two values are obtained: a and b for example. Third, calculate the distance between the centroids of the triangular faces (c). Finally, the CPENTA aspect ratio is calculated as:

$$AR = \frac{MAX(a, b, c)}{MIN(a, b, c)} \tag{2-162}$$



CPENTA



Average Triangle

Face Skew Angle of CPENTA

The face skew angle of a CPENTA is defined as the maximum skew angle among its three quadrilateral faces and 2 triangular faces. Each triangular face is treated as a CTRIA3 element and each quadrilateral face as a warped CQUAD4 element.

i.e.

$$FSA = \underset{i}{MAX}(SAF_i) \qquad i = 1, 2, 3, 4, 5 \tag{2-163}$$

where SAF_i is the SKEW ANGLE of face i, with face i treated as a CQUAD4 or CTRIA3.

2

Face Taper of CPENTA

The face taper of a CPENTA is defined as the maximum taper among the three quadrilateral faces of the CPENTA. Each face is treated as a warped CQUAD4.

i.e.

$$FT = \underset{i}{MAX}(TF_i) \qquad i = 1, 2, 3 \tag{2-164}$$

where TF_i is the TAPER of face i, with face i treated as a CQUAD4.

Face Warp Angle of CPENTA

The face warp angle of a CPENTA is defined as the maximum warp angle among the three quadrilateral faces of the CPENTA. Each face is treated as a warped CQUAD4.

i.e.

$$FWA = \underset{i}{MAX}(WAF_i) \qquad i = 1, 2, 3 \tag{2-165}$$

where WAF_i is the warp angle of face i, with face i treated as a CQUAD4.

CPENTA Twist Angle

The twist angle is defined as the rotation of one triangular face with respect to the opposite triangular face. To compute the twist angle, a reference axis is generated between the centroids of the two triangular faces. The three edges of each triangular face are projected onto a plane whose normal is parallel to the reference axis. The maximum angle between the corresponding edges of the two projected triangles is the CPENTA twist angle.

i.e.

$$TA = \underset{i}{MAX}(AE_1, AE_2, AE_3) \quad i = 1, 2, 3 \tag{2-166}$$

where AEi = ANGLE (EDGE i - EDGE i’)

CPENTA Edge Angle

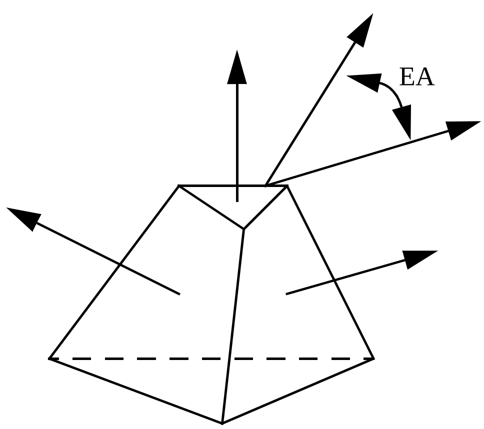
An edge angle is the absolute value of the angle between two faces meeting at an edge subtracted from 90 degrees. For warped faces, the projected planes for each face are used to compute the face normals used in the angle calculation. The penta edge angle is defined as the maximum edge angle in the CPENTA.

i.e.

$$EA = |90 - MAX(EA_i)| \quad i = 1, 2, ..., 9 \tag{2-167}$$

where EA_i = ANGLE (Nk_i, Nl_i)

Nk_i and Nl_i are the normal vector of face k and l that have a common edge i.



CPENTA Hoe Normal Offset (HNO) and Hoe Tangent Offset (HTO)

The Hoe normal and Hoe Tangent offsets are defined for the CPENTA are defined in the same way as that for the 10 noded CTETRA with the only difference being that CPENTA has 9 edges.

2.20.9 CHEXA/CHEX20 Shape Verifications

The following characteristics of the shape of the hexahedron CHEXA element are calculated and checked:

- CHEXA Aspect Ratio

- Face Skew Angle of CHEXA

- Face Taper of CHEXA

- Face Warp Angle of CHEXA

- CHEXA Twist Angle

- CHEXA Edge Angle

For 9-21 noded CHEXA elements, the following checks are also performed:

- CHEXA Hoe Normal Offset

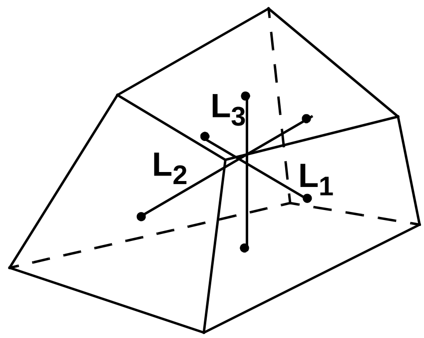
- CHEXA Hoe Tangent Offset

In addition *GENESIS* checks the convexity and topology of each face based on the four corner nodes. The convexity test checks that there are no re-entrant angles in any of the six faces of the CHEXA and the topology test checks that there are no collapsed nodes, i.e. checks that the eight nodes of the CHEXA have different coordinates.

CHEXA Aspect Ratio

The aspect ratio of a CHEXA element is calculated as the quotient between the maximum distance between centroids of opposite faces and the minimum distance between centroid of opposite faces.

$$AR = \frac{MAX(L_1, L_2, L_3)}{MIN(L_1, L_2, L_3)}$$



2

Face Skew Angle of CHEXA

The face skew angle of a HEXA is defined as the maximum skew angle among the six faces of the CHEXA. Each face is treated as a CQUAD4 element to permit the inclusion of warped faces. For example:

$$FSA = \underset{i}{MAX}(SAF_i) \quad i = 1, ..., 6 \tag{2-168}$$

where SAF_i is the SKEW ANGLE of face i, with face i treated as a CQUAD4.

Face Taper of CHEXA

The face taper of a CHEXA is defined as the maximum taper among the six faces of the CHEXA. Each face is treated as a warped CQUAD4. For example:

$$FT = \underset{i}{MAX}(TF_i) \quad i = 1, ..., 6 \tag{2-169}$$

where TF_i is the TAPER of face i, with face i treated as a CQUAD4.

Face Warp Angle of CHEXA

The face warp angle of a CHEXA is defined as the maximum warp angle among the six faces of the CHEXA. Each face is treated as a warped CQUAD4. For example:

$$FWA = \underset{i}{MAX}(WAF_i) \quad i = 1, ..., 6 \tag{2-170}$$

where WAF_i is the warp of face i, with face i treated as a CQUAD4.

CHEXA Twist ANGLE

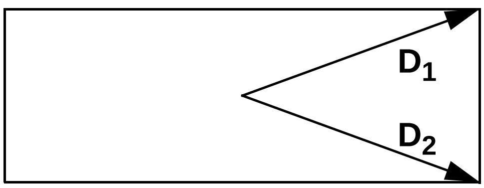
The twist angle is defined as the maximum rotation of one face with respect to its opposite face. To explain how the rotation of each face is calculated two auxiliary vectors D1 and D2 (the Diagonal Vectors) are first defined:

D1 =0.25*(PL1+PL2)

D2 =0.25*(Pl1-PL2)

where PL1 and PL2 are the plane vectors associated with a face of the CHEXA treated as a CQUAD4.

The diagonal vectors for a special case of a flat rectangular face are shown in the figure below:



Then, for each pair of opposing faces a reference plane that is perpendicular to the axis through the center of the faces is constructed. The Diagonal vectors D1 and D2 of each of the two opposite faces are projected onto the plane. The difference between the projections of the D1 vectors and the difference between the D2 vectors are calculated as 1 and 2. The maximum between 1 and 2 corresponds to the rotations of a face with respect to its opposite face. Finally, the twist angle of the CHEXA is calculate as the maximum relative rotation of its faces. For example:

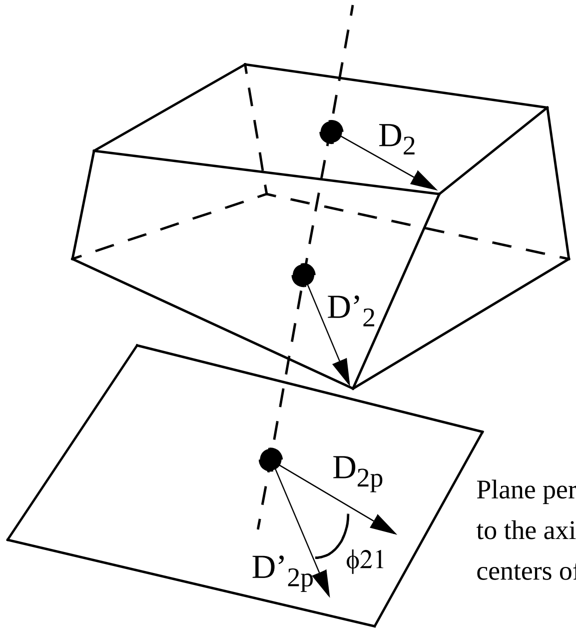
TA = MAX { TAF_1, TAF_2, TAF_3 }

where:

TAFi = MAX { 1, 2 }   i=1,2,3

j = ANGLE { Djp, Djp’}   j=1,2

Djp and Djp are the projection of the diagonal vectors Dj and Dj’ onto the auxiliary plane.



2

CHEXA Edge Angle

An edge angle is the absolute value of the angle between two faces meeting at an edge subtracted from 90 degrees. For warped faces, the projected planes for each face are used to compute the face normals used in the angle calculation. The hexa edge angle is defined as the maximum edge angle in the CHEXA. For example:

$$EA = \left|90 - MAX(EA_i)\right| \qquad i = 1, 2, ..., 12 \qquad (2\text{-}171)$$

where EAi = ANGLE (Nki, Nli).

Nki and Nli are the normal vectors of face k and l that have a common edge i.

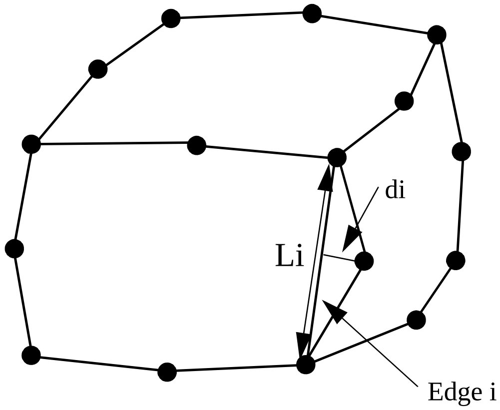
CHEXA Hoe Normal Offset (HNO)

The hoe normal offset of an edge of the CHEXA is defined as the ratio between the midside node's perpendicular offset distance and the distance between the corner nodes of the edge. The hoe normal offset of the CHEXA is defined as the maximum of the hoe normal of its twelve edges. For example:

$$HNO = \underset{i}{MAX}(HNE_i) \quad i = 1, 2, ..., 12 \tag{2-172}$$

where HNE_i = HOE normal offset of edge i

$$HNE_i = \frac{d_i}{L_i} \tag{2-173}$$



Normal Offset of an edge of CHEXA

2

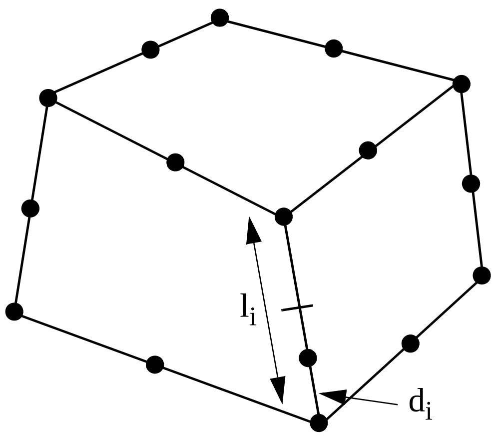
CHEXA Hoe Tangent Offset (HTO)

The hoe tangent offset of an edge of the CHEXA is defined as the ratio of the distance between the real and ideal midside node location and the distance between the corner nodes of the edge. If the midside node has a normal offset different than zero, then its projection on the line defined by the two corner nodes is used in the calculations. The hoe tangent offset of the CHEXA is defined as the maximum of the hoe tangent offset of its twelve edges. For example:

$$\mathrm{HTO} = \underset{i}{\mathrm{MAX}}(\mathrm{HTE}_i) \qquad i = 1, ..., 12 \tag{2-174}$$

where HTE_i = HOE tangent offset of edge i

$$\mathrm{HTE}_i = \frac{0.5L_i - d_i}{L_i} \tag{2-175}$$



Tangent Offset of an edge of CHEXA

2.20.10 CPYRA Shape Verifications

The following characteristics of the pyramid element CPYRA are calculated and checked:

- CPYRA Aspect Ratio

- CPYRA Face Skew Angle

- CPYRA Vertex Angle

- CPYRA Face Warp Angle

For 13 noded CPYRA elements, the following checks are also performed

- CPYRA Hoe Normal Offset

- CPYRA Hoe Tangent Offset

In addition, *GENESIS* checks the convexity of the quadrilateral faces and their topology. The convexity test checks that there are no re-entrant angles in the quadrilateral face of the CPYRA and the topology test checks that there are no collapsed nodes, i.e. that the five or thirteen nodes of the CPYRA have different coordinates.

CPYRA Aspect Ratio

The aspect ratio of a CPYRA element is defined as the maximum of the aspect ratios of its quadrilateral base and four triangular faces. The quadrilateral and triangular faces are treated as CQUAD4 and CTRIA3 elements respectively.

CPYRA Face Skew Angle

The face skew angle of a CPYRA element is defined as the maximum of the skew angles of its quadrilateral base and four triangular faces. The quadrilateral and triangular faces are treated as CQUAD4 and CTRIA3 elements respectively.

CPYRA Vertex Angle

The maximum and minimum vertex angles of all faces of a CPYRA element are calculated by treating the quadrilateral face as a CQUAD4 element and the triangular faces as CTRIA3 elements.

CPYRA Face Warpage Angle

The face warpage angle of a CPYRA element is defined as the warpage angle of its quadrilateral base which is calculated by considering the base as a CQUAD4 element.

CPYRA Hoe Normal Offset

The definition is the same as that of all second order solid elements (CHEXA, CTETRA and CPENTA)

CPYRA Hoe Tangent Offset

The definition is the same as that of all second order solid elements (CHEXA, CTETRA and CPENTA)

CHAPTER 3

Input Data Description

3

3

3.1 Overview

The purpose of this part of the manual is to provide you with the details necessary to create *GENESIS* input data. Another manual that you may wish to reference is:

- Analysis Example Problems

Data to execute the program is segmented into the following three parts:

1. EXECUTIVE CONTROL
2. SOLUTION CONTROL
3. BULK DATA

The data for each of these functions are described in this volume.

GENESIS analysis is based on the finite element analysis method, where the general structure is modeled as an assemblage of idealized parts or elements. These elements are connected at grid points. Different elements may have different shapes and material properties. Loads are applied at the grid points or, in some cases, to the elements themselves.

3.2 Executive Control

The executive control portion of the input data is used to control the overall program flow, data checking, and diagnostic printing. The following keywords are detailed in **Executive Control** (p. 181):

ID, **DIRALL**, **DIRDAF**, **DIRSAF**, **DIRSMS**, **EIGMETHOD**, **ESLCONF**, **ESLDISP**, **IOBUFF**, **K2UU**, **K2UU1**, **M2UU**, **M2UU1**, **CHECK**, **REDUCE**, **SOL**, **POST**, **THREADS**, **DIAG**, **UFDATA**, **CEND**

Only the first four characters of each keyword need be used.

Comments are allowed and are indicated by a “$” as the first character.

3.3 Solution Control

The solution control section of the *GENESIS* input data is used to set up the various load cases for the design problem. There are static, frequency calculation, buckling load factor calculation, heat transfer, direct dynamic response and modal dynamic response load cases. In addition there is the capability for static load case combinations, called loadcoms, which are a linear combination of static load cases.

In the static load cases the user specifies the boundary conditions, SPCs and MPCs, as well as the point, pressure, thermal, centrifugal deformation and gravity loadings. In general, the user can request that some, all, or none of the applied loads, displacements, reaction forces, grid point stresses, element stresses, element strains, element forces and element strain energies be written to the output file and/or to post processing files.

In the static load case combinations the user can define the scale factor that is to be applied to each static load case as they are added together. Note that a maximum of one of these load cases can contain thermal loads. The user can request that some, all, or none of the displacements, grid point stresses, element stresses, strains and element forces be written to the output file and/or to post processing files.

In the buckling load cases the user references an existing static load case to study the stability of the structure. The boundary conditions, SPCs and MPCs are not selected in the buckling load cases themselves, rather they are specified in the static load cases. In buckling analysis, *GENESIS* will print the buckling load factors to the output file and optionally the user can request that some or all of the buckling mode shapes be printed to the output file or to post processing files.

In the frequency calculation load cases the user specifies the boundary conditions, SPCs and MPCs, as well as the frequency calculation method. The user can also request that some, all, or none of the eigenvectors be written to the output file. In addition, the user can specify that element strain energy is output for all elastic elements for all modes or for selected modes.

In heat transfer loadcases, the user specifies boundary conditions, SPC's and MPC's, as well as the thermal loading. The user can request that grid point temperatures be written to the output and/or post processing file.

In direct dynamic response, the user specifies the boundary conditions, SPCs and MPCs, as well as the frequencies of the cyclic loading. The loading can be point, pressure or gravity. In general, the user can request that some, all or none of the displacements, velocities, accelerations, grid point stresses, element stresses, strains and forces will be written to the output file and/or post processing files.

In modal dynamic response, the user specifies a frequency calculation load case for modal reduction of the problem, and the boundary condition, SPCs and MPCs. The loading can be point, pressure or gravity, and is applied at frequencies defined by the user. In general, the user can request that some, all or none of the displacements, velocities, accelerations, grid point stresses, element stresses, strains and forces will be written to the output file and/or post processing files.

Each load case begins with the delimiter **LOADCASE** #, where # is the load case number. Each static load case combination begins with the delimiter **LOADCOM** #, where # is the combination load case number.

The LOADCASE and LOADCOM numbers must be strictly increasing, but not necessarily inclusive, i.e.,

```
LOADCASE 1
     .
     .
LOADCASE 2
     .
     .
LOADCASE 6
     .
     .
LOADCASE 9
```

3

3.4 Bulk Data

This section describes the format of the bulk data for *GENESIS*.

The bulk data section of an input data file begins after the **BEGIN BULK** delimiter and ends at the **ENDDATA** entry.

Any data in the input file following the ENDDATA entry are ignored.

A bulk data entry consists of one or more logical lines. Each logical line consists of ten fields. A logical line may be entered in one of three formats:

- Single field
- Free field
- Double field.

Each single field or free field logical line consists of one physical line in the input data. A double field logical line consists of two physical lines in the input data. Free field is indicated by at least one comma (,) in columns 1 through 8 of the line. Double field is indicated by an asterisk (*) as one of the first eight characters.

For single field lines, each field is eight columns.

Free field lines have fields separated by commas. Each field may have no more than 16 characters. Blank spaces are allowed before or after commas. Each free field line should have no more than eighty characters in total.

A double field entry consists of two input lines, each with six fields of 8, 16, 16, 16, 16, 8 columns, respectively. The last field of the first input line is ignored. The first field of the second input line must begin with an asterisk.

For example, the following data could be entered in each of the formats as below:

1	2	3	4	5	6	7	8	9	10
GRID	10		1.3217	9.3E+2	11.009				

Single field:

```
1------12------23------34------45------56------67------78------89------90------0
GRID        10              1.3217    9.3E+2  11.009
```

Free field:

```
GRID,10,,1.32174631,9.32135E+2,11.009
```

Double field:

```
1------12--------------23--------------34--------------45--------------5X------X
X------X6--------------67--------------78--------------89--------------90------0
GRID*           10                              1.32174631      9.32135E+2
*                11.009
```

Note that fields do not have to be right or left justified.

Each field of a logical line contains data that is classified as integer, character or real. Real data <u>must</u> have a decimal point and can be written as: 10.0, 1.+1, 100.E-1, 1.0E1, etc. Integer and character data can have no more than eight digits/non-blank characters, even on free field or double field lines.

The second, and subsequent, lines in a bulk data entry must have a “+” (for single field or free field) or a “*” (for double field) as the first non-blank character in field 1.

Different logical lines in a bulk data entry may have different formats.

For example

```
         1         2         3         4         5         6         7         8
12345678901234567890123456789012345678901234567890123456789012345678901234567890
GENEL         99               2       1       2       2       2       3
+              2       4       2       5       2       6
+,UD,,3,1,3,2,3,3
+,3,4,3,5,3,6
*                      Z    3.2043875E-6             0.0             0.0
*                    0.0             0.0             0.0    1.2226743E-6
*                    0.0             0.0             0.0    4.3848729E-7
*           1.3318754E-7             0.0   -2.2109642E-7             0.0
*           5.1842855E-7             0.0             0.0    4.6726853E-6
*                    0.0    4.5446227E-6
```

Lines in the bulk data section that have a “$” character as the first non-blank character are taken as comment lines and are ignored.

```
$ THIS IS A COMMENT LINE. THIS LINE IS IGNORED.
```

Completely blank lines are also allowed.

3.5 Analysis Model Data

The analysis model defines the finite element model on which the design is based. This section identifies the various bulk data statements which define the geometry, coordinates, loads, etc.

3.5.1 Geometry

Grid Points

DATA	INFORMATION	PAGE
GRID	Grid point location, coordinate system selection	524
GRDSET	Default options for GRID statements	523

Scalar Points

DATA	INFORMATION	PAGE
SPOINT	Scalar point list	695

Coordinate Systems

DATA	INFORMATION	PAGE
CORD1C	Cylindrical coordinate system definition based on GRID data	439
CORD2C	Cylindrical coordinate system definition based on user supplied points	445
CORD1R	Rectangular coordinate system definition based on GRID data	441
CORD2R	Rectangular coordinate system definition based on user supplied points	447
CORD1S	Spherical coordinate system definition based on GRID data	443
CORD2S	Spherical coordinate system definition based on user supplied points	449

3.5.2 Elements

Elastic Line Elements

DATA	INFORMATION	PAGE
BAROR	Default for orientation and property for CBAR	378
BEAMOR	Default for orientation and property for CBEAM	384
CBAR	Connection definition for uniform bar element	393
CBEAM	Connection definition for tapered beam element	396
PBAR	Property definition for CBAR	571
PBARL	Property definition for CBAR using dimension	574
PBEAM	Property definition for CBEAM	584
PBEAML	Property definition for CBEAM using dimension	589
CROD	Connection definition for rod with axial stiffness	462
PROD	Property definition for CROD	644
CTUBE	Connection definition for tube element with axial stiffness	474
PTUBE	Property definition for CTUBE (to be converted to PROD)	654

Elastic Surface Elements

DATA	INFORMATION	PAGE
CTRIA3	Connection definition for a triangle with bending and membrane stiffness	467
CQUAD4	Connection definition for a quadrilateral with bending and membrane stiffness	457
CTRIA6	Connection definition for a curved triangle with bending and membrane stiffness	469
CQUAD8	Connection definition for a curved quadrilateral with bending and membrane stiffness	459
PSHELL	Property definition for homogeneous CTRIA3, CTRIA6, CQUAD4 and CQUAD8 elements	648

PCOMP	Property definition for composite CTRIA3, CTRIA6, CQUAD4 and CQUAD8 elements	604
PCOMPG	Property definition for composite CTRIA3, CTRIA6, CQUAD4 and CQUAD8 elements	609
CSHEAR	Connection definition for a quadrilateral element with shearing and optional extensional stiffness	463
PSHEAR	Property definition for CSHEAR	646

Elastic Axisymmetric Elements

DATA	INFORMATION	PAGE
CTRIAX6	Connection definition for axisymmetric 6-noded triangle element	472
PAXIS	Property definition for CTRIAX6 element	565

Elastic Solid Elements or Fluid Elements

DATA	INFORMATION	PAGE
CTETRA	Connection definition for four-sided solid with four or ten grid points	465
CPENTA	Connection definition for five-sided solid with six or fifteen grid points	451
CHEXA	Connection definition for six-sided solid with 8 to 21 grid points	428
CHEX20	Connection definition for six-sided solid with 8 to 21 grid points	426
CPYRA	Connection definition for five-sided solid with 5 or 13 grid points	454
PSOLID	Property definition for CTETRA, CPENTA, CHEXA, CHEX20 and CPYRA elements.	652

Bushing Element

DATA	INFORMATION	PAGE
CBUSH	Connection definition for bushing element	400
PBUSH	Property definition for a generalized spring-damper	600

Weld Element

DATA	INFORMATION	PAGE
CWELD	Connection definition for weld element	479
PWELD	Property definition for a generalized connector	658

Gap Element

DATA	INFORMATION	PAGE
CGAP	Connection definition for gap element	411
PGAP	Property definition for gap elements	620

Elastic Scalar Elements

DATA	INFORMATION	PAGE
CELAS1	Connection definition for scalar spring	407
CELAS2	Connection and property definition for scalar spring	408
CELAS3	Connection definition for scalar spring on scalar points only	409
CELAS4	Connection and property definition for scalar spring on scalar points only	410
PELAS	Property definition for CELAS1	618

Elastic Vector Element

DATA	INFORMATION	PAGE
CVECTOR	Connection definition for vector spring	475
PVECTOR	Property definition for CVECTOR	655

General Element

DATA	INFORMATION	PAGE
GENEL	Defines the connection and stiffness or flexibility of a general element. Optionally defines the associated rigid body matrix	518

Interpolation Elements

DATA	INFORMATION	PAGE
BOLT	Defines multipoint constraints to connect the top and bottom sections of a bolt model with a bolt control grid.	386
RBE3	Defines the displacement at a reference grid as a weighted average of the displacements of a set of other grids	677
RSPLINE	Defines a multipoint constraint of grids as an interpolation using a beam like equation	688

Rigid Elements

DATA	INFORMATION	PAGE
RBAR	Defines rigid bar with six degrees of freedom at each end	672
RBE1	Defines rigid body connection to an arbitrary number of grid points	674
RBE2	Defines rigid body connected to an arbitrary number of grid points	676
RROD	Defines pin-ended rigid rod	687

User Supplied Stiffness Elements

DATA	INFORMATION	PAGE
K2UU	Executive control command to load a file containing a user-supplied stiffness matrix	199
K2UU1	Executive control command to load a file containing a scalable user-supplied stiffness matrix	200
PK2UU	Defines the properties of K2UU1	623

Mass Elements

DATA	INFORMATION	PAGE
CMASS1	Connection definition for a scalar mass element	431
CMASS2	Connection and property definition of a scalar mass element	432
CMASS3	Connection definition for a scalar mass element on scalar points only	433
CMASS4	Connection and property definition of a scalar mass element on scalar points only	434
CONM2	Defines concentrated mass at a grid point	435
CONM3	Defines the location of a concentrated mass at a grid point	437
PCONM3	Defines the properties of CONM3	614
PMASS	Defines the properties of CMASS1	641

User Supplied Mass Elements

DATA	INFORMATION	PAGE
M2UU	Executive control command to load a file containing a user-supplied mass matrix	203
M2UU1	Executive control command to load a file containing a scalable user-supplied mass matrix	204
PM2UU	Defines the properties of M2UU1	642

Damping Elements

Viscous Damping Elements

DATA	INFORMATION	PAGE
CBUSH	Connection definition for a bushing element	400
CDAMP1	Connection definition for a scalar damping element	403
CDAMP2	Connection and property definition for a line damper with extensional and rotational damping	404
CDAMP3	Connection definition for a scalar damping element on scalar points only	405
CDAMP4	Connection and property definition for a scalar damping element on scalar points only	406
CVISC	Connection definition for a line damper with extensional and rotational damping	478
PBUSH	Property definition for a generalized spring-damper	600
PDAMP	Property definition for CDAMP1 element	617
PVISC	Property definition for CVISC element	658

Structural Damping Elements

The elastic elements CELAS1, CELAS2, CELAS3, CELAS4, CBUSH, CVECTOR, CWELD, CROD/CTUBE, CBAR, CBEAM, CTRIA3, CQUAD4, CTRIA6, CQUAD8, CSHEAR, CTRIAX6, CTETRA, CPENTA, CHEXA, CHEX20 and CPYRA can also be used to add structural damping to the structure (using the GE coefficient in the MAT1, MAT2, MAT3, MAT8, MAT9, MAT11, PELAS, PBUSH, PCOMP, PVECTOR and CELAS2 data).

Heat Transfer Elements

Heat Boundary Elements

DATA	INFORMATION	PAGE
BDYOR	Default values for CHBDYG and CHBDYP	383
CHBDY	Defines a heat transfer boundary element for thermal flux and convection loads (available types of area are “POINT”, “LINE”, “AREA3”, “AREA4” and “ELCYL”)	416
CHBDYE	Defines a heat transfer boundary element for thermal flux and convection loads (by referring to a heat conduction element)	419
CHBDYG	Defines a heat transfer boundary element for thermal flux and convection loads (available types of area are “AREA3”, “AREA4”, “AREA6” and “AREA8”)	423
CHBDYP	Defines a heat transfer boundary element for thermal flux and convection loads (available types of area are “POINT”, “LINE”, “ELCYL” and “TUBE”)	424
CONV	Defines a convection boundary condition	438
PCONV	Defines the properties for convection boundary condition	616
PHBDY	Property definition for CHBDY or CHBDYP elements	621

Heat Conduction elements

The elastic elements CELAS1, CELAS2, CELAS3, CELAS4, CROD/CTUBE, CBAR, CBEAM, CTRIA3, CQUAD4, CTRIA6, CQUAD8, CTRIAX6, CTETRA, CPENTA, CHEXA, CHEX20 and CPYRA are also conduction elements.

The CGLUE connection acts as a perfect heat conductor.

3.5.3 Materials

Isotropic

DATA	INFORMATION	PAGE
MAT1	Defines elastic material properties for isotropic elements	530

Anisotropic

DATA	INFORMATION	PAGE
MAT2	Defines anisotropic material properties for two-dimensional elements	533
MAT3	Defines orthotropic material properties for axisymmetric elements	535
MAT8	Defines orthotropic material properties for two-dimensional elements	540
MAT9	Defines anisotropic material properties for solid elements	542
MAT11	Defines orthotropic material properties for solid elements	545

Heat Transfer

DATA	INFORMATION	PAGE
MAT4	Defines thermal material properties for isotropic elements	537
MAT5	Defines thermal material properties for anisotropic two-dimensional, axisymmetric and solid elements	538

Fluid

DATA	INFORMATION	PAGE
MAT10	Defines material properties for fluid elements	544

3.5.4 Nonstructural Mass

(Activated by **NSM** = SID)

DATA	INFORMATION	PAGE
NSM	Select nonstructural mass per unit area or length for **PSHELL**, **PCOMP**, **PSHEAR**, **PBAR**, **PBARL**, **PBEAM**, **PBEAML** or **PROD**	553
NSM1	Select nonstructural mass per unit area or length for **PSHELL**, **PCOMP**, **PSHEAR**, **PBAR**, **PBARL**, **PBEAM**, **PBEAML** or **PROD**	554
NSML	Select nonstructural lumped mass for **PSHELL**, **PCOMP**, **PSHEAR**, **PBAR**, **PBARL**, **PBEAM**, **PBEAML** or **PROD**	557
NSML1	Select nonstructural lumped mass for **PSHELL**, **PCOMP**, **PSHEAR**, **PBAR**, **PBARL**, **PBEAM**, **PBEAML** or **PROD**	558
NSMADD	Define a union of NSM, NSM1, NSML and NSML1sets	556

3.5.5 Boundary Conditions

Single-point Constraints

(Activated by **SPC** = SID)

DATA	INFORMATION	PAGE
SPC	Defines single-point constraints	690
SPC1	Defines single-point constraints	691
SPCADD	Define a union of SPC/SPC1 sets	693

Multi-point Constraints

(Activated by **MPC** = SID)

DATA	INFORMATION	PAGE
MPC	Defines a linear relationship for two or more degrees of freedom	549
MPCADD	Define a union of MPC sets	551

Multi-point constraints are also generated by the rigid and interpolation elements: RBAR, RBE1, RBE2, RBE3, RROD, RSPLINE and BOLT. Multi-point constraints generated by these elements apply to all structural load cases. Rigid and interpolation elements are ignored in heat transfer analysis.

Enforced Displacement or Temperature

(Activated by **LOAD**=SID1 and **SPC**=SID2, or **HEAT** = SID1 and **SPC** = SID2)

DATA	INFORMATION	PAGE
SPC	Defines value for enforced displacement or temperature	690
SPCD	Defines value for enforced displacement or temperature	694

Guyan Reduction Degrees of Freedom

(Activated by **ASET**=SID)

DATA	INFORMATION	PAGE
ASET2	Defines a set of free degrees of freedom used for Guyan Reduction in a frequency loadcase or boundary degrees of freedom for superelement reduction	376
ASET3	Defines a set of free degrees of freedom used for Guyan Reduction in a frequency loadcase or boundary degrees of freedom for superelement reduction	377

Craig-Bampton Modes

(Activated by **QSET**=SID)

DATA	INFORMATION	PAGE
QSET2	Defines a set of generalized degrees of freedoms for Craig-Bampton modes in a Guyan Reduction loadcase	663
QSET3	Defines a set of generalized degrees of freedoms for Craig-Bampton modes in a Guyan Reduction loadcase	664

Support (Reference) Degrees of Freedom

(Activated by **SUPORT**=SID)

DATA	INFORMATION	PAGE
SUPORT1	Defines reference degrees of freedom in a free body (Inertia Relief) analysis	696

3.5.6 Loads

Static Loads

Concentrated Static Loads

(Activated by **LOAD**=SID)

DATA	INFORMATION	PAGE
FORCE	Defines concentrated load at grid point	513
FORCE1	Defines concentrated load at grid point	514
LOAD	Defines a linear combination of loads	528
MOMENT	Defines moment at grid point	547
MOMENT1	Defines moment at grid point	548

Distributed Static Loads

(Activated by **LOAD**=SID)

DATA	INFORMATION	PAGE
PLOAD1	Defines distributed or point load on BAR and BEAM elements	624
PLOAD2	Defines normal pressure loads on surface (TRIA3, QUAD4 and SHEAR) elements	628
PLOAD4	Defines pressure loads on surfaces of HEXA, PENTA, HEX20, TETRA, PYRA, TRIA3, QUAD4 and SHEAR elements	630
PLOAD5	Defines pressure loads on surface (TRIA3, QUAD4 and SHEAR) elements	634
PLOADA	Defines distributed load on BAR and BEAM elements	636
PLOADX1	Defines pressure loads on axisymmetric elements (CTRIAX6)	638

Temperature Loads

(Activated by **TEMPERATURE**=SID)

DATA	INFORMATION	PAGE
TEMP	Defines temperature at grid points	707
TEMPD	Specifies default temperature at all grid points	708

Gravity Load

(Activated by **GRAVITY**=SID)

DATA	INFORMATION	PAGE
GRAV	Defines gravity load vector	521

Centrifugal Load

(Activated by **CENTRIFUGAL**=SID)

DATA	INFORMATION	PAGE
RFORCE	Defines a centrifugal load	679

Deform Load

(Activated by **DEFORM**=SID)

DATA	INFORMATION	PAGE
DEFORM	Defines a load due to non-elastic initial deformation of a ROD, BAR, or BEAM	487

Equivalent Static Load

(Activated by **ESLOAD**=SID)

DATA	INFORMATION	PAGE
External Files	The information ESLOAD uses is in external files created by third party software. See:: **ESLCONF and ESLDISP**	N.A

3

Dynamic Loads

Dynamic Loads

(Activated by **DLOAD**=SID)

DATA	INFORMATION	PAGE
DLOAD	Defines a linear combination of dynamic loads	496
RLOAD1	Defines a dynamic load	682
RLOAD2	Defines a dynamic load	684
RLOAD3	Defines a dynamic load at a point	686
DAREA	Defines dynamic loading degree of freedom	486
DELAY	Defines dynamic loading delay time	488
DPHASE	Defines dynamic loading phase lead	499
TABLED1	Defines dynamic loads as a function of frequency	701
TABLED2	Defines dynamic loads as a function of frequency	702
TABLED3	Defines dynamic loads as a function of frequency	703
TABLED4	Defines dynamic loads as a function of frequency	705

Dynamic Loading Frequencies

(Activated by **FREQUENCY**=SID)

DATA	INFORMATION	PAGE
FREQ	Defines a set of dynamic loading frequencies	515
FREQ1	Defines a linear set of dynamic loading frequencies	516
FREQ2	Defines a logarithmic set of dynamic loading frequencies	517

Dynamic Modal Damping

(Activated by **SDAMPING**=SID)

DATA	INFORMATION	PAGE
TABDMP1	Defines modal damping as a function of frequency	699

Random Loads

(Activated by **RANDOM**=SID)

DATA	INFORMATION	PAGE
RANDPS	Defines power spectral densitly load factors	670
RANDT1	Defines time lags for autocorrelation	671
TABRND1	Defines power spectral densitly as a tabular function of frequencies. Referenced by **RANDPS**.	706

Heat Transfer Loads

Heat Generation

(Activated by **HEAT**=SID)

DATA	INFORMATION	PAGE
QVOL	Defines volumetric heat generation for a conductive element	668

Convection

(Activated by **HEAT**=SID1 and **SPC**=SID2)

DATA	INFORMATION	PAGE
CHBDY	Defines ambient temperature points	416
CONV	Defines a convection boundary condition	438

3

Thermal Vector Flux

(Activated by **HEAT**=SID)

DATA	INFORMATION	PAGE
QVECT	Defines thermal flux vector for CHBDY, CHBDYE, CHBDYG, CHBDYP elements	666

Thermal Flux

(Activated by **HEAT**=SID)

DATA	INFORMATION	PAGE
QHBDY	Defines thermal flux into a set of grid points	662
QBDY1	Defines thermal flux into a CHBDY, CHBDYE, CHBDYG, CHBDYP element	660
QBDY2	Defines thermal flux into the grid points of a CHBDY, CHBDYE, CHBDYG, CHBDYP element	661

3.5.7 Problem Control

Frequency Analysis

(Activated by **METHOD** = SID or **CBMETHOD** = SID)

DATA	INFORMATION	PAGE
EIGR	Defines frequency calculation data	500
EIGRL	Defines frequency calculation data for the Lanczos or SMS method	503

Buckling Analysis

3

(Activated by **STATSUB** = LID and **METHOD** = SID)

DATA	INFORMATION	PAGE
EIGR	Defines load factor calculation data	500
EIGRL	Defines frequency calculation data for the Lanczos method	503

3.5.8 Miscellaneous

Parameters

DATA	INFORMATION	PAGE
DISTOR	Override limits for warnings and errors of distortion parameters	489
PARAM	Specific values for analysis parameters	560

CHAPTER 4

Executive Control

- UFDATA

4

4.1 $

Executive Control Entry: **$** - Comment

Description: Enter a comment line.

Format:

$ Any character data

Example:

```
$ This line is a comment.
```

4.2 CEND

Executive Control Entry: **CEND** - Mark the End of Executive Control

Description: Delimits the Executive Control section of the input file from the Solution Control section.

Format:

CEND

Example:

```
CEND
```

Remarks:

1. If the CEND command is not present in the input, the Solution Control will begin at the line containing the first detected Solution Control command.
2. Some commands cannot be unambiguously detected as Solution Control. If the first Solution Control command is one of: DRESP2, **INCLUDE**, **POSTOUTPUT**, SENSITIVITY, SSOL, **TIMES** or TSURF, then the CEND delimiter is required.

4

4.3 CHECK

Executive Control Entry: **CHECK** - Program Flow Control

Description: Stops the program after checking the input data.

Format:

CHECK

Example:

```
CHECK
```

Remarks:

1. Mutually exclusive with **REDUCE**.
2. This can be used to save time when large amounts of input data are being assembled.
3. CHECK can also be specified on the command line using the -check flag. For example.

   ```
   genesis -check mydata.dat
   ```

   If any command mutually exclusive with CHECK appears in the datafile, it takes precedence over any value specified on the command line.

4.4 DIAG

Executive Control Entry: **DIAG** - Program Diagnostic Control

Description: Enables diagnostic printing or alternate algorithms.

Format:

DIAG = n1,n2,n3,...

Example:

```
DIAG=87
```

Option	Meaning
ni	Diagnostic value to set(Integer>0).

Remarks:

1. See **The DIAG Command** (Sec. A.2) for a description of the information printed for supported values of “ni”.
2. Note that the use of undocumented values of “ni” may produce unexpected results.
3. Multiple DIAG commands are allowed.
4. DIAG values can also be specified on the command line using the -diag=*n1,n2,...* flag. For example.

   ```
   genesis -diag=324 mydata.dat
   ```

4

4.5 DIRALL

Executive Control Entry: **DIRALL** - Scratch File Directory Control

Description: Selects a directory for all scratch files.

Format:

DIRALL = directory name

Example:

```
DIRALL = /localscratch
```

Remarks:

1. For decreased I/O times, this directory should be on a disk directly attached to the machine where *GENESIS* is running.
2. This command is equivalent to DIRDAF, DIRSAF and DIRSMS when these commands all use the same directory name.
3. The DIRALL directory can also be specified on the command line using the -dirall=*directory* flag. For example.

   ```
   genesis -dirall=/tmp/scratch mydata.dat
   ```

   If the DIRALL, DIRDAF, DIRSAF or DIRSMS command appears in the datafile, it takes precedence over any value specified on the command line.

4.6 DIRDAF

Executive Control Entry: **DIRDAF** - Scratch File Directory Control

Description: Selects a directory for direct access scratch files.

Format:

DIRDAF = directory name

Example:

```
DIRDAF = /localscratch
```

Remarks:

1. For decreased I/O times, this directory should be on a disk directly attached to the machine where *GENESIS* is running.

4.7 DIRSAF

Executive Control Entry: **DIRSAF** - Scratch File Directory Control

Description: Selects a directory for sequential access scratch files.

Format:

DIRSAF = directory name

Example:

```
DIRSAF = /localscratch
```

Remarks:

1. For decreased I/O times, this directory should be on a disk directly attached to the machine where *GENESIS* is running.

4

4.8 DIRSMS

Executive Control Entry: **DIRSMS** - Scratch File Directory Control

Description: Selects a directory for sparse matrix solver scratch files.

Format:

DIRSMS = directory name

Example:

```
DIRSMS = /localscratch
```

Remarks:

1. For decreased I/O times, this directory should be on a disk directly attached to the machine where *GENESIS* is running.

4

4.9 EIGMETHOD

Executive Control Entry: **EIGMETHOD** - External Eigenvalue Solver Control

Description: Defines the name of an external shared object (DLL) that will solve a general eigenvalue problem.

Format:

EIGMETHOD = *mthid*, full_path_to_shared_object

Examples:

```
EIGMETHOD = 7, /home/mrt/work/eigm_remote.so
EIGMETHOD = 25, D:\users\mrt\eigm_remote.dll
```

Option	Meaning
mthid	Unique EIGMETHOD identification number (Integer >0).

Remarks:

1. For the EIGMETHOD to be used, an activated **EIGR** bulk data entry that references *mthid* must exist.
2. The shared object must export seven required interface functions. The interface functions have the following Fortran declarations:

```
SUBROUTINE EIGCAPS (MTHID, ICAP, IRES)
INTEGER MTHID, ICAP, IRES

SUBROUTINE EIGINIT (MTHID, NEQ, NWANT, ILFLAG, ELEFT, IRFLAG,
                    ERIGHT, NRESCV, NDAMPS, NCTX, ICTX, IERR)
INTEGER MTHID, NEQ, NWANT, ILFLAG, IRFLAG, NRESCV, NDAMPS, IERR
DOUBLE PRECISION ELEFT, ERIGHT
INTEGER(KIND=LONG) NCTX
INTEGER(KIND=LONG) ICTX(NCTX)

SUBROUTINE EIGINPUT (MTHID, IWHAT, NROWS, ICOL, IROWS, VALUES,
                     NCTX, ICTX, IERR)
INTEGER MTHID, IWHAT, NROWS, ICOL, IERR
INTEGER IROWS(NROWS)
DOUBLE PRECISION VALUES(NROWS)
INTEGER(KIND=LONG) NCTX
INTEGER(KIND=LONG) ICTX(NCTX)

SUBROUTINE EIGSOLVE (MTHID, NCTX, ICTX, NFOUND, IERR)
INTEGER MTHID, NFOUND, IERR
INTEGER(KIND=LONG) NCTX
INTEGER(KIND=LONG) ICTX(NCTX)
```

4

```
SUBROUTINE EIGPAIRS (MTHID, NVS, LEVECS, MSTART, MEND,
                     NCTX, ICTX, EVALS, EVECS, IERR)
INTEGER MTHID, NVS, LEVECS, MSTART, MEND, IERR
DOUBLE PRECISION EVALS(NVS), EVECS(LEVECS, NVS)
INTEGER(KIND=LONG) NCTX
INTEGER(KIND=LONG) ICTX(NCTX)

SUBROUTINE EIGMDAMP (MTHID, IDAMP, IMOD, NCTX, ICTX, DAMP, IERR)
INTEGER MTHID, IDAMP, IMOD, IERR
DOUBLE PRECISION DAMP(IMOD)
INTEGER(KIND=LONG) NCTX
INTEGER(KIND=LONG) ICTX(NCTX)

SUBROUTINE EIGFINIS (MTHID, NCTX, ICTX, IERR)
INTEGER MTHID, IERR
INTEGER(KIND=LONG) NCTX
INTEGER(KIND=LONG) ICTX(NCTX)
```

The Fortran KIND type *LONG* is system dependent. On most systems, this should match the result of SELECTED_INT_KIND(18).

Note that if a language other than Fortran is used to create the shared object, care must be taken to ensure that the correct interface function names are exported. The actual required function names are system dependent. For example, using the C language, the functions should be named as follows:

<table>
<tr><td>Microsoft Windows</td><td>EIGCAPS
EIGINIT
EIGINPUT
EIGSOLVE
EIGPAIRS
EIGMDAMP
EIGFINIS</td></tr>
<tr><td>Linux</td><td>eigcaps_
eiginit_
eiginput_
eigsolve_
eigpairs_
eigmdamp_
eigfinis_</td></tr>
</table>

See **Using External Eigenvalue Solver Methods** (Sec. 2.13.3). For more details contact VR&D.

4.10 ESLCONF

Executive Control Entry: **ESLCONF** - ESL Reader Configuration Definition

Description: Define data to configure an external ESL reader defined by an ESLDISP executive control entry.

Format:

ESLCONF = *rid*, character data

Examples:

```
ESLCONF = 7, nodout; esldisp_grok_dyna.conf
ESLCONF = 25, nodout
```

Option	Meaning
rid	ESL reader identification number (Integer >0).

Remarks:

3. The ESL reader identification number must be defined by an **ESLDISP** executive control entry.
4. There may be at most one ESLCONF entry per reader identification number.
5. The format of the character data is dependent on the specific ESL reader used. Consult the documentation for the selected ESL reader module to determine what (if any) configuration data is required.

4

4.11 ESLDISP

Executive Control Entry: **ESLDISP** - External Displacement Reader Control

Description: Defines the name of an external shared object (DLL) that will read displacements to be used to define equivalent static loading.

Format:

ESLDISP = *rid*, full_path_to_shared_object

Examples:

```
ESLDISP = 7, /home/mrt/work/esldisp_grok.so
ESLDISP = 25, D:\users\mrt\esldisp_dyna_nodout.dll
```

Option	Meaning
rid	Unique ESL reader identification number (Integer >0).

Remarks:

1. **ESLOAD** solution control entries must exist to create equivalent static loads.
2. The shared object must export one required interface function. The interface function has the following Fortran declaration:

```
SUBROUTINE ESLDISP(INDEX, MODE, NGRID, IDGRID, DISP, IERR, CDATA)
INTEGER INDEX, MODE, NGRID, IERR
INTEGER IDGRID(2,NGRID)
DOUBLE PRECISION DISP(6,NGRID)
CHARACTER*(*) CDATA
```

Note that if a language other than Fortran is used to create the shared object, care must be taken to ensure that the correct interface function name is exported. The actual required function name is system dependent. For example, using the C language, the function should be named as follows:

Microsoft Windows	ESLDISP
Solaris, Linux, HP-UX	esldisp_
AIX	esldisp

For more details contact VR&D.

4

4.12 GNMASS

Executive Control Entry: **GNMASS** - User Reduced Mass Control

Description: Defines the name of an external shared object (DLL) that will calculate mass matrices for the MAAUSER solution control command.

Format:

GNMASS = full_path_to_shared_object

Examples:

```
GNMASS = /home/mrt/work/gnmass.so
```

```
GNMASS = D:\users\mrt\gnmass.dll
```

Remarks:

1. Only one GNMASS command is allowed.
2. The **MAAUSER**=YES solution control entry must exist to request user-defined mass for an ASET loadcase.
3. The shared object must export one required interface function. The interface function has the following Fortran declaration:

```
  SUBROUTINE GNMASS(UDV,IASET,NDVT,NEQR,NEQRL,IUSERL,
 *                  RMASS,IERROR)
  INTEGER NDVT, NEQR, NEQRL, IUSERL, IERROR
  DOUBLE PRECISION UDV(NDVT),  RMASS(NEQRL)
  INTEGER  IASET(2,NEQR)
```

Note that if a language other than Fortran is used to create the shared object, care must be taken to ensure that the correct interface function name is exported. The actual required function name is system dependent. For example, using the C language, the function should be named as follows:

Microsoft Windows	GNMASS
Solaris, Linux, IRIX, OSF1 HP-UX	gnmass_
AIX	gnmass

For more details see **Using User Supplied Mass Matrix in Guyan Reduction Load Cases** (Sec. 2.13.2).

4

4.13 ID

Executive Control Entry: **ID** - Comment

Description: Enter a comment line.

Format:

ID Any character data

Example:

```
ID MyProjectName
```

Remarks:

1. Previously, the ID command was used to set the “project name”. The project name is used to name post-processing and other output files. The project name is now set to be the same as the input filename base. The project name can be changed to be different from the input file name base using the -p *pname* command line option. For example:

```
genesis -p MYPNAME myinput.dat
```

4

4.14 INCLUDE

Executive Control Entry: **INCLUDE**

Description: Select an external file that contains executive control statements.

Format:

```
INCLUDE 'file name'
```

Alternate Format:

```
INCLUDE = file name
```

Examples:

INCLUDE `'myscript.lua'`

INCLUDE = `K2UU_LIST.TXT`

Option	Meaning
file name	External file name. The user must provide the file name according to the machine installation.

Remarks:

1. Multiple INCLUDE data are allowed in the executive control.
2. The external file cannot contain INCLUDE data statements.
3. The external file cannot contain the **CEND** delimiter.
4. The file name is limited to 240 characters.
5. If the quoted format is used, and the line does not end with a quote character, additional lines will be read until the closing quote is found. Leading and trailing spaces on continued and continuation lines are discarded.

4

4.15 IOBUFF

Executive Control Entry: **IOBUFF** - Input/Output Control

Description: Defines memory buffering characteristics for scratch file Input/Output

Format:

IOBUFF = *n*,*m*K

Examples:

```
IOBUFF = 64,256K
```

Option	Meaning
n	The number of Input/Output buffers to use (Integer > 1).
*m*K	m is the size of each buffer in kilowords (Integer > 7).

Remarks:

1. The use of I/O buffers will take memory in addition to that specified by the **LENVEC** entry. The total amount of additional memory used will be *n***m* kilowords (1 word = 4 bytes).
2. The number of buffers must be greater than the number of threads specified by the **THREADS** entry.
3. The minimum allowable buffersize is 8 kilowords.
4. If invalid values are specified, they will be reset to the closest acceptable values.
5. The IOBUFF value can also be specified on the command line using the -iobuff=*n*,*m*K flag. For example.

   ```
   genesis -iobuff=16,1024K mydata.dat
   ```

   If the IOBUFF command appears in the datafile, it takes precedence over any value specified on the command line.
6. The system administrator may define a value for IOBUFF in an installation policy file. In this case, any IOBUFF specified in the input data or on the command line will be ignored.

4

4.16 K2UU

Executive Control Entry: **K2UU** - User Stiffness Matrix Control

Description: Selects a file containing a user stiffness matrix to add to the program calculated stiffness matrix.

Format:

K2UU = file name

Example:

```
K2UU = reduc.KAA
```

Remarks:

1. The format of the K2UU file is explained in Section **2.4.14**.
2. Multiple K2UU commands are allowed.
3. The length of the file name can be up to 60 characters. The file name can contain a full path.

4.17 K2UU1

Executive Control Entry: **K2UU1** - User Stiffness Matrix Control, Altenate Format 1

Description: Selects a file containing a user stiffness matrix to add to the program calculated stiffness matrix.

Format:

K2UU1 = *pid*, file name

Example:

```
K2UU1 = 10, reduc.KAA
```

Option	Meaning
pid	Identification number of a **PK2UU** bulk data entry (Integer >0).

Remarks:

1. The format of the K2UU file is explained in Section **2.4.14**
2. Multiple K2UU1 commands are allowed.
3. The length of the file name can be up to 60 characters. The file name can contain a full path.

4.18 LENVEC

Executive Control Entry: **LENVEC** - Memory Control

Description: Specifies the amount of memory the program should use.

Format:

LENVEC = *n*

Alternate Format:

LENVEC = *m*K

Alternate Format:

LENVEC = *m*M

Alternate Format:

LENVEC = *m*G

Examples:

```
LENVEC = 85000000
LENVEC = 1500000K
LENVEC = 1500M
```


Option	Meaning
n	The number of words for the main storage array in *GENESIS*. The default value of n is installation dependent. On most systems, one word is 4 bytes. (1,000,000,000>Integer>0).
*m*K	m is the number of words in thousands for the main storage array in *GENESIS*. (Integer>0).
*m*M	m is the number of words in millions for the main storage array in *GENESIS*. (Integer>0).
*m*G	m is the number of words in billions for the main storage array in *GENESIS*. (Integer>0).

Remarks:

1. Following is a list of maximum recommended values for LENVEC for various memory sizes. Using the largest value of LENVEC to fit physical memory will result in the best performance on most computers. Larger values may be used using virtual memory, if available, but performance may be degraded.

Physical RAM	LENVEC
1024 MB	200M
2 GB	400M
8 GB	1500M
32 GB	6000M

2. This command is system dependent and may not be available in custom installations.
3. The LENVEC value can also be specified on the command line using the -lenvec=*value* flag. For example.

   ```
   genesis -lenvec=2000M mydata.dat
   ```

   If the LENVEC command appears in the datafile, it takes precedence over any value specified on the command line.
4. On 32-bit systems, the total amount of memory specified cannot exceed 500,000,000 words.
5. The K, M or G option is required when the number of words is one billion or larger. For example, to specifiy 1,000,000,000 words, use 1000000K or 1000M or 1G.
6. The system administrator may define a value for LENVEC in an installation policy file. In this case, any LENVEC specified in the input data or on the command line will be ignored.

4.19 M2UU

Executive Control Entry: **M2UU** - User Mass Matrix Control

Description: Selects a file containing a user mass matrix to add to the program calculated mass matrix.

Format:

M2UU = file name

Example:

```
M2UU = reduc.MAA
```

Remarks:

1. The format of the M2UU file is explained in Section **2.4.14**.
2. Multiple M2UU commands are allowed.
3. The length of the file name can be up to 60 characters. The file name can contain a full path.

4

4.20 M2UU1

Executive Control Entry: **M2UU1** - User Mass Matrix Control, Altenate Format 1

Description: Selects a file containing a user mass matrix to add to the program calculated mass matrix.

Format:

M2UU1 = *pid*, file name

Example:

```
M2UU1 = 10, reduc.KAA
```

Option	Meaning
pid	Identification number of a **PM2UU** bulk data entry (Integer >0).

Remarks:

1. The format of the M2UU file is explained in Section **2.4.14**
2. Multiple M2UU1 commands are allowed.
3. The length of the file name can be up to 60 characters. The file name can contain a full path.

4.21 POST

Executive Control Entry: **POST** - Post-Processing Format Control

Description: Selects the format for post-processing files.

Format:

$$\textbf{POST} = \left\{ \begin{array}{c} \texttt{BINARY} \\ \texttt{FORMAT} \\ \texttt{PLOT} \\ \texttt{OUTPUT2} \\ \texttt{PUNCH} \\ \texttt{PATRAN} \\ \texttt{IDEAS} \end{array} \right\}$$

Example 1:

```
POST=PUNCH
```

Example 2:

```
POST=PUNCH,OUTPUT2
```

Remarks:

1. Post-processing files can contain grid point displacements, velocities, accelerations, grid point temperatures, applied loads, reaction forces, element forces, stresses, strains and mode shapes. Post-processing data is only written for output results that are requested in the solution control section of the input data. A separate file is written for each design cycle.
2. See **Post-Processing Data** (Sec. 7.3) for a detailed discussion of available post processing formats. The available formats are:

POST Option	File Format
BINARY	GENESIS Binary
FORMAT	GENESIS ASCII
PLOT	GENESIS ASCII with structure
OUTPUT2	NASTRAN OUTPUT2
PUNCH	NASTRAN PUNCH
PATRAN	MSC.Patran 2.5
IDEAS	I-deas Universal

3. The displacements, velocities, accelerations and mode shapes are written in the basic or global coordinate system, depending on the Solution Control command **POSTOUTPUT**. The default is the basic coordinate system.

4

4. Multiple formats can be listed separated with comma. All requested analysis results will be written to a separate post-processing file for each format listed.
5. To add model data to OUTPUT2 formatted files, PARAMeter **IPRM16** can be used. By default, model data will not be written to OUTPUT2 files.

4.22 REDUCE

Executive Control Entry: **REDUCE** - Program Flow Control

Description: Causes the program to only reduce matrices and loads to a specified boundary set of degrees of freedom.

Format:

REDUCE

Example:

```
REDUCE
```

Remarks:

1. Mutually exclusive with **CHECK**.
2. When the REDUCE command is used, the **BOUNDARY** solution control command must appear above the first loadcase.
3. The only loadcase types allowed are static or natural frequency. When the REDUCE command is used, only superelement reduction is performed. No static or eigenvalue results are calculated. Eigenvector output requests are used only if the parameter **SEMPC** is set to non-zero, in which case they only define the grids for which to write recovery MPC data. No eigenvectors are output.

4.23 SOL

Executive Control Entry: **SOL** - Input File Compatibility Control

Description: Selects the compatibility format mode of the input file.

Format:

SOL n

Example:

```
SOL COMPAT1
```

Remark:

1. This command allows *GENESIS* to more easily use input files created by different preprocessors by requiring less hand editing. If n is “COMPAT0”, then the input format is in the legacy GENESIS format. If n is “COMPAT1” or any other string or any integer, then the input is in compatibility level 1 mode. This changes the way *GENESIS* interprets some data on **PBAR** and **PSHELL** entries. See the following table.

n	Input format mode specifics
COMPAT0	Interpret fields 2 and 3 of the second continuation line of PBAR entries as shear areas, AS1 and AS2. Interpret field 6 of PSHELL entries as the bending stiffness, D. Interpret field 8 of PSHELL entries as the transverse shear thickness, TS.
COMPAT1	Interpret fields 2 and 3 of the second continuation line of PBAR entries as shear area factors, K1=AS1/A and K2=AS2/A. Interpret field 6 of PSHELL entries as the bending stiffness factor, $DF=12\ D/T^3$. Interpret field 8 of PSHELL entries as the transverse shear factor, TSF=TS/T.

2. The compatibility format mode can also be specified on the command line using the -sol=*value* flag. For example.

   ```
   genesis -sol=compat1 mydata.dat
   ```

   If the SOL command appears in the datafile, it takes precedence over any value specified on the command line.
3. If no compatibility format mode is specified by a SOL command or the -sol command line option, COMPAT1 mode is used.
4. If n is an integer or any string other than “COMPAT0”, then COMPAT1 mode is used

5. PSHELL and/or PBAR entries written by *GENESIS* in the *pname*.OPT file will be in the same format mode as the input data. PBAR and/or PSHELL entries written by *GENESIS* should only be copied to input files with the same SOL COMPATi value as the original input file. Otherwise, the entries must be edited to ensure correct interpretation of the data.

4.24 THREADS

Executive Control Entry: **THREADS** - Parallel Control

Description: Selects number of parallel threads.

Format:

THREADS = n

Example:

THREADS = 4

Option	Meaning
n	The number of parallel threads to use (Integer > 0).

Remarks:

1. This command is intended to be used on computers with multiple CPU cores (either multiple physical CPUs or a single multi-core CPU). If the number of threads specified is greater than the number of CPU cores available, performance will be degraded.
2. The THREADS value can also be specified on the command line using the -threads=*value* flag. For example.

   ```
   genesis -threads=4 mydata.dat
   ```

   If the THREADS command appears in the datafile, it takes precedence over any value specified on the command line.
3. The system administrator may define a value for THREADS in an installation policy file. In this case, any THREADS specified in the input data or on the command line will be ignored.

4.25 UFDATA

Executive Control Entry: **UFDATA** - User Dynamic Linear Function Data

Description: Selects a file containing data to create a linear combination of dynamic displacements, velocities or accelerations.

Format:

UFDATA = file name

Alternate Format:

UFDATA ‘file name’

Examples:

```
UFDATA = ufdata.txt
UFDATA = /home/user/advance_dynamics/ufdata.txt
```

Remarks:

1. The format of the UFDATA file is explained in Section **2.16.1**.
2. Only one UFDATA command is allowed.
3. The file name is limited to 240 characters.
4. The results of the user linear combinations can be output using the solution control commands **UFDISP**, **UFVELO** and/or **UFACCE**. These commands can be placed in direct or modal frequency response loadcases.

CHAPTER 5

Solution Control

5

5.1 Output Headers

The second through fourth lines of each page of output contain the project **TITLE**, **SUBTITLE**, and load case **LABEL** respectively. TITLE and SUBTITLE appearing above the first loadcase will be used as loadcase defaults as well as for output pages not specific to any loadcase. TITLE and SUBTITLE appearing after the LOADCASE command apply to that loadcase only. The load case LABEL is defined after the **LOADCASE** command. For example,

```
TITLE=TEN BAR TRUSS PROBLEM
SUBTITLE=GRAVITY AND THERMAL LOADS
LOADCASE 3
LABEL=GRAVITY LOADS
    .
    .
LOADCASE 5
LABEL=THERMAL LOADS ( 100 DEGREES )
    .
    .
```

On pages of output which contain information about the entire project, such as an echo of the input data, only the TITLE and SUBTITLE are printed. On pages which contain output for a specific load case, say the displacements in LOADCASE 5, the LABEL "THERMAL LOADS (100 DEGREES)" will also be printed. Note that all three header commands are optional (blank lines are the default).

GENESIS reads up to the 80th character in a line of input data. The statement names TITLE, SUBTITLE and LABEL, as well as the equal sign and blanks are included. Therefore, long titles, subtitles and labels that exceed the 80th column are not completely printed on the output file.

5.2 Static Loadcases

The static loads, point loads and pressure(traction) loads are specified with the command **LOAD**=nl, where nl is the load set number. Similarly the temperature, gravity,centrifugal and deformation loads are specified with the **TEMPERATURE**=nt, **GRAVITY**=ng, **CENTRIFUGAL**=nc and **DEFORM**=nd commands. For example:

```
LOADCASE 1
   SPC=2
   MPC=4
   LOAD=5
LOADCASE 2
   SPC=4
   GRAV=5
   TEMP=1
   CENT=6
   DEFORM=10
```

In this example the static load set number 5 is applied in LOADCASE 1 and gravity load set 5, thermal load set 1, centrifugal load 6 and deformation load 10 are applied in LOADCASE 2. Note that the different load types can have the same set numbers and that the load set and boundary condition commands can be in any order after the LOAD CASE command. The static, thermal, centrifugal, deformation and gravity loads are specified in the BULK DATA section of the input data. The thermal load set number can be the LOADCASE number of a heat transfer LOADCASE.

If the user wishes for the applied load vector to be written to the output file the command **OLOAD**=no must be used. Similarly, the displacements and reaction forces are requested with the commands **DISPLACEMENT**=nd and **SPCFORCE**=nr. The values of no, nd and nr can be ALL for output from all grid points, NONE for no output (the default), POST for post processing only or ##, where ## is the number of a previously defined set of grid points. For example;

```
SET 1 = 1,2,5,6
LOADCASE 1
   OLOAD=NONE
   DISP=ALL
   SPC=2
   MPC=4
   LOAD=5
LOADCASE 2
   SPC=4
   DISP=ALL
   SPCF=1
   GRAV=5
   TEMP=1
   CENT=6
```

In this example the displacements at all the grid points and the applied loads at none of the grid points are written to the output file for LOADCASE 1. For LOADCASE 2 the displacements at all the grid points and the reaction forces at grid points 1, 2, 5, and 6 are written to the output file.

The user can request that element forces, stresses, or strains be written to the output file using the commands **FORCE**=nf, **STRESS**=ns, and **STRAIN**=nn, where nf, ns, and nn are equal to ALL, POST, NONE, or ## where ## is the number of a previously appearing set of element numbers, i.e.:

```
SET 4 = 1,4,9
LOADCASE 4
   STRESS=ALL
   STRAIN=4
```

In this example all of the element stresses and the strains for elements 1, 4, and 9 are output for LOADCASE 4.

Grid stresses can also be requested for grids connected to solid elements (**CPENTA**, **CTETRA**, **CHEXA**, **CHEX20**, **CPYRA** and **CTRIAX6**) by the user. To do this, the command **GSTRESS** = ng can be used. Parameter ng could be ALL, POST, NONE or ##, where ## is the number of a set of grid numbers, i.e.:

```
SET 25 = 100 THRU 500
LOADCASE 5
   GSTRESS = 25
   LOAD = 10
```

In this example, stresses at all grids between 100 and 500 will be printed.

In addition, the user can request that element strain energies be output to the post processing file. To do this, the command **ESE** = POST can be used.

5.3 Equivalent Static Loadcase

The equivalent static loads are specified with the command **ESLOAD**=nl, character data, where nl is the equivalent statc reader number (**ESLDISP**) and character data contains information passed to the reader to choose which displacements to read.

For example:

```
LOADCASE 1
   SPC=2
   MPC=4
   ESLOAD=5,time=1.0E-2
LOADCASE 2
   SPC=4
   ESLOAD=6,time=2.0E-2
```

In this example the equivalent static load set number 5 is applied in LOADCASE 1 and the equivalent static load set set 6 in LOADCASE 2. Note that the load set and boundary condition commands can be in any order after the LOADCASE command. The equivalent static load reader (**ESLDISP**) and equivalent static load reader configuration (**ESLCONF**) are specified in the exectuive control.

The user can specify single point constraints (**SPC**), multipoint constraints (**MPC**) and reference degrees of freedom for a free body analysis using the command **SUPORT** = ns.

All output request available for a regular static loadcase are available for the equivalent static loadcases, namely: applied loads (**OLOAD**); displacements (**DISPLACEMENT**); reaction forces (**SPCFORCE**), element forces (**FORCE**); element stresses (**STRESS**); element strains (**STRAIN**); grid stresses (**GSTRESS**); and element strain energies (**ESE**).

Static loads defined bulk data, such as point, pressure, temperature, gravity, centrifugal and deform loads **cannot** be used in an equivalent static loadcase.

5

Note:

The equivalent static loadcase is used to implement the “Equivalent Static Load Method” for optimization using responses from nonlinear structural analysis. The nonlinear analysis must be performed by a third-party code. Using special plug-in modules (readers) that read the results directly from those codes, Genesis can generate equivalent static load which will produce the same displacements. Genesis optimization of the equivalent static load linear problem gives an approximate solution to the optimum solution of the nonlinear problem. An iterative process consisting of nonlinear analysis followed by Genesis optimization of the equivalent static load linear problem converges to the true optimum of the nonlinear problem, and has proven to be an effective and efficient method.

VR&D has developed readers for several popular third-party nonlinear analysis programs. Contact VR&D or your distributor for more information.

5.4 Static Loadcase with Inertia Relief

In addition to the load and boundary conditions selected in a regular static loadcase or in an equivalent static loadcase, the user can specify reference degrees of freedom for a free body analysis using the command **SUPORT** = ns. For example,

```
LOADCASE 1
   SUPORT = 10
   LOAD = 1
LOADCASE 2
   SUPORT = 20
   GRAV = 2
```

In this example, the support degrees of freedom in set 10 are applied to LOADASE 1 and the support degrees of freedom in set 20 are applied to LOADCASE 2.

To make the program automatically select 6 support dofs, the user can specify the following command: **SUPORT** = AUTO. For example,

```
LOADCASE 1
   SUPORT = AUTO
   LOAD = 1
LOADCASE 2
   SUPORT = 20
   GRAV = 2
```

The PARAMeter **INREL**=-2 can be used to set SUPORT=AUTO as the default for all static loadcases. In this case, to avoid inertia relief in a loadcase, use SUPORT=NONE in that loadcase.

5.5 Nonlinear Static Loadcases

GENESIS can solve nonlinear static contact analysis, where the potential contact points are modeled with **CGAP** elements . The CGAP element is a simple bi-linear spring element, where one stiffness is used when the gap is in the “closed” state and another stiffness is used when the gap is in the “open” state. *GENESIS* can also solve nonlinear static contact analysis, where potential contacting surfaces are defined with **BCPAIR** entries. BCPAIR entries must be selected in a loadcase using the **BCONTACT**=bc command. When surface-to-surface contact analysis is performed, contact clearances, forces and pressures can be output using the **CDISP**, **CFORCE** and **CPRESSURE** commands, respectively.

Nonlinear analysis is activated by specifying nonlinear static control parameters with the command **NLPARM**=nd, where nd is the nonlinear parameter set number.

For example:

```
LOADCASE 10
   LABEL = NONLINEAR GAP ANALYSIS
   LOAD = 4
   SPC = 5
   NLPARM = 10
LOADCASE 15
   LABEL = NONLINEAR SURFACE-TO-SURFACE CONTACT ANALYSIS
   LOAD = 4
   SPC = 5
   BCONTACT = 15
   NLPARM = 10
   CDISP = ALL
   CPRESSURE = ALL
   CFORCE = ALL
LOADCASE 20
   LABEL = LINEAR ANALYSIS
   LOAD = 4
   SPC = 5
```

In this example the nonlinear parameter set 10 is selected using the NLPARM command. In the bulk data an **NLPARM** data entry with set id 10 should exist to specify the nonlinear control parameters. To solve the nonlinear system, initial gap states (open/closed) for each gap element will be assumed. The linear static system will be solved, and the displacements will be used to calculate the actual gap states. If the actual gap states match the assumed gap states, then the nonlinear system is solved. If not, then assumed states are updated, and another static iteration begins. The maximum allowed iterations is specified on the NLPARM bulk data entry. If the system is not solved after MAXITER iterations, the program will stop with an error message. If there are no CGAP elements in the model and no BCPAIR set is selected in the loadcase, the NLPARM entry will have no effect on the analysis.

If a static loadcase does not have an NLPARM entry, as in loadcase 20 above, no nonlinear analysis will be performed, and the CGAP elements will behave like linear spring elements. In this case, the linear stiffness for each gap element will be determined by the initial gap opening (U0) from the corresponding **PGAP** entry. Likewise, in all non-static loadcases, the CGAP elements will behave like linear spring elements.

5.6 Frequency Calculation Loadcases

The frequency calculation method is specified with the command **METHOD**=ne, where ne is the number of an EIGR or EIGRL entry in the BULK DATA section. The user can request that all, none, or a specific set of mode shapes be written to the output data file using the command **SVECTOR**=nv, where nv is ALL, POST, NONE, or ##, where ## is the number of a previously defined set of grids. Also, the user can request the calculation of strain energies for all elastic elements for selected modes using the command **ESE** = nv. For example:

```
SET 1 = 1,2
LOAD CASE 1
   METHOD=1
   SPC=2
   MPC=4
   SVEC=ALL
LOAD CASE 2
   SPC=4
   SVEC=1
   METHOD=1
   ESE=1
```

In this example the frequency calculation parameters are found in **EIGR** or **EIGRL** 1 and all of the mode shapes are written to the output file for LOADCASE 1. Mode shapes 1 and 2 and element strain energies are output for all elastic elements for modes 1 and 2 are written for LOADCASE 2

Any non-zero enforced displacement values on **SPC** bulk data referenced by a frequency calculation loadcase will be ignored. Instead, those displacements will be constrained to zero.

5.7 Frequency Calculations using Guyan Reduction

In addition to the method, boundary conditions and mode tracking selected in a regular frequency loadcase, the user can specify free degrees of freedom for Guyan reduction analysis using the command **ASET** = na. The user can request that the reduced matrices be printed to post processing files using the commands **KAA** = POST, **MAA** = POST.The user can also request that the reduced matrices be printed to DMIG files using the commands **KAA** = DMIG, **MAA** = DMIG. The user may replace the *GENESIS* reduced mass matrix with his own reduced mass matrix using the GNMASS routine. To select that option, the command **MAAUSER** = YES is used. For example

```
SET 1 = 1,2
LOADCASE 1
   METHOD =1
   SPC = 2
   MPC = 4
   SVEC = ALL
   ASET = 10
LOADCASE 2
   METHOD = 1
   SPC = 4
   MODTRK = ALL
   ESE = 1
   ASET = 20
   KAA = POST
   MAA = POST
   MAAUSER = YES
```

In this example, the ASET degrees of freedom in **ASET** 10 are applied in LOADCASE 1 and the ASET degrees of freedom in SET 20 are applied in LOADCASE 2. For LOADCASE 2, *GENESIS* will print the reduced stiffness and mass. In LOADCASE 2, the user supplied mass matrix calculated in subroutine GNMASS is requested. Also, in LOADCASE 2, mode tracking will be performed. Mode tracking will not be performed for LOADCASE 1.

5

5.8 Frequency Calculations using Guyan Reduction and Craig- Bampton Modes

To specify Craig-Bampton modes in a Guyan reduction loadcase, the user needs to specify a set of generalized degrees of freedoms with the command **QSET**=nq and the additional eigenvalue control command **CBMETHOD**=nl. All the commands available for a regular Guyan reduction loadcase can be used with Craig-Bampton modes. For example:

```
SET 1 = 1,2
LOADCASE 200
   LABEL = GUYAN REDUCTION LOADCASE WITH CRAIG BAMPTON MODE SHAPES
   METHOD = 2
   SPC = 4
   MODTRK = ALL
   ESE = 1
   ASET = 20
   QSET = 25
   CBMETHOD = 1
   KAA = POST
   MAA = POST
   MAAUSER = YES
   SVEC = ALL
```

In this example, the LOADCASE 200 is used to calculate both the Craig-Bampton modes using CBMETHOD=1 and the Guyan reduced modes using METHOD = 2. The boundary conditions are specified by SPC=4. The generalized degrees of freedom are specified in set 25. For this loadcase, *GENESIS* will print the reduced stiffness and mass. In this load case, the user supplied mass matrix calculated in subroutine GNMASS is requested. Also, mode tracking will be performed for the reduced eigenvalue problem.

5.9 Superelement Reduction

If the executive control command **REDUCE** is used, stiffness, mass, damping and loads can be reduced to a specified set of degrees of freedom using static condensation. To perform a superelement analysis, a large model is divided into smaller pieces (called superelements). A superelement is reduced to its boundary (or interface) degrees of freedom. Then, a model with only the collection of all boundary degrees of freedom of all superelements along with all degrees of freedom of non-reduced pieces (the residual structure) can be assembled and solved. For a superelement model, the boundary degrees of freedom for reduction analysis are specified using the command **BOUNDARY** = na. This command must be used before the first loadcase definition, and applies to all loadcases in the model. The user can request that the reduced matrices be printed to post processing files using the commands **KAA**, **MAA** , **K4AA** and **ALOAD**. For example

```
BOUNDARY = 20
SPC = 2
MPC = 4
ALOAD = DMIG
LOADCASE 1
   LOAD =1
   KAA = DMIG
LOADCASE 2
   LOAD = 2
LOADCASE 3
   LOAD = 3
LOADCASE 4
   LOAD = 4
```

In this example, the ASET degrees of freedom in BOUNDARY 20 are applied. The reduced stiffness for LOADCASE 1 will be printed to the DMIG postprocessing file (the reduced stiffness is the same for all loadcases, since they all use the same boundary conditions). The reduced load vectors for all LOADCASEs will also be printed to the DMIG postprocessing file. These DMIG matrices use the format of the **DMIG** bulk data entry, and can be included into the residual structure model where they would be selected with the **K2GG** and **P2G** solution control commands.

5.10 Buckling Calculation Loadcases

Each buckling loadcase references a static loadcase using the **STATSUB** command. The referenced static loadcase must contain all the loads and boundary conditions needed for buckling analysis. The buckling load factor calculation method and any mode shape output requests must be specified in the buckling loadcase itself. The buckling load factor calculation method is specified with the command **METHOD**=ne, where ne is the number of an **EIGR** or **EIGRL** entry in the BULK DATA section. The user can request that all, none, or a specific set of mode shapes be written to the output data file using the commands **SVECTOR**=nv or **DISPLACEMENT**=nv, where nv is ALL, POST, NONE, or ##, where ## is the number of a previously defined set of grids. For example:

```
LOADCASE 101
   LOAD=1
   SPC=2
   MPC=4
LOADCASE 201
   STATSUB=101
   SVEC=ALL
   METHOD=1
```

In this example the LOADCASE 101 correspond to a static loadcase. LOADCASE 201 is the buckling loadcase. The buckling loadcase references LOADCASE 101 using the STATSUB command. The load factor calculation parameters are found in the EIGR entry with ID 1 and all of the buckling mode shapes are written to the output file for LOADCASE 201.

5.11 Heat Transfer Loadcases

The thermal loads are specified with the command **HEAT**=nh, where nh is the thermal load set number. The grid point temperatures are requested with the command **THERMAL**=nt, where nt can be ALL, NONE, POST or ##, where ## is the number of a previously appearing set of grid point numbers. For example:

```
LOADCASE 6
   HEAT = 10
   THERMAL = ALL
```

The command **OLOAD**=nl can be used to request the printing of the applied flux vector. Similarly, the command **SPCFORCE**=nr can be used to request printing of the heat flux at grids with constrained (specified) temperatures. Heat flux output is not written to the post processing file. Heat transfer loadcases must be specified before all other loadcases.

5.12 Static Loadcase Combinations

The preceding static load cases are combined in a linear manner with scale factors that are specified with the **LOADSEQ** command. For example:

```
LOADCASE 1
   SPC=1
   LOAD=1
   DISP=ALL
LOADCASE 2
   SPC=2
   LOAD=2
LOADCOM 3
   LOADSEQ = 1.,2.
   DISP=ALL
   STRESS=ALL
```

In this example the displacements from LOADCASE 1 are added to twice the displacements of LOADCASE 2. The element forces, stresses, and strains are then recovered from these new displacements. Note that load and boundary conditions cannot be specified in the LOADCOM. Applied load vectors (**OLOAD**) and reaction forces (**SPCFORCE**) are not available for LOADCOM's. Thermal loads may only appear for one LOADCASE in a LOADCOM. If more than one LOADCASE includes thermal loads, a fatal error will result. The sequence coefficients on the LOADSEQ data must be zero for preceding heat transfer, dynamic response and frequency calculation LOADCASES. Preceding LOADCOMs are skipped and must not have sequence coefficients.

For example:

```
LOADCASE 1
   LOAD=1
LOADCASE 2
   LOAD=2
LOADCASE 3
   METHOD=3
LOADCASE 4
   LOAD=4
LOADCOM 5
   LOADSEQ = 1.0, 0.0, 2.0
LOADCOM 6
   LOADSEQ = 0.0, 2.0, 0.0, 2.0
```

In this example LOADCOM 5 is a combination of LOADCASE 2 and twice LOADCASE 4. LOADCOM 6 is a combination of twice LOADCASE 2 and twice LOADCASE 4.

Note: The bulk data **LOAD** allows load sets to be combined in a regular load case. This command allows for an alternative method for doing load combinations. However, the LOAD command is not as general as the LOADCOM command. The LOAD command can only be used to combine FORCEx, MOMENTx and PLOADx sets.

5.13 Single Loadcase

If the project has a single load case the LOADCASE delimiter is not needed. For example a single frequency load case could be specified as;

```
TITLE = SINGLE FREQUENCY LOAD CASE
LABEL = SIMPLY SUPPORTED
SPC=2
METHOD=5
```

5.14 Enforced Displacement Loadcase

An enforced displacement is specified by the commands **LOAD** = nl and **SPC** = ns. The enforced displacement is given in an **SPCD** statement. The degree of freedom being specified has to be constrained using a **SPC1** data entry. For example:

```
LOADCASE 1
LOAD=3
SPC=5
```

In this example, bulk data must be provided with an SPCD statement (SID=3) and an SPC1 statement (SID=5).

Alternatively, an enforced displacement may be specified by **SPC** = ns that references one or more **SPC** bulk data entries that give the enforced displacements. However, if more than one loadcase is desired, with different values of the enforced displacements, it is more efficient to use SPCD instead of SPC.

5.15 Enforced Temperature Loadcase

An enforced temperature is specified by the commands **HEAT** = nl and **SPC** = ns. The enforced temperature is given in a **SPCD** statement. The grid point being specified has to be constrained using a **SPC1** data entry. For example;

```
LOADCASE 1
HEAT=3
SPC=5
```

In this example, bulk data must be provided with an SPCD statement (SID=3) and an SPC1 statement (SID=5).

Enforced temperatures may also be specified by **SPC** = ns that references **SPC** bulk data entries giving the enforced values. However, the loadcase must also have the **HEAT** = nl activator to indicate heat transfer analysis rather than structural analysis.

5.16 Thermal Loads from a Heat Transfer Loadcase

The static thermal loads can come from the solution of a heat transfer LOADCASE by referencing the LOADCASE ID with the **TEMPERATURE** command. For example:

```
LOADCASE = 3
   LABEL = CALCULATE THERMAL LOADS
   HEAT = 6
   THERMAL = ALL
LOADCASE 10
   LABEL = STATIC SOLUTION
   TEMP = 3
   STRESS = ALL
```

In this example, the temperatures from the solution of LOADCASE 3 are used as the loads in LOADCASE 10.

5

5.17 Direct Frequency Response Loadcase

The frequency response loads are specified with the command **DLOAD**=nd, where nd is the dynamic load set number. The frequency set for which the dynamic loads are applied is specified by **FREQUENCY**=nf command, where nf is the frequency set number. Single and multipoint constraints are specified with the **SPC** and **MPC** commands respectively. For example:

```
LOADCASE 10
   LABEL = DYNAMIC RESPONSE
   SPC = 4
   DLOAD = 6
   FREQ = 1
```

In this example the dynamic load set is 6, the loading frequency set is 1, and the single point constraint set is 4.

Enforced displacements, velocities or accelerations can be applied in dynamic response analysis. In this case, the DLOAD data should reference (directly or indirectly through **DLOAD** bulk data) **RLOAD1** or **RLOAD2** data that, in turn, references **SPCD** data. The degrees of freedom being enforced must also be constrained using **SPC1** data. Any non-zero enforced displacement values on **SPC** bulk data referenced by a frequency response loadcase will be ignored. Instead, those displacements will be constrained to zero.

Displacements, velocities, accelerations, element forces, element stresses, element strains, and grid point stresses are requested with the **DISPLACEMENT**, **VELOCITY**, **ACCELERATION**, **FORCE**, **STRESS**, **STRAIN**, and **GSTRESS** commands respectively. User functions of displacements, velocities and accelerations (associated to the **UFDATA** executive control command) are requested with the **UFDISP**, **UFVELO** and **UFACCE** respectively. The solution control command **DYNOUTPUT** controls whether the dynamic analysis results are output in real and imaginary or magnitude and phase components. Applied loads (**OLOAD**) and reaction forces (**SPCFORCE**) cannot be requested for dynamic response analysis.

5.18 Modal Frequency Response Loadcase

The frequency response loads are specified with the command **DLOAD**=nd, where nd is the dynamic load set number. The frequency set at which the dynamic load is applied is specified by **FREQUENCY**=nf command, where nf is the frequency set number. The mode shapes that are used for the analysis are specified by the command **MODES**=nm, where nm is the LOADCASE number of a frequency calculation load case. Single and multipoint constraints are not specified in a modal dynamic response load case because they are already specified in the frequency calculation load case. Modal damping is specified with the command **SDAMPING**=ns, where ns the modal damping table number. For example:

```
LOADCASE 4
   LABEL = MODES FOR MODAL DYNAMIC RESPONSE
   METHOD = 4
   SPC = 5
   MPC = 7
LOADCASE 20
   LABEL = DYNAMIC RESPONSE
   DLOAD = 3
   FREQ = 2
   SDAMP = 12
   MODES = 4
```

In this example the dynamic load set is 3, the loading frequency set is 2, and the modal damping table number is 12. The mode shapes used in this analysis come from frequency calculation LOADCASE 4, which also specifies the SPC and MPC set data.

Enforced displacements, velocities or accelerations can be applied in dynamic response analysis. In this case, the DLOAD data should reference (directly or indirectly through **DLOAD** bulk data) **RLOAD1** or **RLOAD2** data that, in turn, references **SPCD** data. The degrees of freedom being enforced must also be constrained using **SPC1** data in the referenced eigenvalue loadcase.

Displacements, velocities, accelerations, element forces, element stresses, element strains, and grid point stresses are requested with the **DISPLACEMENT**, **VELOCITY**, **ACCELERATION**, **FORCE**, **STRESS**, **STRAIN**, and **GSTRESS** commands respectively. User functions of displacements, velocities and accelerations (associated to the **UFDATA** executive control command) are requested with the **UFDISP**, **UFVELO** and **UFACCE** respectively. The solution control command **DYNOUTPUT** controls whether the dynamic analysis results are output in real and imaginary or magnitude and phase components. Applied loads (**OLOAD**) and reaction forces (**SPCFORCE**) cannot be requested for dynamic response analysis.

Alternatively, rather than making a second loadcase for calculating the modes for the modal reduction, the **METHOD**=ne, **SPC**=ns, and **MPC**=nc commands can be used instead of the MODES=nm command to specify the mode calculation parameters and boundary conditions. For example:

```
LOADCASE 1
   LABEL = MODAL DYNAMIC RESPONSE
   DLOAD = 3
   FREQ = 2
   SDAMP = 12
   METHOD = 4
   SPC = 5
   MPC = 7
```

5.19 Random Loadcases

The random response power spectral density factors are specified with the command **RANDOM**=nd, where nd is the random analysis load set number.

To use random response, the user needs to use frequency responses loadcases. Random response can not exist without frequency responses loadcases.

For example:

```
LOADCASE 4
   LABEL = MODES FOR MODAL DYNAMIC RESPONSE
   METHOD = 4
   SPC = 5
   MPC = 7
LOADCASE 20
   LABEL = DYNAMIC RESPONSE
   DLOAD = 3
   FREQ = 2
   SDAMP = 12
   MODES = 4
   RANDOM = 10
   DISPLACEMENT(RPRINT,PSDF,CRMS)=ALL
```

In this example the random load set 10 is selected using the RANDOM command. In the bulk data a **RANDPS** data entry with load set id 10 should exist to generate corresponding spectral density results and optionally a **RANDT1** data entry should be used to perform autocorrelations. The data entry **RANDPS** lists the frequency response loadcase needed (for example Loadcase 20). The dynamic loadcase is defined in LOADCASE 20. In load case 20, the dynamic load set is 3, the loading frequency set is 2, and the modal damping table number is 12. The mode shapes used in this analysis come from frequency calculation LOADCASE 4, which also specifies the SPC and MPC set data.

Displacements, velocities, accelerations, element forces, element stresses and element strains can requested with the **DISPLACEMENT**, **VELOCITY**, **ACCELERATION**, **FORCE**, **STRESS**, and **STRAIN** commands respectively.

In this example, the power spectral density function of the displacements, as well as the cumulative root means square of the displacements are requested to be printed to the output file.

5.20 Defaults

Any SPC, MPC, METHOD, HEAT, LOAD, GRAV, CENT, DEFORM, TEMP, DLOAD, MODES, SDAMP or FREQ, as well as all the output requests that occur above the first LOADCASE delimiter becomes a default for all the LOADCASES. For example:

```
SPC=3
METHOD=3
SVECT=ALL
LOADCASE 2
LOADCASE 3
   LOAD =5
   DISP=ALL
LOADCASE 4
   SPC=2
```

In this example LOADCASE 2 is a frequency calculation load case with SPC set 3 and LOADCASE 4 is a frequency calculation load case with SPC set 2. LOADCASE 3 is a static load case with SPC set 3 and load set 5.

NOTE: If heat transfer LOADCASEs are mixed with any other LOADCASEs, the SPC and MPC commands **cannot** occur above the first LOADCASE delimiter.

5.21 Other General Output Control Commands

The command **ECHO** is used to control the printing of the input data in the output file. If ECHO=UNSORT then the input data is echoed in the output as it appears in the input file. If ECHO=SORT then the input data is printed in input format by item, i.e. grid points, elements, loads, etc. If ECHO=BOTH then the input data is printed in both sorted and unsorted formats. If optimization data present, sorted echo are printed using values updated by initial design values. If ECHO=NONE then no input data is printed. The ECHO command can appear anywhere in the solution control section of the input data.

The **LINE** command is used to override the system dependent defaults for the number of lines of data printed on each page of the output file. The usual default is 64 lines per page. The number of characters per line can also be overridden. The default is usually 132 characters per line. Only 80 and 132 characters per line are allowed.

The command **GRMASS** is used to control the printing of the grid mass matrix. It can be either POST or NONE. The default is NONE. Currently, only the BINARY, FORMAT or PLOT post processing formats are available.

The command **SUMMARY** controls the printing of a summary table of the analysis and design problem sizes. SUMMARY = YES, the default, is used to request printing of the table. SUMMARY = NO is used when no printing is desired.

The commands **MASS** and **VOLUME** are used to control the printing of the mass and volume summary tables at each design cycle. They can be either YES or NO. The default is NO.

The command **DYNOUTPUT** is used to control the format of all printed complex responses of dynamic analysis. The default is to print the results in rectangular format (Real and Imaginary); the other possibility is to print the results in polar format (Magnitude and Phase). The syntax is; DYNOUTPUT=REAL, DYNOUTPUT=IMAGINARY or DYNOUTPUT=RECTANGULAR to get rectangular format, or DYNOUTPUT=MAGNITUDE, DYNOUTPUT=PHASE or DYNOUTPUT=POLAR to get polar format.

The command **POSTOUTPUT** is used to specify the output coordinate system for displacements, velocities, accelerations, basis or perturbation vectors, shape changes, and mode shapes in the post processing file. The default is the basic coordinate system. The syntax is POSTOUTPUT = BASIC, POSTOUTPUT = GENERAL or POSTOUTPUT = GLOBAL. GLOBAL and GENERAL have the same meaning.

The command **TIMES** is used to specify the printing of CPU and ELAPSED times. The syntax is TIMES = SCREEN, PRINT, BOTH or NONE. The default is NONE.

5.22 Loadcase Definition

Loadcase Delimiters

LOADCASE	Defines beginning load case
LOADCOM	Defines beginning of load case which is a linear combination of preceding static load cases
SUBCASE	Defines beginning load case (synonym for LOADCASE)
SUBCOM	Defines beginning of load case which is a linear combination of preceding static load cases (synonym for LOADCOM)

Loadcase Control

LOADSEQ	Defines coefficients for linear combinations in LOADCOM
SUBSEQ	Defines coefficients for linear combinations in LOADCOM (synonym for LOADSEQ)

5.23 Data Selection

Static Load Selection

LOAD	Selects static load condition
TEMPERATURE	Selects temperature set for static load
GRAVITY	Selects gravity set for static load
CENTRIFUGAL	Selects a set for a centrifugal load
DEFORM	Selects a set for a deformation load
P2G	Selects a DMIG matrix to add to static loading.
ESLOAD	Selects an equivalent static load based on an external reader
BOLTSUB	Selects a a loadcase to use for bolt preloading

Analysis Constraint Selection

SPC	Selects set of single-point constraints
MPC	Selects set of multipoint constraints

Inertia Relief Support Degrees of Freedom

SUPORT	Selects a set of support degrees of freedom for inertia relief analysis

Frequency Solution Conditions

METHOD	Selects conditions for frequency analysis

Guyan Reduction Free Degrees of Freedom and Mass Matrix

ASET	Selects a set of free degrees of freedom for Guyan reduction
MAAUSER	Select the use of a user supplied mass matrix for Guyan reduction

Craig-Bampton Modes for Guyan Reduction

CBMETHOD	Selects conditions for frequency analysis to calculate Craig-Bampton modes

QSET	Select GRIDs and components for storing a set of generalized degrees of freedoms

Buckling Solution Selection

STATSUB	Selects the static loadcase
METHOD	Selects conditions for frequency analysis

Heat Transfer Load Selection

HEAT	Selects heat transfer load condition

Direct Frequency Response Load Selection

DLOAD	Selects dynamic load condition
FREQUENCY	Selects dynamic loading frequencies

Modal Frequency Response Load Selection

DLOAD	Selects dynamic load condition
FREQUENCY	Selects dynamic loading frequencies
SDAMPING	Selects modal damping table
MODES	Selects mode shape calculation LOADCASE
MODESELECT	Removes selected modes from the modal basis

Modal Frequency Response Load Selection (Alternate Style)

DLOAD	Selects dynamic load condition
FREQUENCY	Selects dynamic loading frequencies
SDAMPING	Selects modal damping table
METHOD	Selects conditions for frequency analysis

Random Response Calculation Control(

RANDOM	Selects random response control data

Nonlinear Static Calculation Control(

BCONTACT	Selects contact surface pair set
NLPARM	Selects nonlinear static control data

Auxilliary Matrix Selection

B2GG	Selects a DMIG matrix to add to the system viscous damping matrix
K2GG	Selects a DMIG matrix to add to the system stiffness matrix
K2PP	Selects a DMIG matrix to add to the system dynamic matrix for frequency response analysis.
K42GG	Selects a DMIG matrix to add to the system structural damping matrix
M2GG	Selects a DMIG matrix to add to the system mass matrix.

Nonstructural Mass Selectionl

NSM	Selects nonstructural mass to be used in selected elements.

Automatic SPC Controll

NSM	Selects nonstructural mass to be used in selected elements.

5.24 Output Selection

Analysis Output Control

TITLE	Specifies text for second line on each printed page
SUBTITLE	Specifies text for third line on each printed page
LABEL	Specifies text for fourth line on each printed page which contains analysis results
LINE	Sets the number of data lines and columns per printed page. Default is installation dependent - usually 64 lines/page and 132 columns
ECHO	Requests echo of the input data
ECHOON	Marks a point in the input to turn on ECHO
ECHOOFF	Marks a point in the input to turn off ECHO
SUMMARY	Controls the printing of analysis and design summary tables
MASS	Controls the printing of a mass summary table
VOLUME	Controls the printing of a volume summary table
TIMES	Controls the printing of CPU and elapsed times
DYNOUTPUT	Controls the printing of dynamic analysis results. Used to select rectangular or polar format.
POSTOUTPUT	Controls the output coordinate system for grid point results in the post processing file.

Set Definition

SET	Defines list of grid numbers, element numbers, or mode numbers for use of output requests

Analysis Output Requests

ACCELERATION	Requests dynamic accelerations for a set of grid points. Output is in the general coordinate system.
OLOAD	Selects a set of applied loads or applied fluxes for output in static and heat transfer analysis. Not used for loadcoms. Output is in the general coordinate system. OLOAD is not used in dynamic response.
SPCFORCE	Requests the static analysis single-point forces of constraint for a set of points (reaction forces). Not used for loadcoms. Output is in the general coordinate system. In heat transfer analysis, requests the reaction fluxes for a set of points. SPCFORCES is not used in dynamic response.
DISPLACEMENT	Requests displacements for a set of grid points. Output is in the general coordinate system.
PRESSURE	Same as DISPLACEMENT
VECTOR	Same as DISPLACEMENT
FORCE	Requests the forces for the structural elements. Output is in the element coordinate system. No output for solid, mass and rigid elements.
ESE	Requests the element strain energy. This request is available for statics and for frequency calculation load cases.
MCONTRIB	Requests modal contribution tables. This request is available for modal frequency response load cases.
STRESS	Requests the stresses for a set of structural elements. Output is in the element coordinate system, except for solid elements where output is in the material coordinate system. No output for mass, damping or rigid elements.
GSTRESS	Requests the stresses for a set of grids corresponding to solid elements. Output is in the basic coordinate system.
STRAIN	Requests the strains for a set of structural elements. Output is in the element coordinate system, except for solid elements where output is in the material coordinate system. No output for scalar, line, mass, damping or rigid elements.
SVECTOR	Requests solution set mode shape output. Output is in the general coordinate system.
THERMAL	Request temperatures for a set of grid points.
VELOCITY	Requests dynamic velocities for a set of grid points. Output is in the general coordinate system.
CDISP	Requests the contact clearances for nonlinear contact analysis.
CFORCE	Requests the contact forces for nonlinear contact analysis.
CPRESSURE	Requests the contact pressures for nonlinear contact analysis.

5

User Function of Dynamic Results Requests (Results related to UFDATA)

UFDISP	Requests the user linear combination of dynamic displacements for a set of field points.
UFVELO	Requests the user linear combination of dynamic velocities for a set of field points.
UFACCE	Requests the user linear combination of dynamic accelerations for a set of field points.

Grid Mass Matrices Output Requests

GRMASS	Requests the printing of the grid mass matrix

Reduced Matrices Output Requests

ALOAD	Requests the printing of the reduced load vector
KAA	Requests the printing of the reduced stiffness matrix
K4AA	Requests the printing of the reduced elemental structural damping matrix
MAA	Requests the printing of the reduced mass matrix

5.25 Summary of Loadcase Definitions

INFORMATION		STATIC	STATIC LOADCOM	NATURAL FRE-QUENCY	BUCKLING	DIRECT DYNAMIC	MODAL DYNAMIC	HEAT TRANSFER
LOADCASE ACTIVATORS	BCONTACT	X		X		X	X	
	BOLTSUB	X						
	CBMETHOD			X				
	CENTRIFU-GAL	X						
	DEFORM	X						
	DLOAD					X	X	
	ESLOAD	X						
	FREQUENCY					X	X	
	GRAVITY	X						
	HEAT							X
	LOAD	X						
	LOADSEQ		X					
	METHOD			X	X		X	
	MODES						X	
	MODESE-LECT						X	
	NLPARM	X						
	SDAMPING						X	
	STATSUB				X			
	RANDOM					X	X	
	TEMPERA-TURE	X						
BOUNDARY CONDITION ACTIVATORS	MPC	X		X		X	X	X
	SPC	X		X		X	X	X
SPECIAL DOF SET ACTIVATORS	ASET			X				
	BOUNDARY	X		X				
	QSET			X				
	SUPORT	X						
USER MASS	MAAUSER			X				

INFORMATION		LOADCASE TYPE						
		STATIC	STATIC LOADCOM	NATURAL FRE-QUENCY	BUCKLING	DIRECT DYNAMIC	MODAL DYNAMIC	HEAT TRANSFER
OUTPUT REQUEST ACTIVATORS	**ACCELERA-TION**					**X**	**X**	
	ALOAD	**X**						
	CDISP	**X**						
	CFORCE	**X**						
	CPRESSURE	**X**						
	DISPLACE-MENT	**X**	**X**	**X**	**X**	**X**	**X**	
	ERP					**X**	**X**	
	ESE	**X**	**X**	**X**				
	FORCE	**X**	**X**			**X**	**X**	
	GSTRESS	**X**	**X**			**X**	**X**	
	KAA	**X**		**X**				
	K4AA	**X**		**X**				
	MAA	**X**		**X**				
	MCONTRIB						**X**	
	OLOAD	**X**						**X**
	SPCFORCE	**X**						
	STRAIN	**X**	**X**			**X**	**X**	
	STRESS	**X**	**X**			**X**	**X**	
	SVECTOR			**X**	**X**			
	THERMAL							**X**
	VELOCITY					**X**	**X**	
	UFACCE					**X**	**X**	
	UFDISP					**X**	**X**	
	UFVELO					**X**	**X**	

5.26 Solution Control Data

The format of the Solution Control data is free-field. In presenting general formats for each entry embodying all options, the following conventions are used:

1. Upper-case letters must be typed as shown.
2. Lower-case letters indicate that a substitution must be made.
3. Braces { } indicate that a choice of contents is mandatory.
4. Brackets [ ] contain an option that may be omitted or included by the user.
5. Bold options or values are the default values.
6. Physical data entry consists of information input in columns 1 through 72 of a data entry. Most Solution Control data is limited to a single physical entry.
7. Logical entry may have more than 72 columns with the use of continuation data.
8. Comment lines can be input by using the dollar sign ($) in the first column of the line.

If the first four characters of a mnemonic are unique relative to all other Solution Control entries, the characters following can be omitted.

5.26.1 $

Solution Control Entry: **$ -** Comment

Description: Enter a comment line.

Format:

$ any character data

Example:

```
$ This line is a comment.
```

5.26.2 ACCELERATION

Solution Control Entry: **ACCELERATION** - Analysis Output Request

Description: Requests acceleration vector output

Format:

$$\mathrm{ACCELERATION} = \left\{ \begin{matrix} \textbf{NONE} \\ \mathrm{n} \\ \mathrm{ALL} \\ \mathrm{POST} \\ \mathrm{POST,\ n} \\ \mathrm{POST,\ ALL} \\ \mathrm{BOTH} \\ \mathrm{BOTH,\ n} \\ \mathrm{BOTH,\ ALL} \end{matrix} \right\}$$

Alternate Format:

$$\mathrm{ACCE}\left(\left[\left\{ \begin{matrix} \mathrm{SORT1} \\ \mathrm{SORT2} \end{matrix} \right\},\right]\left\{ \begin{matrix} \mathrm{PRINT} \\ \mathrm{PUNCH} \\ \mathrm{PLOT} \\ \mathrm{PRINT,\ PLOT} \\ \mathrm{PRINT,\ PUNCH} \end{matrix} \right\}\left[,\left\{ \begin{matrix} \mathrm{RPRINT} \\ \mathrm{RPUNCH} \\ \mathrm{RPRINT,\ RPUNCH} \end{matrix} \right\},\left\{ \begin{matrix} \mathrm{PSDF} \\ \mathrm{ATOC} \\ \mathrm{CRMS} \\ \mathrm{PSDF,\ ATOC,\ CRMS} \\ \mathrm{RALL} \end{matrix} \right\}\right]\right) = \left\{ \begin{matrix} \textbf{NONE} \\ \mathrm{n} \\ \mathrm{ALL} \end{matrix} \right\}$$

Examples:

```
ACCELERATION = 5
ACCELERATION(PRINT,PUNCH) = 17
ACCELERATION(PLOT) = ALL
ACCE(RPRINT,PSDF,CRMS) = ALL
ACCE = POST
```

Option	Meaning
NONE	Default. No accelerations will be output.
n	Set identification of previously appearing SET data. Only accelerations of points whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	Accelerations for all points will be output. Both magnitude and phase as well as real and imaginary components can be output. See DYNOUTPUT command.
POST	Accelerations for all points will be output to the post processing file.
POST, n	Accelerations of points in set n will be output to the post processing file. n should be integer > 0 or blank.
POST, ALL	Same as POST.

Option	Meaning
BOTH	Accelerations of all points will be output to both the output file and the post-processing file.
BOTH, n	Accelerations of points in set n will be output to both the output file and the post-processing file. n should be integer > 0 or blank.
BOTH, ALL	Same as BOTH.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2..
SORT1	Requests random results be sorted first by grid ID and second by frequency.
SORT2	Requests random results be sorted first by frequency and second by grid ID.
RPRINT	Requests random results printed to the output file.
RPUNCH	Requests random results printed to the punch post-processing file.
PSDF	Requests output for power spectral density function from random analysis
ATOC	Requests output for autocorrelation functions from random analysis
CRMS	Requests output for cumulative root mean square, root mean square (RMS) and number of zero crossings (N0) from random analysis.
RALL	Request out for PSDF, ATOC and CRMS

Remarks:

1. Acceleration output is only available for dynamic analysis.
2. ACCELERATION = NONE allows overriding an overall output request.
3. Accelerations written to the output file are always in the general coordinate system. The coordinate system used for accelerations written to the post-processing file is controlled by the **POSTOUTPUT** solution control command (default = basic).
4. When the POST command is used, no results are printed in the output file.
5. When the POST command is used, the **POST** command must also appear in the executive section to create the post processing output file.
6. Either magnitude and phase or real and imaginary components can be output. See **DYNOUTPUT** (p. 275).
7. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

8. The RPRINT, RPUNCH, PSDF, ATOC, CRMS and RALL options can only be used in frequency response loadcases that contains the **RANDOM** solution control command.

5.26.3 ALOAD

Solution Control Entry: **ALOAD** - Reduced Load Output Request

Description: Requests printing of the reduced load vector.

Format:

$$\text{ALOAD} = \left\{ \begin{matrix} \text{DMIG} \\ \textbf{NONE} \end{matrix} \right\}$$

Example:

```
ALOAD = DMIG
```

Option	Meaning
DMIG	The reduced load vector will be output to the DMIG post processing file.
NONE	Default. Do not output the reduced load vector.

Remarks:

1. The ALOAD command can only be used in conjuction with the **REDUCE** executive control command and the **BOUNDARY** solution control command.
2. The reduced load is the static condensation of the total applied load to the boundary degrees of freedom. The total applied load is the sum of external (**LOAD**), centrifugal (**CENTRIFUGAL**), thermal (**TEMPERATURE**), gravity (**GRAVITY**) and deformation (**DEFORM**).
3. The DMIG post-processing file is named *pname*xx.DMIG where *pname* is the project name and xx is the design cycle. This file contains data according to the **DMIG** bulk data format.
4. The reduced loads for all static loadcases with the ALOAD output request will be put in a DMIG matrix named PASET. Each static loadcase will correspond to a different column in the matrix.

5.26.4 ASET

Solution Control Entry: **ASET** - Natural Frequency Loadcase Control

Description: Select the independent degrees of freedom used by a natural frequency loadcase.

Format:

ASET = n

Example:

```
ASET = 10
```

Option	Meaning
n	Set identification of ASET and hence must appear on a **ASET2** or **ASET3** entry in the bulk data (Integer>0).

Remarks:

1. ASET2 and ASET3 data will not be used unless selected in the Solution Control.
2. The ASET command can only be used in a eigenvalue loadcases.
3. *GENESIS* uses the Guyan reduction method to reduce the stiffness and mass matrices.

5.26.5 AUTOSPC

Solution Control Entry: **AUTOSPC** - Control Automatic SPC

Description: Sets parameters to control automatic single point constraints for low stiffness degrees of freedom.

Format:

$$\text{AUTOSPC} = \begin{Bmatrix} \text{YES} \\ \textbf{NO} \end{Bmatrix}$$

Alternate Format:

$$\text{AUTOSPC}\left(\left[\begin{Bmatrix} \textbf{PRINT} \\ \text{NOPRINT} \end{Bmatrix}\right][, \text{EPS} = \text{v}]\right) = \begin{Bmatrix} \text{YES} \\ \textbf{NO} \end{Bmatrix}$$

Examples:

```
AUTOSPC = YES
AUTOSPC(NOPRINT) = YES
AUTOSPC(PRINT,EPS=1.0e-6) = YES
```

Option	Meaning
YES	Set analysis parameter **AUTOSPC** to YES.
NO	Set analysis parameter **AUTOSPC** to NO.
PRINT	Set analysis parameter **PRGPST** to YES.
NOPRINT	Set analysis parameter **PRGPST** to NO.
EPS = v	Set analysis parameter **EPZERO** to v. (Real > 0.0).

Remarks:

1. It is recommended to only use AUTOSPC above the first loadcase. While this command may be placed inside a loadcase, it is not loadcase-specific, and will apply to all loadcases. If multiple AUTOSPC commands appear in the solution control, subsequent entries will override earlier entries such that the last one will have effect.
2. This command changes the value of certain analysis parameters. If the bulk data also contains **PARAM** entries for these parameters, those will override any settings given here.

5.26.6 B2GG

Solution Control Entry: **B2GG** - External Matrix Selection

Description: Select DMIG matrices to add to the viscous damping matrix.

Format:

B2GG = *matrix*

Example:

```
B2GG = B0000001
```

Option	Meaning
matrix	Name of a matrix defined by **DMIG** bulk data entries (Character).

Remarks:

1. The B2GG command must appear above the first loadcase definition and affects the viscous damping for all frequency response loadcases.
2. The matrix identified by *matrix* must be real and symmetric (the DMIG header must specify 6 for the form and 1 or 2 for the type).
3. The matrix name may have at most 8 characters.
4. The matrix may be scaled using the analysis parameter **CB2**.

5.26.7 BCONTACT

Solution Control Entry: **BCONTACT** - Contact Set Selection

Description: Selects the contact surface pair set to define potential contact areas in the structural model

Format:

BCONTACT = n

Example:

```
BCONTACT= 5
```

Option	Meaning
n	Set identification of a contact surface pair set and hence must appear on a **BCPAIR** or **BCPADD** entry in the bulk data (Integer > 0)

Remarks:

1. BCPAIR data will not be used unless selected in Solution Control.
2. The BCONTACT command can either point to a bulk data BCPADD or to BCPAIR data.
3. In a static loadcase, BCONTACT triggers a nonlinear contact analysis. The **NLPARM** command must be used together with BCONTACT to set the nonlinear solution control parameters.
4. In a natural frequency or frequency response loadcase, BCONTACT triggers a linear, glue-like connection. A contact point of the referenced surface pairs will be linearly connected only if the initial offset at that point is less than or equal to zero. If the friction coefficient on a referenced BCPAIR is non-zero, the connected points between those surfaces will bond all three translation degrees of freedom. If the friction coefficient is zero, the connected points will only bond the direction normal to the surface faces.

5

5.26.8 BEGIN BULK

Solution Control Entry: **BEGIN BULK**- Mark the End of Solution Control

Description: Delimits the Solution Control section of the input file from the Bulk Data section.

Format:

BEGIN BULK

Example:

```
BEGIN BULK
```

Remarks:

1. The BEGIN BULK delimiter is required.

5

5.26.9 BOLTSUB

Solution Control Entry: **BOLTSUB** - Bolt Preloading Control

Description: Selects a static loadcase that defines bolt preloading.

Format:

BOLTSUB = n

Examples:

```
BOLTSUB = 7
BOLT = 5
```

Option	Meaning
n	Static **LOADCASE** number (Integer > 0).

Remarks:

1. The referenced bolt preloading loadcase must come before the loadcase that includes the BOLTSUB command.
2. BOLTSUB cannot reference load combinations (**LOADCOM**). The bulk data statement **LOAD** may be used to combine loads in a regular loadcase.
3. This command takes the displacements of bolt control grids calculated in the referenced loadcase and applies them as enforced displacements. If there are no **BOLT** elements in the model, then this command will do nothing. See **BOLT Element** (Sec. 2.8.3) a general discussion.

5.26.10 BOUNDARY

Solution Control Entry: **BOUNDARY** - Superelement Reduction Control

Description: Select the independent degrees of freedom to use for superelement reduction.

Format:

BOUNDARY = n

Example:

```
BOUNDARY = 10
```

Option	Meaning
n	Set identification of ASET and hence must appear on a **ASET2** or **ASET3** entry in the bulk data (Integer>0).

Remarks:

1. The BOUNDARY command can only be used in conjuction with the **REDUCE** executive control command.
2. The BOUNDARY command must appear above the first loadcase definition and is only applicable to static or natural frequency loadcases.
3. The stiffness and mass matrices and load vectors will be reduced to the degrees of freedom in the referenced ASET.

5.26.11 CBMETHOD

Solution Control Entry: **CBMETHOD** - Eigenvalue Calculation Method Selection

Description: Selects the frequency calculation parameters to be used to calculate Craig-Bampton modes in a Guyan reduction loadcase

Format:

CBMETHOD = n

Example:

```
CBMETHOD = 33
```

Option	Meaning
n	Set identification number of an **EIGR** or **EIGRL** statement. (Integer > 0)

Remarks:

1. This command must be used together with the **QSET**, **METHOD** and **ASET** commands.

5.26.12 CDISP

Solution Control Entry: **CDISP**- Analysis Output Request

Description: Requests form of contact clearance output

Format:

$$\text{CDISP} = \left\{ \begin{array}{c} \textbf{NONE} \\ \text{n} \\ \text{ALL} \\ \text{POST} \\ \text{POST, n} \\ \text{POST, ALL} \\ \text{BOTH} \\ \text{BOTH, n} \\ \text{BOTH, ALL} \end{array} \right\}$$

Alternate Format:

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

Examples:

```
CDISP= ALL
CDISP(PRINT,PUNCH) = 19
CDISP(PLOT) = ALL
CDISP=70
```

Option	Meaning
NONE	Default. No contact clearances will be output.
n	Set identification of previously appearing SET data. Only contact clearances at grids whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	All non-zero contact clearances at all grids will be output.
POST	Contact clearances at all grids will be output to the post processing file.
POST, n	Contact clearances of all grids in set n will be output to the post processing file. n should be integer > 0 or blank.
POST, ALL	Same as POST.
BOTH	Contact clearances at all grids will be output to both the output file and the post-processing file.

5

BOTH, n	Contact clearances at all grids in set n will be output to both the output file and the post-processing file. n should be integer > 0 or blank.
BOTH, ALL	Same as BOTH.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2.

Remarks:

1. Contact clearances are only calculated for nonlinear contact analysis loadcases. Thus, CDISP should only be used in loadcases that also have the **BCONTACT** command.
2. When the POST option is used, no contact clearance results are printed to the output file.
3. When the POST option is used, the **POST** command must be used in the executive section of the input data to generate the post processing file. Contact clearances are only available for PUNCH and OUTPUT2 post-processing formats. If another POST format is selected, no contact clearance results will be output.
4. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

5.26.13 CENTRIFUGAL

Solution Control Entry: **CENTRIFUGAL** - Static Load Selection

Description: Selects the external centrifugal static load set to be applied to the structural model

Format:

CENTRIFUGAL = n

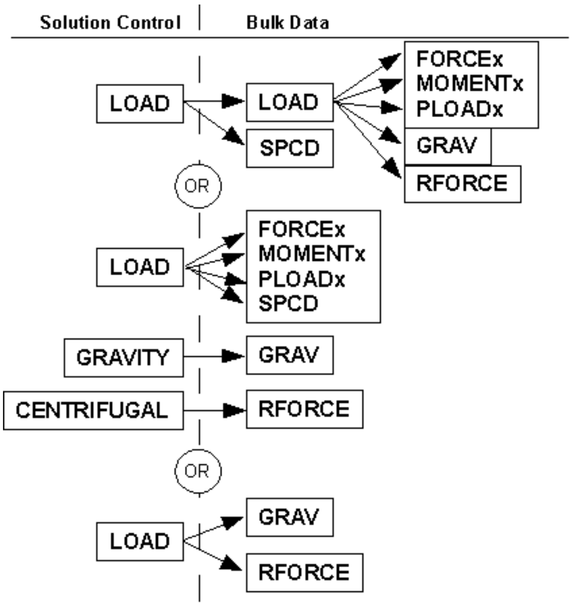
Example:

```
CENTRIFUGAL = 3
CENT = 8
```

Option	Meaning
n	Set identification of a unique **RFORCE** entry in the Bulk Data (Integer > 0).

Remarks:

1. The total load applied in a load case will be the sum of external (**LOAD**), thermal (**TEMPERATURE**), gravity (**GRAVITY**), centrifugal (**CENTRIFUGAL**) and deformation (**DEFORM**) loads.
2. Centrifugal loading can be selected in static load cases with **either** the Solution Control command **CENTRIFUGAL**=SID **or** with the Solution Control command **LOAD**=SID. If Solution Control LOAD points to an RFORCE or to a bulk data **LOAD** that references an RFORCE entry, then Solution Control CENTRIFUGAL must not be used.



5.26.14 CFORCE

Solution Control Entry: **CFORCE** - Analysis Output Request

Description: Requests form of contact force output

Format:

$$\mathrm{CFORCE} = \left\{ \begin{array}{c} \textbf{NONE} \\ \mathrm{n} \\ \mathrm{ALL} \\ \mathrm{POST} \\ \mathrm{POST,\ n} \\ \mathrm{POST,\ ALL} \\ \mathrm{BOTH} \\ \mathrm{BOTH,\ n} \\ \mathrm{BOTH,\ ALL} \end{array} \right\}$$

Alternate Format:

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

Examples:

```
CFORCE= ALL
CFORCE(PRINT,PUNCH) = 19
CFORCE(PLOT) = ALL
CFORCE=70
```

5

Option	Meaning
NONE	Default. No contact forces will be output.
n	Set identification of previously appearing SET data. Only contact forces at grids whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	All non-zero contact forces at all grids will be output.
POST	Contact forces at all grids will be output to the post processing file.
POST, n	Contact forces of all grids in set n will be output to the post processing file. n should be integer > 0 or blank.
POST, ALL	Same as POST.
BOTH	Contact forces at all grids will be output to both the output file and the post-processing file.

BOTH, n	Contact forces at all grids in set n will be output to both the output file and the post-processing file. n should be integer > 0 or blank.
BOTH, ALL	Same as BOTH.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2.

Remarks:

1. Contact forces are only calculated for nonlinear contact analysis loadcases. Thus, CFORCE should only be used in loadcases that also have the **BCONTACT** command.
2. When the POST option is used, no contact force results are printed to the output file.
3. When the POST option is used, the **POST** command must be used in the executive section of the input data to generate the post processing file. Contact forces are only available for PUNCH and OUTPUT2 post-processing formats. If another POST format is selected, no contact force results will be output.
4. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

5.26.15 CPRESSURE

Solution Control Entry: **CPRESSURE**- Analysis Output Request

Description: Requests form of contact pressure output

Format:

$$\text{CPRESSURE} = \left\{ \begin{array}{c} \textbf{NONE} \\ \text{n} \\ \text{ALL} \\ \text{POST} \\ \text{POST, n} \\ \text{POST, ALL} \\ \text{BOTH} \\ \text{BOTH, n} \\ \text{BOTH, ALL} \end{array} \right\}$$

Alternate Format:

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

Examples:

```
CPRESSURE= ALL
CPRESSURE(PRINT,PUNCH) = 10
CPRESSURE(PLOT) = ALL
CPRESSURE=5
```

Option	Meaning
NONE	Default. No contact pressure will be output.
n	Set identification of previously appearing SET data. Only contact pressures at grids whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	All non-zero contact pressures at all grids will be output.
POST	Contact pressures at all grids will be output to the post processing file.
POST, n	Contact pressures at all grids in set n will be output to the post processing file. n should be integer > 0 or blank.
POST, ALL	Same as POST.
BOTH	Contact pressures at all grids will be output to both the output file and the post-processing file.

BOTH, n	Contact pressures at all grids in set n will be output to both the output file and the post-processing file. n should be integer > 0 or blank.
BOTH, ALL	Same as BOTH.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2.

Remarks:

1. Contact pressures are only calculated for nonlinear contact analysis loadcases. Thus, CPRESSURE should only be used in loadcases that also have the **BCONTACT** command.
2. When the POST option is used, no contact pressure results are printed to the output file.
3. When the POST option is used, the **POST** command must be used in the executive section of the input data to generate the post processing file. Contact pressure is only available for PUNCH and OUTPUT2 post-processing formats. If another POST format is selected, no contact pressure results will be output.
4. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

5.26.16 DEFORM

Solution Control Entry: **DEFORM** - Static Load Selection

Description: Selects inital element deformations

Format:

DEFORM = n

Example:

```
DEFORM = 3
DEFO = 8
```

Option	Meaning
n	Set identification of **DEFORM** entries in the Bulk Data (Integer > 0).

Remarks:

1. The total load applied in a load case will be the sum of external (**LOAD**), thermal (**TEMPERATURE**), gravity (**GRAVITY**), centrifugal (**CENTRIFUGAL**) and deformation (**DEFORM**) loads.
2. DEFORM entries in the Bulk Data will not be used unless they are activated with the DEFORM statement in the Solution Control.
3. DEFORM data can be used directly in static and inertia relief loadcases and indirectly in buckling loadcases.

5.26.17 DISPLACEMENT

Solution Control Entry: **DISPLACEMENT** - Analysis Output Request

Description: Requests form of displacement vector output

Format:

$$\text{DISPLACEMENT} = \left\{ \begin{array}{c} \textbf{NONE} \\ \text{POST} \\ \text{BOTH} \\ \left[\left\{ \begin{array}{c} \text{POST} \\ \text{BOTH} \end{array} \right\}, \right] \left\{ \begin{array}{c} n \\ \text{ALL} \end{array} \right\} \end{array} \right\}$$

Alternate Format:

$$\text{DISP}\left(\left[\left\{ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right\}, \right] \left\{ \begin{array}{c} \text{PRINT} \\ \text{PUNCH} \\ \text{PLOT} \\ \text{PRINT, PLOT} \\ \text{PRINT, PUNCH} \end{array} \right\} \left[, \left\{ \begin{array}{c} \text{RPRINT} \\ \text{RPUNCH} \\ \text{RPRINT, RPUNCH} \end{array} \right\} , \left\{ \begin{array}{c} \text{PSDF} \\ \text{ATOC} \\ \text{CRMS} \\ \text{PSDF, ATOC, CRMS} \\ \text{RALL} \end{array} \right\} \right] \right) = \left\{ \begin{array}{c} \textbf{NONE} \\ n \\ \text{ALL} \end{array} \right\}$$

Eigenvalue Format:

$$\text{DISPLACEMENT} = \left\{ \begin{array}{c} \textbf{NONE} \\ \text{POST} \\ \text{BOTH} \\ \left[\left\{ \begin{array}{c} \text{POST} \\ \text{BOTH} \end{array} \right\}, \right] \left\{ \begin{array}{c} n \\ \text{ALL} \end{array} \right\} \left[, \left\{ \begin{array}{c} m \\ \text{ALL} \end{array} \right\} \right] \end{array} \right\}$$

Eigenvalue Alternate Format:

$$\text{DISP}\left(\left\{ \begin{array}{c} \text{PRINT} \\ \text{PUNCH} \\ \text{PLOT} \\ \text{PRINT, PLOT} \\ \text{PRINT, PUNCH} \end{array} \right\} \right) = \left\{ \begin{array}{c} \textbf{NONE} \\ \left\{ \begin{array}{c} n \\ \text{ALL} \end{array} \right\} \left[, \left\{ \begin{array}{c} m \\ \text{ALL} \end{array} \right\} \right] \end{array} \right\}$$

Examples:

```
DISPLACEMENT = 5
DISPLACEMENT(PRINT,PUNCH) = 17
DISPLACEMENT(PLOT) = ALL
DISP = ALL
```

Option	**Meaning**

NONE	Default. No displacements will be output.
n	Set identification of previously appearing SET data. Only displacements of grids whose identification numbers appear in the SET data will be output (Integer > 0).
m	Set identification of previously appearing SET data. Only mode numbers that appear in the SET data will be output (Integer > 0 or blank, default = ALL).
ALL	Displacements for all points will be output.
POST	Displacements will be output only to the post processing file.
BOTH	Displacements will be output to both the output file and the post-processing file.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2..
SORT1	Requests random results be sorted first by grid ID and second by frequency.
SORT2	Requests random results be sorted first by frequency and second by grid ID.
RPRINT	Requests random results printed to the output file.
RPUNCH	Requests random results printed to the punch post-processing file.
PSDF	Requests output for power spectral density function from random analysis
ATOC	Requests output for autocorrelation functions from random analysis
CRMS	Requests output for cumulative root mean square, root mean square (RMS) and number of zero crossings (N0) from random analysis.
RALL	Request out for PSDF, ATOC and CRMS

Remarks:

1. **VECTOR** and **PRESSURE** are alternate forms and are entirely equivalent to DISPLACEMENT.
2. Displacements written to the output file are always in the general coordinate system. The coordinate system used for displacements written to the post-processing file is controlled by the **POSTOUTPUT** solution control command (default = basic).
3. This option can also be used to print mode shapes when using the **METHOD** statement in the load case.
4. When the POST option is used, no displacement results are printed to the output file.

5. For dynamic analysis, either magnitude and phase or real and imaginary components can be output. See **DYNOUTPUT**.
6. When the POST option is used, the **POST** command must be used in the executive section of the input data in order to generate the post processing file.
7. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.
8. The RPRINT, RPUNCH, PSDF, ATOC, CRMS and RALL options can only be used in frequency response loadcases that contain the **RANDOM** solution control command.
9. When using the DISPLACEMENT = n format (no parenthesized options plus an output grid set), and the POST executive control command is used to define a post-processing format, then the output set is only used when writing results to the output file. Results from all grids are written to the post-processing file. To have only the grids in the set written to the post-processing file, use the DISPLACEMENT = BOTH,n format.

5.26.18 DLOAD

Solution Control Entry: **DLOAD** - Dynamic Load Selection

Description: Selects the dynamic load to be applied to a Frequency Response loadcase.

Format:

DLOAD = n

Examples:

```
DLOAD = 73
```

Option	Meaning
n	Set identification of a **DLOAD** bulk data entry or of **RLOAD1**, **RLOAD2** and/or **RLOAD3** bulk data entries for frequency response loadcases (Integer > 0).

5

5.26.19 DYNOUTPUT

Solution Control Entry: **DYNOUTPUT** - Analysis Output Control

Description: Requests printing of real and imaginary or magnitude and phase printing of dynamic analysis results

Format:

$$\text{DYNOUTPUT} = \begin{Bmatrix} \textbf{RECTANGULAR} \\ \text{POLAR} \end{Bmatrix}$$

Examples:

```
DYNOUTPUT = RECTANGULAR
DYNOUTPUT = POLAR
```

Option	Meaning
RECTANGULAR	Default. Real and imaginary components will be printed.
POLAR	Magnitude and phase components will be printed.

Remarks:

1. If no DYNOUTPUT command appears, real and imaginary components will be printed.

5

5.26.20 ECHO

Solution Control Entry: **ECHO** - Output Control

Description: Requests echo of input data

Format:

$$\text{ECHO} = \begin{Bmatrix} \text{BOTH} \\ \text{SORT} \\ \text{UNSORT} \\ \textbf{NONE} \end{Bmatrix}$$

Alternate Format 1:

$$\text{ECHO} = \begin{Bmatrix} \text{BOTH(bd1,bd2,...)} \\ \text{SORT(bd1,bd2,...)} \\ \text{UNSORT(bd1,bd2,...)} \end{Bmatrix}$$

Alternate Format 2:

$$\text{ECHO} = \begin{Bmatrix} \text{BOTH(EXCEPT bd1,bd2,...)} \\ \text{SORT(EXCEPT bd1,bd2,...)} \\ \text{UNSORT(EXCEPT bd1,bd2,...)} \end{Bmatrix}$$

Examples:

```
ECHO = UNSORT
ECHO = SORT(DOPT,EIGRL)
ECHO = BOTH(EXCEPT GRID, CQUAD4, PSHELL)
```

Option	Meaning
BOTH	Both sorted and unsorted echo will be printed.
SORT	Sorted echo (ordered by type of input data) will be printed. This option prints updated values by initial design data if the optimization data present.
UNSORT	Unsorted echo will be printed. This option prints output that resembles the input data.
NONE	Default. No echo will be printed.
bdi	Bulk data entry names.
EXCEPT	Print all data excluding bulk data entry names listed after EXCEPT

5

Remarks:

1. Portions of the unsorted echo can be selectively printed using the **ECHOON** and **ECHOOFF** commands. ECHOON starts the printing and ECHOOFF stops the printing. Multiple pairs of ECHOON and ECHOOFF commands are allowed.

5.26.21 ECHOON

Solution Control Entry: **ECHOON** - Output Control

Description: Requests the unsorted echo of input data to be printed in the output file from the ECHOON command until an ECHOOFF command is encountered

Format:

ECHOON

Example:

```
ECHOON
```

Remarks:

1. The **ECHOOFF** command stops the printing of unsorted **ECHO**.
2. Multiple pairs of ECHOON and ECHOOFF commands are allowed.

5.26.22 ECHOOFF

Solution Control Entry: **ECHOOFF** - Output Control

Description: Requests that the printing of unsorted echo of input data in the output file be stopped

Format:

ECHOOFF

Example:

```
ECHOOFF
```

Remarks:

1. The **ECHOON** command starts the printing of unsorted **ECHO**.
2. Multiple pairs of ECHOON and ECHOOFF commands are allowed.

5

5.26.23 ERP

Solution Control Entry: **ERP**- Analysis Output Request

Description: Requests form of equivalent radiated power output

Format:

$$\text{ERP} = \begin{Bmatrix} \textbf{NONE} \\ \text{ALL} \\ \text{POST} \\ \text{POST, ALL} \\ \text{BOTH} \\ \text{BOTH, ALL} \end{Bmatrix}$$

Alternate Format:

$$\text{ERP}\left(\left[\begin{Bmatrix} \text{SORT1} \\ \text{SORT2} \end{Bmatrix},\right]\begin{Bmatrix} \text{PRINT} \\ \text{PUNCH} \\ \text{PLOT} \\ \text{PRINT, PLOT} \\ \text{PRINT, PUNCH} \end{Bmatrix}\right) = \begin{Bmatrix} \textbf{NONE} \\ \text{ALL} \end{Bmatrix}$$

Examples:

```
ERP= ALL
ERP(PRINT,PUNCH) = ALL
```

Option	Meaning
NONE	Default. No equivalent radiated power results will be output.
ALL	Results for all panels will be output.
POST	Results for all panels will be output to the post processing file.
POST, ALL	Same as POST.
BOTH	Results for all panels will be output to both the output file and the post-processing file.
BOTH, ALL	Same as BOTH.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2..
SORT1	Requests results be sorted first by panel name and second by frequency.
SORT2	Requests results be sorted first by frequency and second by panel name.
ALL	Results for all panels will be output.

Remarks:

1. ERP output is only available for dynamic analysis.
2. **ERPPNL** bulk data must exist to define panels for calculation of equivalent radiated power.
3. ERP = NONE allows overriding an overall output request.
4. When the POST command is used, no results are printed in the output file.
5. When the POST command is used, the **POST** command must also appear in the executive section to create the post processing output file. In the current version, only the PUNCH and OUTPUT2 formats are supported.
6. The SORT1 option is not supported for the PUNCH or OUTPUT2 formats.
7. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

5.26.24 ESE

Solution Control Entry: **ESE** - Analysis Output Request

Description: Requests form of element strain energy output

Format:

$$\mathrm{ESE} = \left\{ \begin{array}{c} \textbf{NONE} \\ \mathrm{POST} \\ \mathrm{BOTH} \\ \left[\left\{ \begin{array}{c} \mathrm{POST} \\ \mathrm{BOTH} \end{array} \right\}, \right] \left\{ \begin{array}{c} \mathrm{n} \\ \mathrm{ALL} \end{array} \right\} \end{array} \right\}$$

Alternate Format:

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

Eigenvalue Format:

$$\mathrm{ESE} = \left\{ \begin{array}{c} \textbf{NONE} \\ \mathrm{POST} \\ \mathrm{BOTH} \\ \left[\left\{ \begin{array}{c} \mathrm{POST} \\ \mathrm{BOTH} \end{array} \right\}, \right] \left\{ \begin{array}{c} \mathrm{n} \\ \mathrm{ALL} \end{array} \right\} \left[, \left\{ \begin{array}{c} \mathrm{m} \\ \mathrm{ALL} \end{array} \right\} \right] \end{array} \right\}$$

Eigenvalue Alternate Format:

$$\mathrm{ESE}\left(\left\{ \begin{array}{c} \mathrm{PRINT} \\ \mathrm{PUNCH} \\ \mathrm{PLOT} \\ \mathrm{PRINT, PLOT} \\ \mathrm{PRINT, PUNCH} \end{array} \right\} \right) = \left\{ \begin{array}{c} \textbf{NONE} \\ \left\{ \begin{array}{c} \mathrm{n} \\ \mathrm{ALL} \end{array} \right\} \left[, \left\{ \begin{array}{c} \mathrm{m} \\ \mathrm{ALL} \end{array} \right\} \right] \end{array} \right\}$$

Examples:

```
ESE = ALL
ESE(PRINT,PUNCH) = 17
ESE(PLOT) = ALL
ESE = 2
```

```
ESE = POST, 3
```

Option	Meaning
NONE	Default. No element strain energy will be output.
n	Set identification of previously appearing SET data. Only strain energy for all elastic elements of modes whose identification numbers appear in the SET data will be output (Integer > 0).
m	Set identification of previously appearing SET data. Only mode numbers that appear in the SET data will be output (Integer > 0 or blank, default = ALL).
ALL	Strain energy for all elastic elements for all modes will be output.
POST	Elastic element strain energy will be output to the post processing file.
BOTH	Elastic element strain energy will be output to both the output file and the post-processing file.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2..

Remarks:

1. Element strain energy is used in static calculation load cases and frequency calculation load cases.
2. The element strain energy is calculated using the following equation);

   $$ESE = \frac{\phi_i^T [K]_j \phi_i}{2} \quad \text{(for frequency calculation loadcases}$$

   where
   ϕ_i is the mode shape associated with mode i. The mode shape is mass normalized.
   $[K]_j$ is the stiffness matrix of element j.
   or

   $$ESE_j = \frac{1}{2} U_j^T [K_j] U_j \quad \text{(for static analysis)}$$

   where
   U_j is the displacement associated with element j.
   $[K_j]$ is the stiffness matrix of element j.
3. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

5

5.26.25 ESLOAD

Solution Control Entry: **ESLOAD -** Equivalent Static Load Selection

Description: Define an equivalent static load based on data returned from an external ESL reader.

Format:

ESLOAD = rid, character data

Examples:

```
ESLOAD = 25, time=1.0E-2
ESLOAD = 7, Condition=timestep:112
```

Option	Meaning
rid	ESL reader identification number (Integer >0).

Remarks:

1. ESLOAD entries must not occur before the first LOADCASE definition.
2. The ESL reader identification number must have been defined by an **ESLDISP** executive control entry.
3. ESLOAD may not be combined with any other static loadcase activator (**LOAD**, **TEMPERATURE**, **GRAVITY**, **CENTRIFUGAL** and **DEFORM**).
4. The format of the character data is dependent on the specific ESL reader used. Consult the documentation for the selected ESL reader module to determine what data is required.

5.26.26 FORCE

Solution Control Entry: **FORCE** - Analysis Output Request

Description: Requests form of element force output

Format:

$$\mathrm{FORCE} = \left\{ \begin{array}{c} \textbf{NONE} \\ \mathrm{n} \\ \mathrm{ALL} \\ \mathrm{POST} \\ \mathrm{POST,\ n} \\ \mathrm{POST,\ ALL} \\ \mathrm{BOTH} \\ \mathrm{BOTH,\ n} \\ \mathrm{BOTH,\ ALL} \end{array} \right\}$$

Alternate Format:

$$\mathrm{FORCE}\left(\left[\left\{ \begin{array}{c} \mathrm{SORT1} \\ \mathrm{SORT2} \end{array} \right\},\right] \left\{ \begin{array}{c} \mathrm{PRINT} \\ \mathrm{PUNCH} \\ \mathrm{PLOT} \\ \mathrm{PRINT,\ PLOT} \\ \mathrm{PRINT,\ PUNCH} \end{array} \right\} \left[, \left\{ \begin{array}{c} \mathrm{RPRINT} \\ \mathrm{RPUNCH} \\ \mathrm{RPRINT,\ RPUNCH} \end{array} \right\}, \left\{ \begin{array}{c} \mathrm{PSDF} \\ \mathrm{ATOC} \\ \mathrm{CRMS} \\ \mathrm{PSDF,\ ATOC,\ CRMS} \\ \mathrm{RALL} \end{array} \right\} \right]\right) = \left\{ \begin{array}{c} \textbf{NONE} \\ \mathrm{n} \\ \mathrm{ALL} \end{array} \right\}$$

Examples:

```
FORCE= ALL
FORCE(PRINT,PUNCH) = 17
FORCE(PLOT) = ALL
FORCE=25
```

Option	Meaning
NONE	Default. No forces will be output.
n	Set identification of previously appearing SET data. Only element forces of elements whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	Forces for all elements will be output.
POST	Forces for all elements will be output to the post processing file.
POST, n	Forces for all elements in set n will be output to the post processing file. n should be integer > 0 or blank.
POST, ALL	Same as POST.
BOTH	Forces for all elements will be output to both the output file and the post-processing file.

BOTH, n	Forces for all elements in set n will be output to both the output file and the post-processing file. n should be integer > 0 or blank.
BOTH, ALL	Same as BOTH.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2..
SORT1	Requests random results be sorted first by element ID and second by frequency.
SORT2	Requests random results be sorted first by frequency and second by element ID.
RPUNCH	Requests random results printed to the punch post-processing file.
PSDF	Requests output for power spectral density function from random analysis
ATOC	Requests output for autocorrelation functions from random analysis
CRMS	Requests output for cumulative root mean square, root mean square (RMS) and number of zero crossings (N0) from random analysis.
RALL	Request out for PSDF, ATOC and CRMS

Remarks:

1. Element forces are printed in the element coordinate system.
2. FORCE produces output for scalar, linear and shell elements.
3. When the POST option is used, no force results are printed to the output file.
4. When the POST option is used, the **POST** command must be used in the executive section of the input data to generate the post processing file.
5. For dynamic analysis, either magnitude and phase or real and imaginary components can be output. See **DYNOUTPUT**.
6. Dynamic forces are not recovered for composite elements.
7. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.
8. The RPRINT, RPUNCH, PSDF, ATOC, CRMS and RALL options can only be used in frequency response loadcases that contains the **RANDOM** solution control command.

9. When using the FORCE = n format (no parenthesized options plus an output element set), and the POST executive control command is used to define a post-processing format, then the output set is only used when writing results to the output file. Results from all elements are written to the post-processing file. To have only the elements in the set written to the post-processing file, use the FORCE = BOTH,n format.

5.26.27 FREQUENCY

Solution Control Entry: **FREQUENCY** - Dynamic Load Selection

Description: Selects the set of frequencies to be used in dynamic analysis

Format:

FREQUENCY = n

Examples:

```
FREQUENCY = 5
```

```
FREQ = 17
```

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

Remarks:

1. The FREQ, FREQ1 or FREQ2 Bulk Data will not be selected unless activated in Solution Control.
2. A frequency set selection is required for dynamic analysis.

5

5.26.28 GRAVITY

Solution Control Entry: **GRAVITY** - Static Load Selection

Description: Selects the gravity set to be applied to the structural model.

Format:

GRAVITY = n

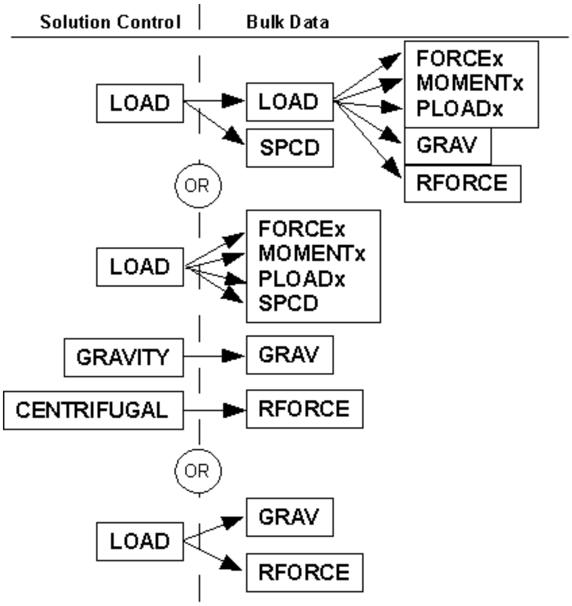
Example:

```
GRAVITY = 1
GRAV = 3
```

Option	Meaning
n	Set identification of a unique **GRAV** entry in the Bulk Data (Integer > 0).

Remarks:

1. The total load applied in a load case will be the sum of external (**LOAD**), thermal (**TEMPERATURE**), gravity (**GRAVITY**), centrifugal (**CENTRIFUGAL**) and deformation (**DEFORM**) loads.
2. Gravity load sets can be selected in static load cases with **either** the Solution Control command **GRAVITY**=SID **or** with the Solution Control command **LOAD**=SID. If Solution Control LOAD points to a GRAV or to a bulk data **LOAD** that references one or more GRAV entries, then Solution Control GRAVITY must not be used.



5

5.26.29 GRMASS

Solution Control Entry: **GRMASS -** Analysis Output Request

Description: Requests printing of the grid mass matrix.

Format:

$$\text{GRMASS} = \begin{Bmatrix} \text{POST} \\ \textbf{NONE} \end{Bmatrix}$$

Example:

```
GRMASS = POST
```

Option	Meaning
POST	The grid mass matrix will be output to the postprocessing file.
NONE	Default. Do not print grid mass matrix.

Remarks:

1. The grid mass matrix is a diagonal matrix. It is calculated by taking the mass of each element, dividing that by the number of grids the element is connected to, and adding that contribution to the corresponding entries in the diagonal matrix.
2. GRMASS is only available for the **POST** = BINARY, POST = FORMAT or POST = PLOT formats.
3. GRMASS is only available when there are structural (non-heat transfer) load cases.

5.26.30 GSTRESS

Solution Control Entry: **GSTRESS** - Analysis Output Request

Description: Requests grid point stress output for CHEXA, CHEX20, CPENTA, CTETRA and CPYRA solid elements and CTRIAX6 axisymmetric elements

Format:

$$\mathrm{GSTRESS} = \left\{ \begin{array}{c} \mathbf{NONE} \\ \mathrm{n} \\ \mathrm{ALL} \\ \mathrm{POST} \\ \mathrm{POST, n} \\ \mathrm{POST, ALL} \\ \mathrm{BOTH} \\ \mathrm{BOTH, n} \\ \mathrm{BOTH, ALL} \end{array} \right\}$$

Alternate Format:

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

Examples:

```
GSTRESS=5
GSTRESS(PRINT,PUNCH) = 17
GSTRESS(PLOT) = ALL
GSTRESS=ALL
```

Option	Meaning
NONE	Default. No grid point stresses will be output.
n	Set identification of previously appearing SET data. Only grid point stresses of grids whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	Stresses for all grid points will be output.
POST	Stresses for all grid points will be output to the post processing file.
POST, n	Stresses for all grid points in set n will be output to the post processing file. n should be integer > 0 or blank.
POST, ALL	Same as POST.
BOTH	Stresses for all grid points will be output to both the output file and the post-processing file.

BOTH, n	Stresses for all grid points in set n will be output to both the output file and the post-processing file. n should be integer > 0 or blank.
BOTH, ALL	Same as BOTH.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2..

Remarks:

1. Stresses are printed in the basic coordinate system.
2. When the POST option is used, no grid stress results are printed to the output file.
3. When the POST option is used, the **POST** command must be used in the executive section of the input data to generate the post processing file.
4. Only grid stresses associated with solid elements **CTETRA**, **CPENTA**, **CHEXA**, **CHEX20** and **CPYRA** and axisymmetric elements, **CTRIAX6**, are calculated.
5. For dynamic analysis, either magnitude and phase or real and imaginary components can be output. See **DYNOUTPUT**.
6. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

5.26.31 HEAT

Solution Control Entry: **HEAT** - Heat Transfer Load Selection

Description: Selects the heat transfer load set to be applied to the heat transfer model

Format:

HEAT = n

Examples:

```
HEAT = 15
```

Option	Meaning
n	Set identification of at least one external load statement and hence must appear on at least one **QVOL**, **QVECT**, **QHBDY**, **QBDY1**, **QBDY2** or **SPCD** entry

Remarks:

1. HEAT set identification numbers must be distinct with respect to **LOAD** set identification numbers.

5

5.26.32 INCLUDE

Solution Control Entry: **INCLUDE**

Description: Select an external file that contains solution control statements.

Format:

```
INCLUDE 'file name'
```

Alternate Format:

```
INCLUDE = file name
```

Examples:

INCLUDE 'SET300.TXT'

INCLUDE = LC40.INP

Option	Meaning
file name	External file name. The user must provide the file name according to the machine installation.

Remarks:

1. The INCLUDE data can be anywhere in the solution control.
2. Multiple INCLUDE data are allowed in the solution control.
3. The external file cannot contain INCLUDE or PROCESS data statements.
4. The external file cannot contain the **BEGIN BULK** delimiter.
5. The file name is limited to 240 characters.
6. If the quoted format is used, and the line does not end with a quote character, additional lines will be read until the closing quote is found. Leading and trailing spaces on continued and continuation lines are discarded.

5.26.33 K2GG

Solution Control Entry: **K2GG** - External Matrix Selection

Description: Select DMIG matrices to add to the stiffness matrix.

Format:

K2GG = *matrix*

Example:

```
K2GG = K0000001
```

Option	Meaning
matrix	Name of a matrix defined by **DMIG** bulk data entries (Character).

Remarks:

1. The K2GG command must appear above the first loadcase definition and affects the stiffness for all structural loadcases.
2. The matrix identified by *matrix* must be real and symmetric (the DMIG header must specify 6 for the form and 1 or 2 for the type).
3. The matrix name may have at most 8 characters.
4. The matrix may be scaled using the analysis parameter **CK2**.

5.26.34 K2PP

Solution Control Entry: **K2PP** - External Matrix Selection for Frequency Response

Description: Select DMIG matrices to add to the dynamic stiffness matrix.

Format:

K2PP = *matrix*

Example:

```
K2PP = KDAMP
```

Option	Meaning
matrix	Name of a matrix defined by **DMIG** bulk data entries (Character).

Remarks:

1. The K2PP command must appear above the first loadcase definition and affects the stiffness for frequency response loadcases.
2. The matrix identified by *matrix* must be symmetric (the DMIG header must specify 6 for the form). The matrix may be real or complex (the DMIG header may specify 1, 2, 3, or 4 for the type).
3. The matrix name may have at most 8 characters.
4. In direct frequency response loadcases, the real and imaginary parts will be added to the system complex matrix for each loading frequency.
5. In modal frequency response loadcases, the real and imaginary parts will be reduced to modal degrees of freedom and added to the modal complex matrix for each loading frequency. The matrix is not used when the natural frequencies and mode shapes are calculated.

5.26.35 K42GG

Solution Control Entry: **K42GG** - External Matrix Selection

Description: Select DMIG matrices to add to the structural damping matrix.

Format:

K42GG = *matrix*

Example:

```
K42GG = K0000001
```

Option	Meaning
matrix	Name of a matrix defined by **DMIG** bulk data entries (Character).

Remarks:

1. The K42GG command must appear above the first loadcase definition and affects the structural damping for all frequency response loadcases.
2. The matrix identified by *matrix* must be real and symmetric (the DMIG header must specify 6 for the form and 1 or 2 for the type).
3. The matrix name may have at most 8 characters.
4. The matrix may be scaled using the analysis parameter **CK42**.

5

5.26.36 K4AA

Solution Control Entry: **K4AA** - Damping Reduction Matrix Output Request

Description: Requests printing of the reduced elemental structural damping matrix.

Format:

$$\textbf{K4AA} = \begin{Bmatrix} \text{DMIG} \\ \textbf{NONE} \end{Bmatrix}$$

Examples:

```
K4AA = DMIG
K4AA = NONE
```

Option	Meaning
DMIG	The reduced elemental structural damping matrix will be output to the DMIG post processing file.
NONE	Default. Do not print reduced damping matrix.

Remarks:

1. The K4AA command can only be used in conjuction with the **REDUCE** executive control command and the **BOUNDARY** solution control command.
2. The DMIG post-processing file is named *pname*xx.DMIG where *pname* is the project name and xx is the design cycle. This file contains data according to the **DMIG** bulk data format.
3. The DMIG matrix name will be Eyyyyyyy where yyyyyyy is the loadcase number.
4. Usually, the K4AA command with the DMIG option should not be used before the first loadcase. It should only be used inside one loadcase. Otherwise, multiple copies of the same reduced damping matrix may be printed in the DMIG file.
5. By default, this command will produce a real matrix, suitable for use with the **K42GG** command. To produce imaginary terms of a complex matrix, suitable for use with the **K2PP** command, use **PARAM**,IPRM4,1.

5.26.37 KAA

Solution Control Entry: **KAA** - Guyan Reduction Matrix Output Request

Description: Requests printing of the reduced stiffness matrix.

Format:

$$\mathbf{KAA} = \begin{Bmatrix} \text{DMIG} \\ \text{POST} \\ \textbf{NONE} \end{Bmatrix}$$

Examples:

```
KAA = DMIG
KAA = POST
```

Option	Meaning
DMIG	The reduced stiffness matrix will be output the the DMIG post processing file.
POST	The reduced stiffness matrix will be output to the KAA post processing file.
NONE	Default. Do not print reduced stiffness matrix.

Remarks:

1. The KAA command can only be used in conjuction with the **REDUCE** executive control command and the **BOUNDARY** solution control command *or* in **ASET** eigenvalue loadcases.
2. The KAA post processing file is named "*pname*xxyy.KAA" or "*pname*xxyyyyyyyy.KAA" where *pname* is the project name, xx corresponds to the design cycle number and yy or yyyyyyyy corresponds to the loadcase number. yy is used for a loadcase with id 99 or lower, otherwise yyyyyyyy is used.
3. The format of the ".KAA" file is described in **Guyan Reduced Stiffness Matrix** (Sec. 7.3.5).
4. The format of the ".KAA" file is identical to the format expected by the **K2UU** command.
5. The DMIG post-processing file is named *pname*xx.DMIG where *pname* is the project name and xx is the design cycle. This file contains data according to the **DMIG** bulk data format.
6. The DMIG matrix name will be Kyyyyyyy where yyyyyyy is the loadcase number.
7. Usually, the KAA command with the DMIG option should not be used before the first loadcase. It should only be used inside one loadcase. Otherwise, multiple copies of the same reduced stiffness matrix may be printed in the DMIG file.

5.26.38 LABEL

Solution Control Entry: **LABEL** - Output Control

Description: Defines a label which will appear on the fourth heading line of each page of printer output containing analysis results.

Format:

LABEL = Any Character data

Alternate Format:

LABEL Any Character data

Examples:

```
LABEL = DEMONSTRATION PROBLEM
```

Remarks:

1. LABEL should only appear after the LOADCASE/LOADCOM to which it applies.
2. Each LABEL will output for a single load case only.
3. If no LABEL entry is supplied, the label line will be blank.
4. The length of the label is limited to 72 characters, including blanks for output of 132 characters per line (default). When the user specifies 80 characters per line (e.g. LINE=64,80), the length of the label is limited to 60 characters. Another limitation for the length of the Label is that the input data statement should not exceed the 80th column. If it exceeds the 80th column, the excess portion of the LABEL will be ignored and will not be printed in the output file.
5. If the alternate format is used, there must be at least one blank space after the LABEL keyword and before the character data.

5

5.26.39 LINE

Solution Control Entry: **LINE** - Output Control

Description: Defines the number of data lines per printed page

Format:

$$\text{LINE} = \begin{Bmatrix} * \\ n \end{Bmatrix}, \begin{Bmatrix} * \\ m \end{Bmatrix}$$

Examples:

```
LINE = 35,80
LINE = *,132
```

Option	Meaning
n	Number of data lines per page (Integer > 0) (default is usually 64)
m	Number of characters per line of output (80 or 132) (default is usually 132)
*	Use default values

Remarks:

1. If no LINE data appears, defaults are used.
2. For 11 inch paper, n=64 is recommended; for 8 1/2 inch paper, n=50 is recommended.
3. Default values are system dependent.

5.26.40 LOAD

Solution Control Entry: **LOAD** - Static Load Selection

Description: Selects the external static load set to be applied to the structural model

Format:

LOAD = n

Example:

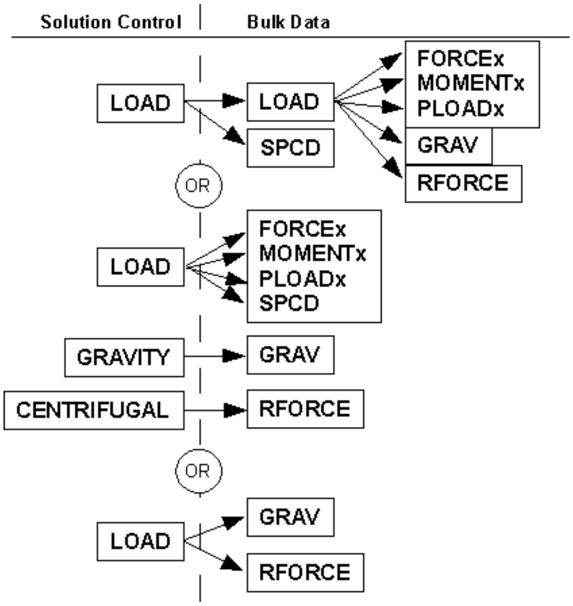
```
LOAD = 15
```

Option	Meaning
n	Set identification of at least one external load statement and hence must appear on at least one **FORCE**, **FORCE1**, **LOAD**, **MOMENT**, **MOMENT1**, **PLOAD1**, **PLOAD2**, **PLOAD4**, **PLOAD5**, **PLOADA**, **PLOADX1**, **GRAV**, **RFORCE** or **SPCD** entry

Remarks:

1. Load data will not be used unless selected in Solution Control.
2. The total load applied in a load case will be the sum of external (**LOAD**), thermal (**TEMPERATURE**), gravity (**GRAVITY**), centrifugal (**CENTRIFUGAL**) and deformation (**DEFORM**) loads.
3. LOAD set identification numbers must be distinct with respect to **HEAT** set identification numbers.

4. The LOAD command can either point to a bulk data **LOAD** or to FORCEx, MOMENTx, PLOADx and SPCD data or to GRAV / RFORCE.



5. To combine gravity / centrifugal loading with other static loading, the **LOAD** bulk data or the **GRAVITY** / **CENTRIFUGAL** solution control commands must be used.

5

5.26.41 LOADCASE

Solution Control Entry: **LOADCASE** - Loadcase Delimiter

Description: Delimits and identifies a load case

Format:

LOADCASE n

Alternate Format:

LOADCASE = n

Example:

```
LOADCASE = 101
```

Option	Meaning
n	Load case identification number (Integer > 0)

Remarks:

1. **SUBCASE** is an alternate form and is entirely equivalent to LOADCASE.
2. There is no limit to the number of load cases.
3. The identification numbers used for heat transfer loadcases must be unique with respect to temperature set IDs defined by **TEMP**/**TEMPD** bulk data.

5.26.42 LOADCOM

Solution Control Entry: **LOADCOM** - Loadcase Delimiter

Description: Delimits and identifies a combination static load case

Format:

LOADCOM n

Alternate Format:

LOADCOM = n

Example:

```
LOADCOM 125
```

Option	Meaning
n	Loadcom identification number (Integer > 2)

Remarks:

1. **SUBCOM** is an alternate form and is entirely equivalent to LOADCOM.
2. A **LOADSEQ** or SUBSEQ entry must appear in this load case.
3. LOADCOM may only be used to combine static load cases.
4. Output requests above the load case level will be utilized as defaults. **OLOAD** and **SPCFORCE** do not produce output in this case.
5. Thermal loads are allowed in only one of the LOADCASES that are combined in the LOADCOM.

5.26.43 LOADSEQ

Solution Control Entry: **LOADSEQ** - Loadcase Sequence Coefficients

Description: Gives the coefficients for forming a linear combination of the previous static load cases

Format:

LOADSEQ = r1 [, r2, r3, ..., rn]

Example:

```
LOADSEQ = 1.0, -1.0, 0.0, 2.0
```

Option	Meaning
r1 to rn	Coefficients of the previously occurring static load cases (Real)

Remarks:

1. **SUBSEQ** is an alternate form and is entirely equivalent to LOADSEQ.
2. A LOADSEQ or SUBSEQ entry must only appear in a **LOADCOM** or SUBCOM load case.
3. A LOADSEQ entry may be more than one line of data. A comma at the end signifies that the data is continued on the next line.
4. The LOADSEQ list applies to the immediately preceding static load cases. For example, the comments describe the following example:

```
DISPL = ALL
LOADCASE 1
LOADCASE 2
LOADCOM 3
$ LOAD CASE 1 - LOAD CASE 2
LOADSEQ = 1.0, -1.0
LOADCASE 11
LOADCASE 12
LOADCOM 13
$ LOAD CASE 11 - LOAD CASE 12
LOADSEQ = 0.0, 0.0, 1.0, -1.0
$ EQUIVALENT TO PRECEDING ENTRY.USE ONLY ONE.
LOADSEQ = 1.0, -1.0
```

5. If more than one static **LOADCASE** that is referenced by a LOADSEQ with a non-zero coefficient contains a thermal load, a fatal error will occur.
6. Coefficients for frequency calculation, heat transfer and dynamic analysis LOADCASES must be zero.

7. There should be no coefficient for preceding LOADCOM’s (i.e. LOADCOM’s are skipped).

5.26.44 M2GG

Solution Control Entry: **M2GG** - External Matrix Selection

Description: Select DMIG matrices to add to the mass matrix.

Format:

M2GG = *matrix*

Example:

```
M2GG = K0000001
```

Option	Meaning
matrix	Name of a matrix defined by **DMIG** bulk data entries (Character).

Remarks:

1. The M2GG command must appear above the first loadcase definition and affects the mass matrix for all structural loadcases.
2. The matrix identified by *matrix* must be real and symmetric (the DMIG header must specify 6 for the form and 1 or 2 for the type).
3. The matrix name may have at most 8 characters.
4. The matrix is not scaled by the analysis parameter **WTMASS**. The matrix should be in mass units. The matrix may be scaled using the analysis parameter **CM2**.

5

5.26.45 MAA

Solution Control Entry: **MAA** - Guyan Reduction Matrix Output Request

Description: Requests printing of the reduced mass matrix.

Format:

$$\text{MAA} = \begin{Bmatrix} \text{DMIG} \\ \text{POST} \\ \textbf{NONE} \end{Bmatrix}$$

Examples:

```
MAA = DMIG
MAA = POST
```

Option	Meaning
DMIG	The reduced stiffness matrix will be output the the DMIG post processing file.
POST	The reduced mass matrix will be output to the MAA postprocessing file.
NONE	Default. Do not print reduced mass matrix.

Remarks:

1. The MAA command can only be used in conjuction with the **REDUCE** executive control command and the **BOUNDARY** solution control command *or* in **ASET** eigenvalue loadcases.
2. The MAA post processing file is named "*pname*xxyy.MAA" or "*pname*xxyyyyyyyy.MAA" where *pname* is the project name, xx corresponds to the design cycle number and yy or yyyyyyyy corresponds to the loadcase number. yy is used for a loadcase with id 99 or lower, otherwise yyyyyyyy is used.
3. The format of the ".MAA" file is described in **Guyan Reduced Mass Matrix** (Sec. 7.3.5).
4. The format of the ".MAA" file is identical to the format expected by the **M2UU** command.
5. The DMIG post-processing file is named *pname*xx.DMIG where *pname* is the project name and xx is the design cycle. This file contains data according to the **DMIG** bulk data format.
6. The DMIG matrix name will be Myyyyyyy where yyyyyyy is the loadcase number.
7. Usually, the MAA command with the DMIG option should not be used before the first loadcase. It should only be used inside one loadcase. Otherwise, multiple copies of the same reduced mass matrix may be printed in the DMIG file.

5.26.46 MAAUSER

Solution Control Entry: **MAAUSER** - Guyan Reduction User Mass Matrix Control

Description: Requests to use the user reduced mass matrix.

Format:

$$\mathbf{MAAUSER} = \begin{Bmatrix} \text{YES} \\ \mathbf{NO} \end{Bmatrix}$$

Example:

```
MAAUSER = YES
```

Option	Meaning
YES	User reduced mass matrix.
NO	Default. Do not use the user reduced mass matrix.

Remarks:

1. This command can only be used on reduced (**ASET**) eigenvalue load cases.
2. This command is used to control the use of user defined mass matrix. The user has to provide the mass matrix calculation using the user subroutine **GNMASS**.

5

5.26.47 MASS

Solution Control Entry: **MASS** - Analysis Output Request

Description: Requests printing of system, property and material mass

Format:

$$\text{MASS} = \begin{Bmatrix} \text{YES} \\ \textbf{NO} \end{Bmatrix}$$

Examples:

```
MASS = YES
```

```
MASS = NO
```

Option	Meaning
YES	The mass summary table will be printed for each design cycle
NO	Default. No mass summary table will be printed

Remarks:

1. If the volume summary table (**VOLUME** = YES) is also requested, the tables will be combined.

5.26.48 MCONTRIB

Solution Control Entry: **MCONTRIB** - Analysis Output Request

Description: Requests a modal contribution table for modal frequency response results.

Format:

$$\text{MCONTRIB=}\left\{\begin{array}{c}\textbf{NONE}\\ \text{n}\\ \text{ALL}\end{array}\right\}$$

Example:

```
MCONTRIB = 6
```

Option	Meaning
NONE	Default. No modal contribution table will be output.
n	Set identification of previously appearing SET data. Only modal contributions of grids whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	Modal contributions for all grid points will be output.

Remarks:

1. For each requested grid, 6 modal contribution tables will be printed (one each for the three translational and three rotational degrees of freedom).
2. Modal contributions are calculated for displacements. The relative modal contributions for velocities and accelerations are identical to those of displacements. See **Modal Contribution** (Sec. 2.16.3).
3. For each degree of freedom, only modes with a relative modal contribution greater than 0.01 are listed.

5.26.49 METHOD

Solution Control Entry: **METHOD** - Eigenvalue Calculation Method Selection

Description: Selects the frequency or buckling calculation parameters to be used

Format:

METHOD = n

Alternate Format:

$$\text{METHOD}\left(\begin{Bmatrix}\text{STRUCTURE}\\ \text{FLUID}\end{Bmatrix}\right) = n$$

Examples:

```
METHOD = 33
METHOD(STRUCTURE) = 5
METHOD(FLUID) = 12
```

Option	Meaning
n	Set identification number of an **EIGR** or **EIGRL** entry. (Integer > 0)

Remarks:

1. Calculated frequencies/load factors will always be printed. Mode shapes can be printed using **SVECTOR** or **DISPLACEMENT** commands.
2. The alternate format can be used if it is desired to use different eigenvalue calculation parameters for the fluid portion from the structure portion of a coupled fluid-structure model. If the model contains fluid elements, then either use one normal format entry to apply the same EIGR/EIGRL to both the fluid and the structure, or use two alternate format entries, METHOD(STRUCTURE) and METHOD(FLUID), to specify the EIGR/EIGRL for the structure and the fluid, respectively.

5

5.26.50 MODES

Solution Control Entry: **MODES** - Modal Dynamic Loadcase Control

Description: Selects a natural frequency LOADCASE to specify modes to be used in modal representation of dynamic response

Format:

MODES = n

Example:

```
MODES = 3
```

Option	Meaning
n	Loadcase identification number of a natural frequency **LOADCASE** (Integer > 0)

Remarks:

1. The modal dynamic response loadcase uses the MPC and SPC sets of the referenced frequency calculation loadcase.

5

5.26.51 MODESELECT

Solution Control Entry: **MODESELECT** - Modal Dynamic Loadcase Control

Description: Selects a subset of the calculated modes to become the basis for modal dynamic frequency response analysis.

Format:

MODESELECT = n

Alternate Format:

MODESELECT (LFREQ=lv HFREQ=hv UNCONSET=m)

Examples:

```
MODESELECT = -3
MODESELECT(HFREQ=1000.0 UNCONSET=10)
```

Option	Meaning
n	Identification number of a **SET** of mode numbers to use or remove. (Integer) See Remark 1.
lv	Lower frequency cutoff. (Real $\geq$ 0.0) See Remarks 2 and 3.
hv	Upper frequency cutoff. (Real $\geq$ 0.0) See Remarks 2 and 3.
m	Identification number of a **SET** of additional mode numbers to include or exclude. (Integer) See Remarks 2 and 3.

5

Remarks:

1. If $n > 0$ then only mode numbers in SET n will be used to form the modal basis. If $n < 0$ then all calculated modes except those mode numbers in SET $|n|$ will be used to form the modal basis. If $n = 0$, then all calculated modes will be used to form the modal basis.
2. If the alternate format is used, then modes whose corresponding frequency is greater than or equal to *lv* and less than or equal to *hv* will be used to form the modal basis. If $m > 0$, then mode numbers in SET m will also be included regardless of their corresponding frequencies. If $m < 0$, then mode numbers in SET $|m|$ will also be excluded regardless of their corresponding frequencies. If $m = 0$, then all modes within the frequency cutoff bounds will be used.
3. In the alternate format, all of the keyword=value pairs inside the parentheses are optional. The defaults are: $lv = 0.0$, $hv = 1.0\times10^{30}$, $m = 0$.
4. Residual vectors will always be included in the modal basis (if requested with **PARAM**,**RESVEC**).
5. If this command is used, PARAMeters **LFREQ** and **HFREQ** will be ignored.

6. This command serves to remove modes from the modal basis, and therefore reduces the accuracy of the modal frequency response solution. If this command is used, the results should be carefully reviewed for adequacy.

5.26.52 MPC

Solution Control Entry: **MPC -** Multipoint Constraint Set Selection

Description: Selects the multipoint constraint set to be applied to the structural or thermal model

Format:

MPC = n

Example:

```
MPC = 17
```

Option	Meaning
n	"n" is the set identification of a multipoint constraint set and hence must appear on at least one **MPC** or **MPCADD** entry in the bulk data. (Integer > 0)

Remarks:

1. MPC data will not be used unless selected in Solution Control.
2. The same MPC set identification cannot be used for both structural and heat transfer load cases.

5.26.53 NLPARM

Solution Control Entry: **NLPARM** - Nonlinear Static Control

Description: Selects a nonlinear parameter set to be used in the static loadcase

Format:

NLPARM = n

Example:

```
NLPARM = 17
```

Option	Meaning
n	Set identification number of an **NLPARM** entry. (Integer > 0)

Remarks:

1. NLPARM bulk data will not be used unless selected in Solution Control.
2. Nonlinearity is only considered for contact analysis and/or for **CGAP** elements in static analysis. This command will have no effect unless there is also a **BCONTACT** command in the loadcase or there are CGAP elements in the model.

5.26.54 NSM

Solution Control Entry: **NSM** - Nonstructural Mass Selection

Description: Selects a Nonstructural Mass set to be used in the model

Format:

NSM = n

Example:

```
NSM = 17
```

Option	Meaning
n	Set identification of a nonstructural mass set and hence must appear on at least one **NSM**,**NSM1**,**NSML**,**NSML1** or **NSMADD** entry in the bulk data (Integer > 0).

Remarks:

1. Nonstructural mass in **NSM**,**NSM1**,**NSML**,**NSML1** or **NSMADD** data will not be used unless selected in Solution Control.
2. At most one NSM entry may appear in the Solution Control.
3. NSM is not independently selectable in individual loadcases. NSM added in one loadcase will be used in all loadcases.

5.26.55 OLOAD

Solution Control Entry: **OLOAD** - Analysis Output Request

Description: Requests form of applied load vector output

Format:

$$\text{OLOAD} = \left\{ \begin{array}{c} \textbf{NONE} \\ \text{n} \\ \text{ALL} \\ \text{POST} \\ \text{POST, n} \\ \text{POST, ALL} \\ \text{BOTH} \\ \text{BOTH, n} \\ \text{BOTH, ALL} \end{array} \right\}$$

Alternate Format:

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

Examples:

```
OLOAD=5
OLOAD(PRINT,PUNCH) = 17
OLOAD(PLOT) = ALL
OLOAD=ALL
```

Option	Meaning
NONE	Default. No applied loads will be output.
n	Set identification of previously appearing SET data. Only non-zero applied loads whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	All non-zero applied loads will be output.
POST	Applied loads will be output to the post processing file (static analysis only).
POST, n	Applied loads for grids in set n will be output to the post processing file. n should be integer > 0 or blank (static analysis only).
POST, ALL	Same as POST.
BOTH	Applied loads will be output to both the output file and the post-processing file.

BOTH, n	Applied loads for grids in set n will be output to both the output file and the post-processing file. n should be integer > 0 or blank.
BOTH, ALL	Same as BOTH.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2..

Remarks:

1. The applied loads will be printed in the general coordinate system, not in the coordinate systems that they were defined by using local coordinate systems.
2. This request cannot be used in a LOADCOM or in a dynamic response loadcase.
3. When the POST option is used, applied loads are printed to the output file only for static analysis LOADCASEs.
4. When the POST option is used, the **POST** command must be used in the executive section of the input data to generate the post processing file.
5. In heat transfer analysis, the applied flux loads are printed. Applied flux loads are not written to the post processing file.
6. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

5.26.56 P2G

Solution Control Entry: **P2G** - External Matrix Selection

Description: Select DMIG matrices to add to the static load vectors.

Format:

P2G = *matrix*

Example:

```
P2G = PASET
```

Option	Meaning
matrix	Name of a matrix defined by **DMIG** bulk data entries (Character).

Remarks:

1. The P2G command must appear above the first loadcase definition and possibly affects the loads of each static loadcases.
2. The matrix identified by *matrix* must be real and rectangular (the DMIG header must specify 9 for the form and 1 or 2 for the type). The number of columns (NCOL) must be equal to the number of static loadcases.
3. Each matrix name may have at most 8 characters.
4. The matrix may be scaled using the analysis parameter **CP2**.

5.26.57 POSTOUTPUT

Solution Control Entry: **POSTOUTPUT** - Analysis Output Control

Description: Requests grid results in the post processing file to be in the basic or general coordinate system

Format:

$$\text{POSTOUTPUT} = \begin{Bmatrix} \textbf{BASIC} \\ \text{GENERAL} \end{Bmatrix}$$

Examples:

```
POSTOUTPUT = BASIC
POSTOUTPUT = GENERAL
```

Option	Meaning
BASIC	Default. Results output in the basic coordinate system.
GENERAL	Results output in the general coordinate system.

Remarks:

1. Only displacements, velocities, accelerations and mode shapes are affected by this command. Applied loads and reaction forces are always written in the general coordinate system.
2. This command only affects results written to the post-processing file. Results printed to the output file are not affected by this command. The output file always contains grid results in the general coordinate system.

5

5.26.58 PRESSURE

PRESSURE is a synonym for **DISPLACEMENT**

5.26.59 PROCESS

Solution Control Entry: **PROCESS**

Description: Run an external program that outputs solution control statements.

Format:

```
PROCESS 'command line'
```

Examples:

```
PROCESS 'loadcase_generator -start 1001'
```

Option	Meaning
command line	External program and arguments. The user must provide the command line according to the machine installation.

Remarks:

1. The PROCESS data can be anywhere in the solution control.
2. Multiple PROCESS data are allowed in the solution control.
3. The external program cannot output INCLUDE or PROCESS data statements.
4. The external program cannot output the **BEGIN BULK** delimiter.
5. The command line is limited to 240 characters.
6. If the line does not end with a quote character, additional lines will be read until the closing quote is found. Leading and trailing spaces on continued and continuation lines are discarded.

5.26.60 QSET

Solution Control Entry: **QSET** - Natural Frequency Loadcase Control

Description: Select a set of grids and components to be used as generalized degrees of freedoms for Craig-Bampton modes in a Guyan reduction loadcase.

Format:

QSET = n

Example:

```
QSET = 10
```

Option	Meaning
n	Set identification of QSET and hence must appear on a **QSET2** or **QSET3** entry in the bulk data (Integer>0).

Remarks:

1. QSET2 and QSET3 data will not be used unless selected in the Solution Control.
2. This command must be used together with the **CBMETHOD**, **METHOD** and **ASET** commands.
3. The Craig-Bampton modes are used to enhance the Guyan reduced mass and stiffness matrices.
4. The total number of Craig-Bampton modes enhancing the reduced matrices will be the minimum of: the number of degrees of freedom specified by the selected QSET2 and QSET3 entries; and the number of modes generated by the corresponding Craig-Bampton calculation. Any extra generalized degrees of freedom or modes will be ignored.

5.26.61 RANDOM

Solution Control Entry: **RANDOM** - Random Response Control

Description: Selects the set of power spectral density loads on the **RANDPS** entries and optionally selects time lags **RANDT1** to be used in random analysis

Format:

RANDOM = n

Examples:

```
RANDOM = 100
```

Option	Meaning
n	Set identification of **RANDPS** and optionally **RANDT1** data entries (Integer > 0).

Remarks:

1. The RANDPS or RANDT1 Bulk Data will not be selected unless activated in Solution Control.
2. At least one RANDPS bulk data entry is required for random analysis.
3. This command can only appear in a frequency response loadcase.

5

5.26.62 SDAMPING

Solution Control Entry: **SDAMPING** - Modal Dynamic Loadcase Control

Description: Selects a table which defines damping as a function of frequency in the modal formulation of dynamic analysis

Format:

SDAMPING = n

Example:

```
SDAMPING = 17
```

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

5

5.26.63 SET

Solution Control Entry: **SET** - Set Definition

Description: Lists identification numbers (grid, element or mode) for output requests

Format:

$$\mathrm{SET}\ \ n\ =\ \begin{Bmatrix} \mathrm{i1, i2, i3, \ldots, in} \\ \mathrm{ALL} \end{Bmatrix}$$

Examples:

```
SET 77 = 5
SET 88 = 5, 6, 7, 8, 9, 15, 16, 77
SET 99 = 1 THRU 100000
SET 101 = 1,2,10,THRU,20,33
SET 105 = 101 THRU 105,110,125 THRU,155
```

Option	Meaning
n	Unique set identification number.
i1, i2, etc.	Element, grid point or mode identification number at which output is requested. (Integer > 0). If no such identification number exists, the request is ignored.
i3 THRU i4	Output at set identification numbers i3 through i4 (i4 > i3).
ALL	Output at all identification numbers.

Remarks:

1. A SET entry may be more than one line of data. A comma (,) at the end of a line signifies a continuation line will follow.

5.26.64 SPC

Solution Control Entry: **SPC** - Single-Point Constraint Set Selection

Description: Selects the single-point constraint set to be applied to the structural or thermal model

Format:

SPC = n

Example:

```
SPC = 10
```

Option	Meaning
n	Set identification of a single-point constraint set and hence must appear on a **SPC1**, **SPC** or **SPCADD** entry in the bulk data (Integer > 0)

Remarks:

1. SPC1 or SPC data will not be used unless selected in Solution Control.
2. The same SPC set identification cannot be used for both structural and heat transfer load cases.
3. The SPC command can either point to a bulk data SPCADD or to SPC and SPC1 data.

5.26.65 SPCFORCE

Solution Control Entry: **SPCFORCE** - Analysis Output Request

Description: Requests form of reaction force of single-point constraint vector output

Format:

$$\text{SPCFORCE} = \begin{Bmatrix} \textbf{NONE} \\ \text{n} \\ \text{ALL} \\ \text{POST} \\ \text{POST, n} \\ \text{POST, ALL} \\ \text{BOTH} \\ \text{BOTH, n} \\ \text{BOTH, ALL} \end{Bmatrix}$$

Alternate Format:

$$\text{SPCFORCE}\left(\begin{Bmatrix} \text{PRINT} \\ \text{PUNCH} \\ \text{PLOT} \\ \text{PRINT, PLOT} \\ \text{PRINT, PUNCH} \end{Bmatrix}\right) = \begin{Bmatrix} \textbf{NONE} \\ \text{n} \\ \text{ALL} \end{Bmatrix}$$

Examples:

```
SPCFORCE=5
SPCFORCE(PRINT,PUNCH) = 17
SPCFORCE(PLOT) = ALL
SPCFORCE= ALL
SPCF=NONE
```

Option	Meaning
NONE	Default. No single point forces of constraints will be output.
n	Set identification of previously appearing SET data. Only single point forces of constraints whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	All non-zero single point forces of constraints for will be output.
POST	Single point forces of the constraints will be output to the post processing file.
POST, n	Single point forces of the constraints in set n will be output to the post processing file. n should be integer > 0 or blank.
POST, ALL	Same as POST.

BOTH	Single point forces of the constraints will be output to both the output file and the post-processing file.
BOTH, n	Single point forces of the constraints in set n will be output to both the output file and the post-processing file. n should be integer > 0 or blank.
BOTH, ALL	Same as BOTH.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2..

Remarks:

1. The constraint forces will not be printed for a **LOADCOM** (SPCFORCES are not available for LOADCOM or for dynamic response LOADCASES).
2. The constraint forces are printed in the general coordinate system.
3. When the POST option is used, no constraint forces are printed to the output file.
4. When the POST option is used, the **POST** command must be used in the executive section of the input data to generate the post processing file.
5. In heat transfer analysis, the reaction fluxes are printed. Reaction fluxes are not written to the post processing file.
6. Reaction forces cannot be printed for dynamic analysis load cases.
7. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

5.26.66 STATSUB

Solution Control Entry: **STATSUB** - Buckling Loadcase Control

Description: Selects a static loadcase for buckling analysis.

Format:

STATSUB = n

Examples:

```
STATSUB = 7
STAT = 5
```

Option	Meaning
n	Static **LOADCASE** number (Integer > 0).

Remarks:

1. The boundary conditions in the referenced static loadcase are used in the buckling loadcase.
2. STATSUB cannot reference load combinations (**LOADCOM**). The bulk data statement **LOAD** may be used to combine loads in a regular loadcase.

5

5.26.67 STRAIN

Solution Control Entry: **STRAIN** - Analysis Output Request

Description: Requests form of element strain output

Format:

$$\text{STRAIN} = \begin{Bmatrix} \textbf{NONE} \\ \text{n} \\ \text{ALL} \\ \text{POST} \\ \text{POST, n} \\ \text{POST, ALL} \\ \text{BOTH} \\ \text{BOTH, n} \\ \text{BOTH, ALL} \end{Bmatrix}$$

Alternate Format:

$$\text{STRAIN}\left(\left[\begin{Bmatrix} \text{SORT1} \\ \text{SORT2} \end{Bmatrix},\right]\begin{Bmatrix} \text{PRINT} \\ \text{PUNCH} \\ \text{PLOT} \\ \text{PRINT, PLOT} \\ \text{PRINT, PUNCH} \end{Bmatrix}\left[,\begin{Bmatrix} \text{RPRINT} \\ \text{RPUNCH} \\ \text{RPRINT, RPUNCH} \end{Bmatrix},\begin{Bmatrix} \text{PSDF} \\ \text{ATOC} \\ \text{CRMS} \\ \text{PSDF, ATOC, CRMS} \\ \text{RALL} \end{Bmatrix}\right]\right) = \begin{Bmatrix} \textbf{NONE} \\ \text{n} \\ \text{ALL} \end{Bmatrix}$$

Examples:

```
STRAIN= ALL
STRAIN(PRINT,PUNCH) = 17
STRAIN(PLOT) = ALL
STRAIN=25
```

Option	Meaning
NONE	Default. No element strains will be output.
n	Set identification of previously appearing SET data. Only element strains of elements whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	Strains for all elements will be output.
POST	Strains for all elements will be output to the post processing file.
POST, n	Strains for all elements in set n will be output to the post processing file. n should be integer > 0 or blank.
POST, ALL	Same as POST.
BOTH	Strains for all elements will be output to both the output file and the post-processing file.

BOTH, n	Strains for all elements in set n will be output to both the output file and the post-processing file. n should be integer > 0 or blank.
BOTH, ALL	Same as BOTH.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2..
SORT1	Requests random results be sorted first by element ID and second by frequency.
SORT2	Requests random results be sorted first by frequency and second by element ID.
RPRINT	Requests random results printed to the output file.
RPUNCH	Requests random results printed to the punch post-processing file.
PSDF	Requests output for power spectral density function from random analysis
ATOC	Requests output for autocorrelation functions from random analysis
CRMS	Requests output for cumulative root mean square, root mean square (RMS) and number of zero crossings (N0) from random analysis.
RALL	Request out for PSDF, ATOC and CRMS

Remarks:

1. This command is ignored for scalar or line elements.
2. For plate/shell elements the strains are printed in the element coordinate system.
3. For solid elements the strains are printed in the material coordinate system.
4. When the POST option is used, no strain results are printed to the output file.
5. When the POST option is used, the **POST** command must be used in the executive section of the input data in order to generate the post processing file.
6. For dynamic analysis, either magnitude and phase or real and imaginary components can be output. See **DYNOUTPUT**.
7. Dynamic strains are not recovered for composite elements.
8. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.
9. The RPRINT, RPUNCH, PSDF, ATOC, CRMS and RALL options can only be used in frequency response loadcases that contains the **RANDOM** solution control command.

5.26.68 STRESS

Solution Control Entry: **STRESS** - Analysis Output Request

Description: Requests form of element stress output

Format:

$$\mathrm{STRESS} = \left\{ \begin{array}{c} \mathbf{NONE} \\ \mathrm{n} \\ \mathrm{ALL} \\ \mathrm{POST} \\ \mathrm{POST,\ n} \\ \mathrm{POST,\ ALL} \\ \mathrm{BOTH} \\ \mathrm{BOTH,\ n} \\ \mathrm{BOTH,\ ALL} \end{array} \right\}$$

Alternate Format:

$$\mathrm{STRESS}\left(\left[\left\{ \begin{array}{c} \mathrm{SORT1} \\ \mathrm{SORT2} \end{array} \right\},\right]\left\{ \begin{array}{c} \mathrm{PRINT} \\ \mathrm{PUNCH} \\ \mathrm{PLOT} \\ \mathrm{PRINT,\ PLOT} \\ \mathrm{PRINT,\ PUNCH} \end{array} \right\}\left[,\left\{ \begin{array}{c} \mathrm{RPRINT} \\ \mathrm{RPUNCH} \\ \mathrm{RPRINT,\ RPUNCH} \end{array} \right\},\left\{ \begin{array}{c} \mathrm{PSDF} \\ \mathrm{ATOC} \\ \mathrm{CRMS} \\ \mathrm{PSDF,\ ATOC,\ CRMS} \\ \mathrm{RALL} \end{array} \right\}\right]\right) = \left\{ \begin{array}{c} \mathbf{NONE} \\ \mathrm{n} \\ \mathrm{ALL} \end{array} \right\}$$

Examples:

```
STRESS= ALL
STRESS(PRINT,PUNCH) = 17
STRESS(PLOT) = ALL
STRESS=25
```

Option	Meaning
NONE	Default. No element stresses will be output.
n	Set identification of previously appearing SET data. Only element stresses of elements whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	Stresses for all elements will be output.
POST	Stresses for all elements will be output to the post processing file.
POST, n	Stresses for all elements in set n will be output to the post processing file. n should be integer > 0 or blank.
POST, ALL	Same as POST.
BOTH	Stresses for all elements will be output to both the output file and the post-processing file.

BOTH, n	Stresses for all elements in set n will be output to both the output file and the post-processing file. n should be integer > 0 or blank.
BOTH, ALL	Same as BOTH.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2..
SORT1	Requests random results be sorted first by element ID and second by frequency.
SORT2	Requests random results be sorted first by frequency and second by element ID.
RPRINT	Requests random results printed to the output file.
RPUNCH	Requests random results printed to the punch post-processing file.
PSDF	Requests output for power spectral density function from random analysis
ATOC	Requests output for autocorrelation functions from random analysis
CRMS	Requests output for cumulative root mean square, root mean square (RMS) and number of zero crossings (N0) from random analysis.
RALL	Request out for PSDF, ATOC and CRMS

Remarks:

1. Stresses are printed in the element coordinate system, except for solid elements. For solid elements, stresses are printed in the material coordinate system.
2. When the POST option is used, no stress results are printed to the output file.
3. When the POST option is used, the **POST** command must be used in the executive section of the input data to generate the post processing file.
4. For dynamic analysis, either magnitude and phase or real and imaginary components can be output. See **DYNOUTPUT**.
5. Dynamic stresses are not recovered for composite elements.
6. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.
7. The RPRINT, RPUNCH, PSDF, ATOC, CRMS and RALL options can only be used in frequency response loadcases that contains the **RANDOM** solution control command.

5.26.69 SUBCASE

SUBCASE is a synonym for **LOADCASE**

5.26.70 SUBCOM

SUBCOM is a synonym for **LOADCOM**

5

5.26.71 SUBSEQ

SUBSEQ is a synonym for **LOADSEQ**

5.26.72 SUBTITLE

Solution Control Entry: **SUBTITLE** - Output Control

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

Format:

SUBTITLE = Any Character data

Alternate Format:

SUBTITLE Any Character data

Example:

```
SUBTITLE = PROBLEM NO. 5-1A
```

Remarks:

1. If no SUBTITLE entry is supplied, the subtitle line will be blank.
2. SUBTITLE appearing before the first **LOADCASE** will be used as a default for all loadcases, as well as for output pages not associated to any specific loadcase. If a SUBTITLE appears after a LOADCASE entry, it will set the subtitle for that loadcase only.
3. The length of the subtitle is limited to 72 characters, including blanks. Another limitation for the length of the Subtitle is that the input data statement should not exceed the 80th column. If it exceeds the 80th column, the excess portion of the SUBTITLE will be ignored and will not be printed in the output file.
4. If the alternate format is used, there must be at least one blank space after the SUBTITLE keyword and before the character data.

5.26.73 SUMMARY

Solution Control Entry: **SUMMARY** - Output Request

Description: Requests printing of analysis model size, design model size and load case summary tables.

Format:

$$\text{SUMMARY} = \left\{ \begin{matrix} \textbf{YES} \\ \text{NO} \end{matrix} \right\}$$

Examples:

```
SUMMARY = YES
```

```
SUMMARY = NO
```

Option	Meaning
YES	Default. The summary tables will be printed after the input data echo.
NO	No summary tables will be printed

5.26.74 SUPORT

Solution Control Entry: **SUPORT** - Inertia Relief Loadcase Control

Description: Select the reference degrees of freedoms used by an inertia relief (free body) static loadcase.

Format:

$$\text{SUPORT} = \begin{Bmatrix} \text{NONE} \\ \text{n} \\ \text{AUTO} \end{Bmatrix}$$

Example:

```
SUPORT = 10
```

Option	Meaning
NONE	Inertia relief analysis will not be used.
n	Set identification of SUPORT and hence must appear on a **SUPORT1** entry in the bulk data (Integer>0).
AUTO	Activate the automatic method of inertia relief analysis.

Remarks:

1. SUPORT1 data will not be used unless selected in the Solution Control.
2. The SUPORT command can only be used in a static loadcases.
3. Different static loadcases can select different SUPORT1 data.
4. The default is NONE unless **PARAM**, **INREL** is set to -2, in which case the default is AUTO.

5.26.75 SVECTOR

Solution Control Entry: **SVECTOR** - Analysis Output Request

Description: Requests mode shape output

Format:

$$\text{SVECTOR} = \left\{ \begin{matrix} \textbf{NONE} \\ \text{POST} \\ \text{BOTH} \\ \left[\left\{ \begin{matrix} \text{POST} \\ \text{BOTH} \end{matrix} \right\} , \right] \left\{ \begin{matrix} \text{n} \\ \text{ALL} \end{matrix} \right\} \left[, \left\{ \begin{matrix} \text{m} \\ \text{ALL} \end{matrix} \right\} \right] \end{matrix} \right\}$$

Alternate Format:

$$\text{SVECTOR}\left(\left\{ \begin{matrix} \text{PRINT} \\ \text{PUNCH} \\ \text{PLOT} \\ \text{PRINT, PLOT} \\ \text{PRINT, PUNCH} \end{matrix} \right\} \right) = \left\{ \begin{matrix} \textbf{NONE} \\ \left\{ \begin{matrix} \text{n} \\ \text{ALL} \end{matrix} \right\} \left[, \left\{ \begin{matrix} \text{m} \\ \text{ALL} \end{matrix} \right\} \right] \end{matrix} \right\}$$

Examples:

```
SVECTOR= ALL
SVECTOR(PRINT,PUNCH) = 17
SVECTOR(PLOT) = ALL
SVECTOR=NONE
```


Option	Meaning
NONE	Default. No mode shapes will be output.
n	Set identification of previously appearing SET data. Only grids whose identification numbers appear in the SET data will be output (Integer > 0).
m	Set identification of previously appearing SET data. Only mode numbers that appear in the SET data will be output (Integer > 0 or blank, default = ALL).
ALL	Mode shapes for all modes will be output.
POST	Mode shapes will be output to the post processing file.
BOTH	Mode shapes will be output to both the output file and the post-processing file.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.

PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2..

Remarks:

1. Output will be presented as a tabular listing of points for each mode shape.
2. Mode shapes written to the output file are always in the general coordinate system. The coordinate system used for mode shapes written to the post-processing file is controlled by the **POSTOUTPUT** solution control command (default = basic).
3. When the POST option is used, no mode shape results are printed to the output file.
4. When the POST option is used, the **POST** command must be used in the executive section of the input data in order to generate the post processing file.
5. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

5.26.76 TEMPERATURE

Solution Control Entry: **TEMPERATURE** - Static Load Selection

Description: Selects the temperature set to be used in thermal loading.

Format:

TEMPERATURE = n

Examples:

```
TEMPERATURE = 15
TEMPERATURE = 7
TEMP = 5
```

Option	Meaning
n	Set identification number of **TEMP** and/or **TEMPD** data or heat transfer LOADCASE number (see Remark 2). (Integer > 0).

Remarks:

1. The total load applied in a load case will be the sum of external (**LOAD**), thermal (**TEMPERATURE**), gravity (**GRAVITY**), centrifugal (**CENTRIFUGAL**) and deformation (**DEFORM**) loads.
2. The solution of a heat transfer loadcase can be used as a static thermal load by specifying the heat transfer LOADCASE number.

5.26.77 THERMAL

Solution Control Entry: **THERMAL** - Analysis Output Request

Description: Requests form of temperature vector output

Format:

$$\text{THERMAL} = \left\{\begin{array}{c}\textbf{NONE}\\ \text{n}\\ \text{ALL}\\ \text{POST}\\ \text{POST, n}\\ \text{POST, ALL}\\ \text{BOTH}\\ \text{BOTH, n}\\ \text{BOTH, ALL}\end{array}\right\}$$

Alternate Format:

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

Examples:

```
THERMAL= 5
THERMAL(PRINT,PUNCH) = 17
THERMAL(PLOT) = ALL
THERMAL=NONE
```

Option	Meaning
NONE	Default. No grid temperatures will be output.
n	Set identification of previously appearing SET data. Only temperatures of points whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	Temperatures for all points will be output.
POST	Temperatures for all points will be output to the post processing file.
POST, n	Temperatures for all points in set n will be output to the post processing file. n should be integer > 0 or blank.
POST, ALL	Same as POST.
BOTH	Temperatures for all points will be output to both the output file and the post-processing file.

BOTH, n	Temperatures for all points in set n will be output to both the output file and the post-processing file. n should be integer > 0 or blank.
BOTH, ALL	Same as BOTH.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2..

Remarks:

1. When the POST option is used, no temperature results are printed to the output file.
2. When the POST option is used, the **POST** command must be used in the executive section of the input data in order to generate the post processing file.
3. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

5.26.78 TIMES

Solution Control Entry: **TIMES** - Output Control

Description: Requests the printing of CPU and elapsed times.

Format:

$$\text{TIMES} = \begin{Bmatrix} \text{PRINT} \\ \text{SCREEN} \\ \text{BOTH} \\ \textbf{NONE} \end{Bmatrix}$$

Example:

```
TIMES = BOTH
```

Option	Meaning
PRINT	CPU and elapsed times for each module will be printed to the output file.
SCREEN	CPU and elapsed times for each module will be displayed on the console as *GENESIS* runs.
BOTH	Times will be printed in the output file and on the screen.
NONE	Default. No times will be printed or displayed.

5

5.26.79 TITLE

Solution Control Entry: **TITLE** - Output Control

Description: Defines a title which will appear on the second heading line of each page of *GENESIS* printer output.

Format:

TITLE = Any Character data

Alternate Format:

TITLE Any Character data

Example:

```
TITLE = BODY PANEL DESIGN
```

Remarks:

1. If no TITLE data is supplied, the second line will be blank.
2. TITLE appearing before the first **LOADCASE** will be used as a default for all loadcases, as well as for output pages not associated to any specific loadcase. If a TITLE appears after a LOADCASE entry, it will set the title for that loadcase only.
3. The length of the title is limited to 72 characters, including blanks. Another limitation for the length of the Title is that the input data statement should not exceed the 80th column. If it exceeds the 80th column, the excess portion of the TITLE will be ignored and will not be printed in the output file.
4. If the alternate format is used, there must be at least one blank space after the TITLE keyword and before the character data.

5.26.80 UFACCE

Solution Control Entry: **UFACCE**- Analysis Output Request

Description: Requests form of user function of dynamic accelerations output

Format:

$$\text{UFACCE} = \begin{Bmatrix} \textbf{NONE} \\ \text{n} \\ \text{ALL} \\ \text{POST} \\ \text{POST, n} \\ \text{POST, ALL} \\ \text{BOTH} \\ \text{BOTH, n} \\ \text{BOTH, ALL} \end{Bmatrix}$$

Alternate Format:

$$\text{UFACCE}\left(\left[\begin{Bmatrix} \text{SORT1} \\ \text{SORT2} \end{Bmatrix},\right]\begin{Bmatrix} \text{PRINT} \\ \text{PUNCH} \\ \text{PRINT, PUNCH} \end{Bmatrix}\right) = \begin{Bmatrix} \textbf{NONE} \\ \text{n} \\ \text{ALL} \end{Bmatrix}$$

Examples:

```
UFACCE = ALL
UFACCE(PRINT,PUNCH) = 17
UFACCE=5
```

Option	Meaning
NONE	Default. No user function of accelerations will be output.
n	Set identification of previously appearing SET data. Only results for field points whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	Results for all field points will be output.
POST	Results for all field points will be output to the post processing file.
POST, n	Results for field points in set n will be output to the post processing file. n should be integer > 0.
POST, ALL	Same as POST.
BOTH	Results for all field points will be output to both the output file and the post-processing file.
BOTH, n	Results for field points in set n will be output to both the output file and the post-processing file. n should be integer > 0.
BOTH, ALL	Same as BOTH.

5

PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
SORT1	Requests results be sorted first by field point ID and second by frequency.
SORT2	Requests results be sorted first by frequency and second by field point ID.

Remarks:

1. UFACCE output is only available for dynamic analysis.
2. The **UFDATA** executive control command must be used to specify the file containing the user function data.
3. UFACCE = NONE allows overriding an overall output request.
4. When the POST command is used, no results are printed in the output file.
5. When the POST command is used, the **POST** command must also appear in the executive section to create the post processing output file. In the current version, only the PUNCH format is supported.
6. Either magnitude and phase or real and imaginary components can be output. See **DYNOUTPUT**.
7. Only PUNCH file format is supported as post-processing file and SORT2 option is not supported for the PUNCH format.
8. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

5.26.81 UFDISP

Solution Control Entry: **UFDISP**- Analysis Output Request

Description: Requests form of user function of dynamic displacements output

Format:

$$\text{UFDISP} = \begin{Bmatrix} \textbf{NONE} \\ \text{n} \\ \text{ALL} \\ \text{POST} \\ \text{POST, n} \\ \text{POST, ALL} \\ \text{BOTH} \\ \text{BOTH, n} \\ \text{BOTH, ALL} \end{Bmatrix}$$

Alternate Format:

$$\text{UFDISP}\left(\left[\begin{Bmatrix} \text{SORT1} \\ \text{SORT2} \end{Bmatrix},\right]\begin{Bmatrix} \text{PRINT} \\ \text{PUNCH} \\ \text{PRINT, PUNCH} \end{Bmatrix}\right) = \begin{Bmatrix} \textbf{NONE} \\ \text{n} \\ \text{ALL} \end{Bmatrix}$$

Examples:

```
UFDISP = ALL
UFDISP(PRINT,PUNCH) = 17
UFDISP=5
```


Option	Meaning
NONE	Default. No user function of displacements will be output.
n	Set identification of previously appearing SET data. Only results for field points whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	Results for all field points will be output.
POST	Results for all field points will be output to the post processing file.
POST, n	Results for field points in set n will be output to the post processing file. n should be integer > 0.
POST, ALL	Same as POST.
BOTH	Results for all field points will be output to both the output file and the post-processing file.
BOTH, n	Results for field points in set n will be output to both the output file and the post-processing file. n should be integer > 0.
BOTH, ALL	Same as BOTH.

PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
SORT1	Requests results be sorted first by field point ID and second by frequency.
SORT2	Requests results be sorted first by frequency and second by field point ID.
n	Set identification of previously appearing SET data. Only results for field points whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	Results for all field points will be output.
POST	Results for all field points will be output to the post processing file.

Remarks:

1. UFDISP output is only available for dynamic analysis.
2. The **UFDATA** executive control command must be used to specify the file containing the user function data.
3. UFDISP = NONE allows overriding an overall output request.
4. When the POST command is used, no results are printed in the output file.
5. When the POST command is used, the **POST** command must also appear in the executive section to create the post processing output file. In the current version, only the PUNCH format is supported.
6. Either magnitude and phase or real and imaginary components can be output. See **DYNOUTPUT**.
7. Only PUNCH file format is supported as post-processing file and SORT2 option is not supported for the PUNCH format.
8. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

5.26.82 UFVELO

Solution Control Entry: **UFVELO**- Analysis Output Request

Description: Requests form of user function of dynamic velocities output

Format:

$$\mathrm{UFVELO} = \left\{ \begin{array}{c} \textbf{NONE} \\ \mathrm{n} \\ \mathrm{ALL} \\ \mathrm{POST} \\ \mathrm{POST, n} \\ \mathrm{POST, ALL} \\ \mathrm{BOTH} \\ \mathrm{BOTH, n} \\ \mathrm{BOTH, ALL} \end{array} \right\}$$

Alternate Format:

$$\mathrm{UFVELO}\left(\left[\left\{ \begin{array}{c} \mathrm{SORT1} \\ \mathrm{SORT2} \end{array} \right\},\right]\left\{ \begin{array}{c} \mathrm{PRINT} \\ \mathrm{PUNCH} \\ \mathrm{PRINT, PUNCH} \end{array} \right\}\right) = \left\{ \begin{array}{c} \textbf{NONE} \\ \mathrm{n} \\ \mathrm{ALL} \end{array} \right\}$$

Examples:

```
UFVELO= ALL
UFVELO(PRINT,PUNCH) = 17
UFVELO=5
```

Option	Meaning
NONE	Default. No user function of velocities will be output.
n	Set identification of previously appearing SET data. Only results for field points whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	Results for all field points will be output.
POST	Results for all field points will be output to the post processing file.
POST, n	Results for field points in set n will be output to the post processing file. n should be integer > 0.
POST, ALL	Same as POST.
BOTH	Results for all field points will be output to both the output file and the post-processing file.
BOTH, n	Results for field points in set n will be output to both the output file and the post-processing file. n should be integer > 0.
BOTH, ALL	Same as BOTH.

PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
SORT1	Requests results be sorted first by field point ID and second by frequency.
SORT2	Requests results be sorted first by frequency and second by field point ID.
n	Set identification of previously appearing SET data. Only results for field points whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	Results for all field points will be output.
POST	Results for all field points will be output to the post processing file.

Remarks:

1. UFVELO output is only available for dynamic analysis.
2. The **UFDATA** executive control command must be used to specify the file containing the user function data.
3. UFVELO = NONE allows overriding an overall output request.
4. When the POST command is used, no results are printed in the output file.
5. When the POST command is used, the **POST** command must also appear in the executive section to create the post processing output file. In the current version, only the PUNCH format is supported.
6. Either magnitude and phase or real and imaginary components can be output. See **DYNOUTPUT**.
7. Only PUNCH file format is supported as post-processing file and SORT2 option is not supported for the PUNCH format.
8. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.

5.26.83 VECTOR

VECTOR is a synonym for **DISPLACEMENT**

5.26.84 VELOCITY

Solution Control Entry: **VELOCITY** - Analysis Output Request

Description: Requests velocity vector output

Format:

$$\mathrm{VELOCITY} = \left\{ \begin{array}{c} \textbf{NONE} \\ \mathrm{n} \\ \mathrm{ALL} \\ \mathrm{POST} \\ \mathrm{POST,\ n} \\ \mathrm{POST,\ ALL} \\ \mathrm{BOTH} \\ \mathrm{BOTH,\ n} \\ \mathrm{BOTH,\ ALL} \end{array} \right\}$$

Alternate Format:

$$\mathrm{VELO}\left(\left[\left\{ \begin{array}{c} \mathrm{SORT1} \\ \mathrm{SORT2} \end{array} \right\},\right] \left\{ \begin{array}{c} \mathrm{PRINT} \\ \mathrm{PUNCH} \\ \mathrm{PLOT} \\ \mathrm{PRINT,\ PLOT} \\ \mathrm{PRINT,\ PUNCH} \end{array} \right\} \left[, \left\{ \begin{array}{c} \mathrm{RPRINT} \\ \mathrm{RPUNCH} \\ \mathrm{RPRINT,\ RPUNCH} \end{array} \right\}, \left\{ \begin{array}{c} \mathrm{PSDF} \\ \mathrm{ATOC} \\ \mathrm{CRMS} \\ \mathrm{PSDF,\ ATOC,\ CRMS} \\ \mathrm{RALL} \end{array} \right\} \right]\right) = \left\{ \begin{array}{c} \textbf{NONE} \\ \mathrm{n} \\ \mathrm{ALL} \end{array} \right\}$$

Examples:

```
VELOCITY= ALL
VELOCITY(PRINT,PUNCH) = 17
VELOCITY(PLOT) = ALL
VELOCITY=5
```

Option	Meaning
NONE	Default. No velocities will be output.
n	Set identification of previously appearing SET data. Only velocities of points whose identification numbers appear in the SET data will be output (Integer > 0).
ALL	Velocities of all points will be output.
POST	Velocities of all points will be output to the post processing file.
POST, n	Velocities of all points in set n will be output to the post processing file. n should be integer > 0 or blank.
POST, ALL	Same as POST.
BOTH	Velocities of all points will be output to both the output file and the post-processing file.

BOTH, n	Velocities of all points in set n will be output to both the output file and the post-processing file. n should be integer > 0 or blank.
BOTH, ALL	Same as BOTH.
PRINT	Requests results printed to the output file.
PUNCH	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be PUNCH.
PLOT	Requests results printed to the post-processing file. If the format of the post-processing file has not been defined by a POST executive control command or a previous output request, then define the format to be OUTPUT2..
SORT1	Requests random results be sorted first by grid ID and second by frequency.
SORT2	Requests random results be sorted first by frequency and second by grid ID.
RPRINT	Requests random results printed to the output file.
RPUNCH	Requests random results printed to the punch post-processing file.
PSDF	Requests output for power spectral density function from random analysis
ATOC	Requests output for autocorrelation functions from random analysis
CRMS	Requests output for cumulative root mean square, root mean square (RMS) and number of zero crossings (N0) from random analysis.
RALL	Request out for PSDF, ATOC and CRMS

Remarks:

1. Velocity output is only available for dynamic analysis.
2. VELOCITY = NONE allows overriding an overall output request.
3. Velocities written to the output file are always in the general coordinate system. The coordinate system used for velocities written to the post-processing file is controlled by the **POSTOUTPUT** solution control command (default = basic).
4. When the POST command is used, no results are printed in the output file.
5. When the POST command is used, the **POST** command must also appear in the executive section to create the post processing output file.
6. Either magnitude and phase or real and imaginary components can be output. See **DYNOUTPUT**.
7. All output requests to the post-processing file will use the same single format. To avoid confusion, it is recommended not to mix the PUNCH and PLOT options on different output requests.
8. The RPRINT, RPUNCH, PSDF, ATOC, CRMS and RALL options can only be used in frequency response loadcases that contains the **RANDOM** solution control command.

5.26.85 VOLUME

Solution Control Entry: **VOLUME** - Analysis Output Request

Description: Requests printing of system, property and material volume

Format:

$$\text{VOLUME} = \begin{Bmatrix} \text{YES} \\ \textbf{NO} \end{Bmatrix}$$

Examples:

```
VOLUME = YES
```

```
VOLUME = NO
```

Option	Meaning
YES	The volume summary table will be printed for each design cycle.
NO	Default. No volume summary table will be output.

Remarks:

1. If the mass summary table (**MASS** = YES) is also requested, the tables will be combined.

5

CHAPTER 6

Bulk Data

6.1 Data Organization

The charts in this, and the following sections show the basic relationships among the data statements for analysis using *GENESIS* for structural analysis. These include all commands that may be included in the input data file for executive control, solution control and bulk data.

The solution control and bulk data is dependent on the particular analysis being performed, although all types of data may be included in a single run.

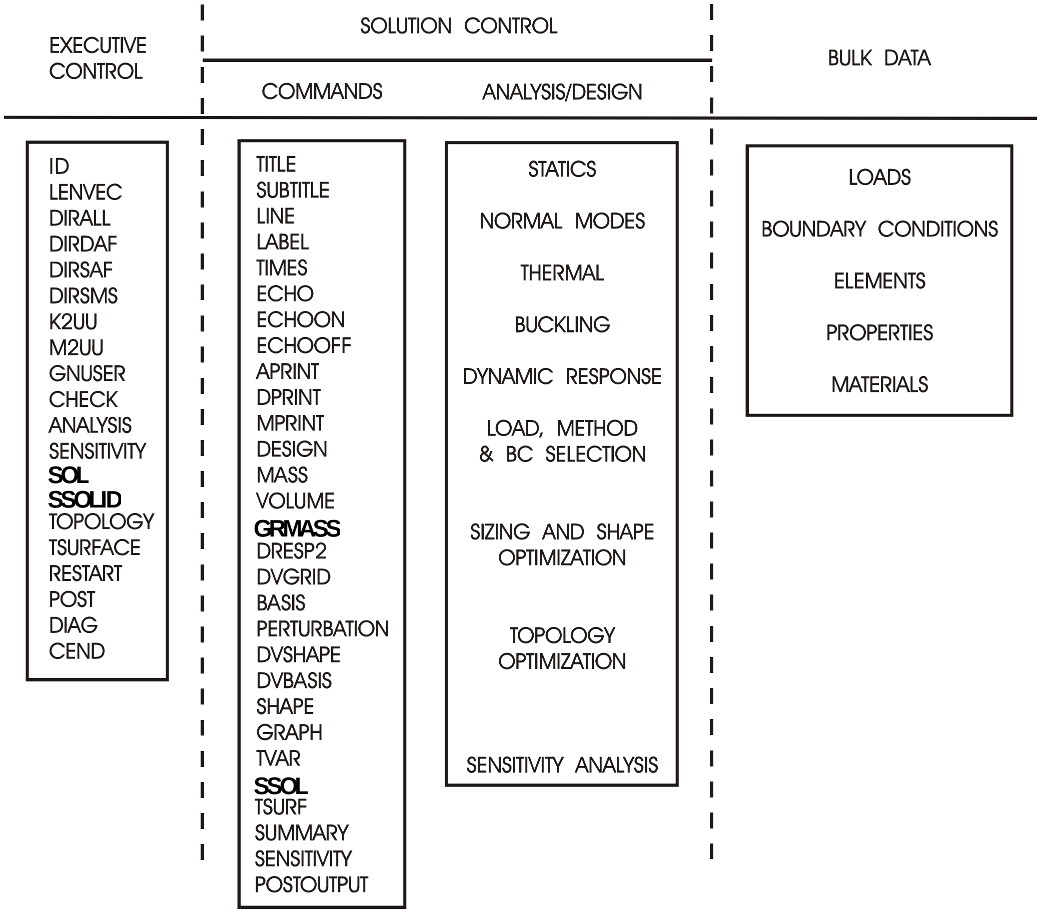


Figure 6-1 General Data Structure

6

6.2 Static and Buckling Analysis Data Relationships

The charts below shows the basic relationships among the data statements for using *GENESIS* for static and buckling analysis. The first chart includes all commands that may be included in the input data file for solution control while the second chart includes all commands that may be included in the input data file for bulk data.

These charts, as well as those in the following sections, can be used to rapidly find the command names required to input a structural model for a particular analysis task. For example, if the user needs to create an analysis model with CTRIA3 elements, the chart shows that PSHELL data must reference MAT1, MAT2 or MAT8 data. Also, the element can be loaded using PLOAD2, PLOAD4 or PLOAD5 data. The chart also shows that the loads must be activated with a LOAD command in the executive control. Further, the chart shows that the LOAD must be included in a LOADCASE.

In general, these charts may be used to be sure that all data is supplied which is appropriate to a particular analysis task.

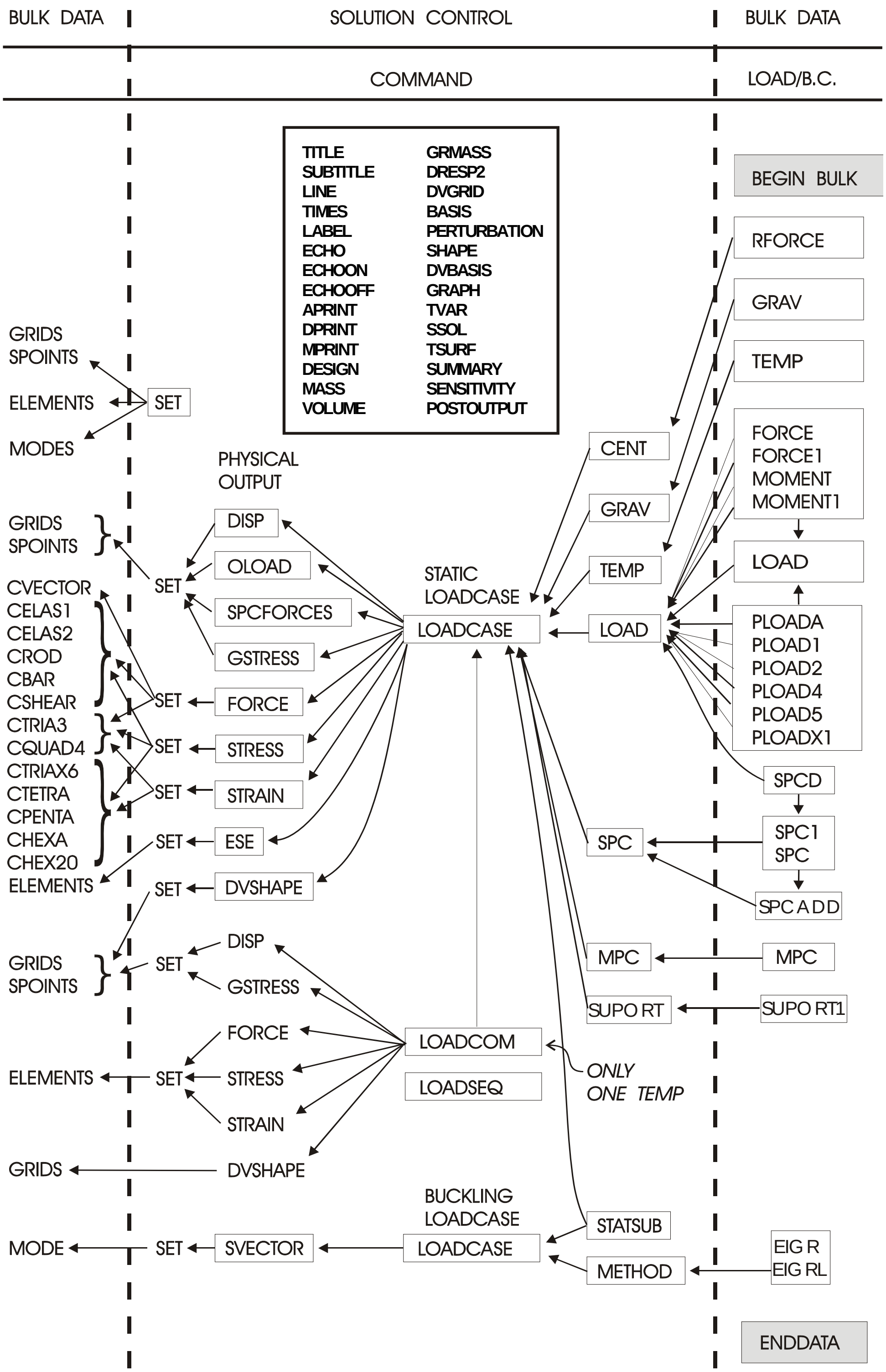


Figure 6-2 Solution Control for Static Analysis

6

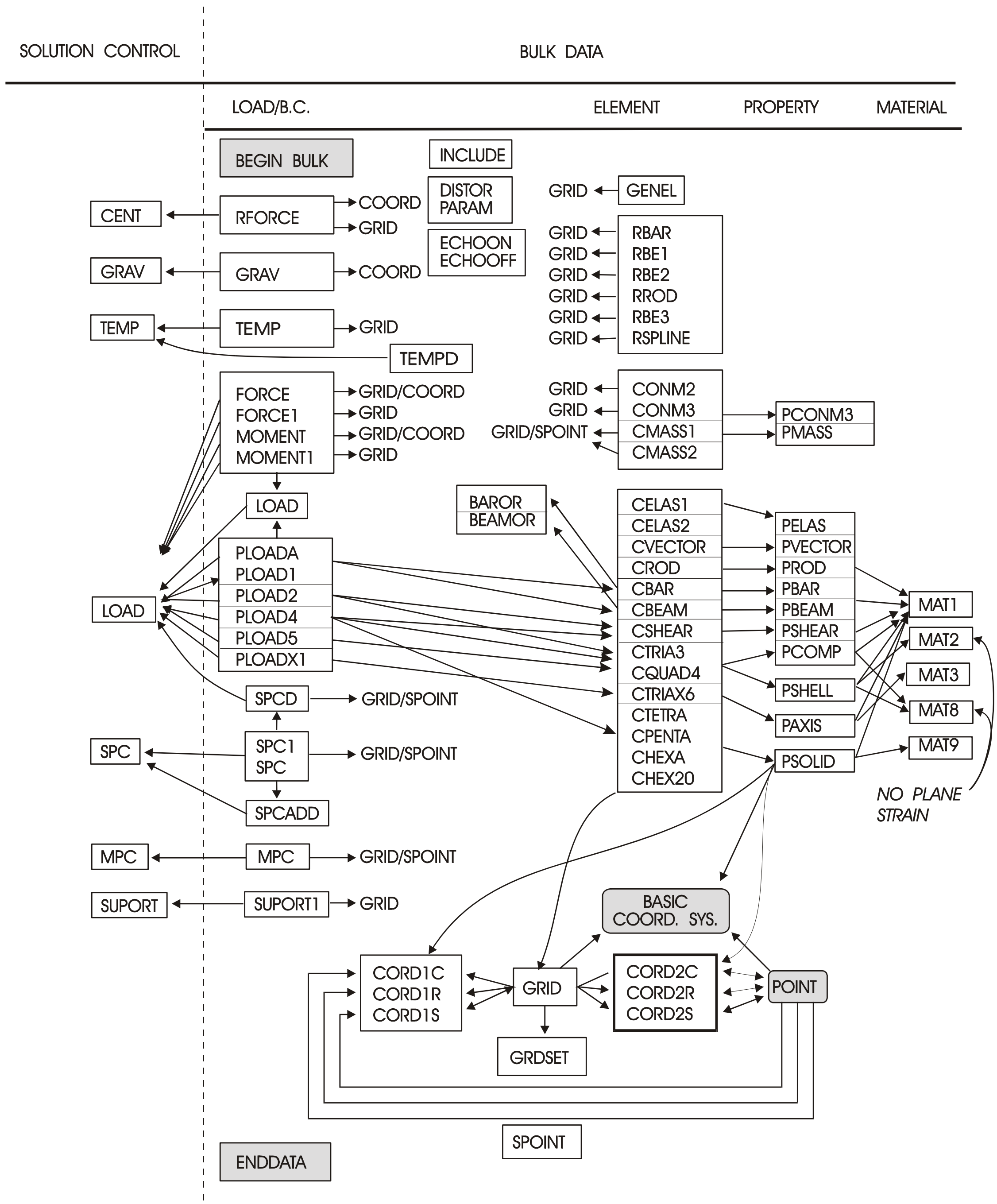


Figure 6-3 Bulk Data for Static Analysis

6.3 Normal Modes Analysis Data Relationships

The charts below show the basic relationships among the data statements for using *GENESIS* for normal modes analysis. They include all commands that may be included in the input data file for solution control and bulk data.

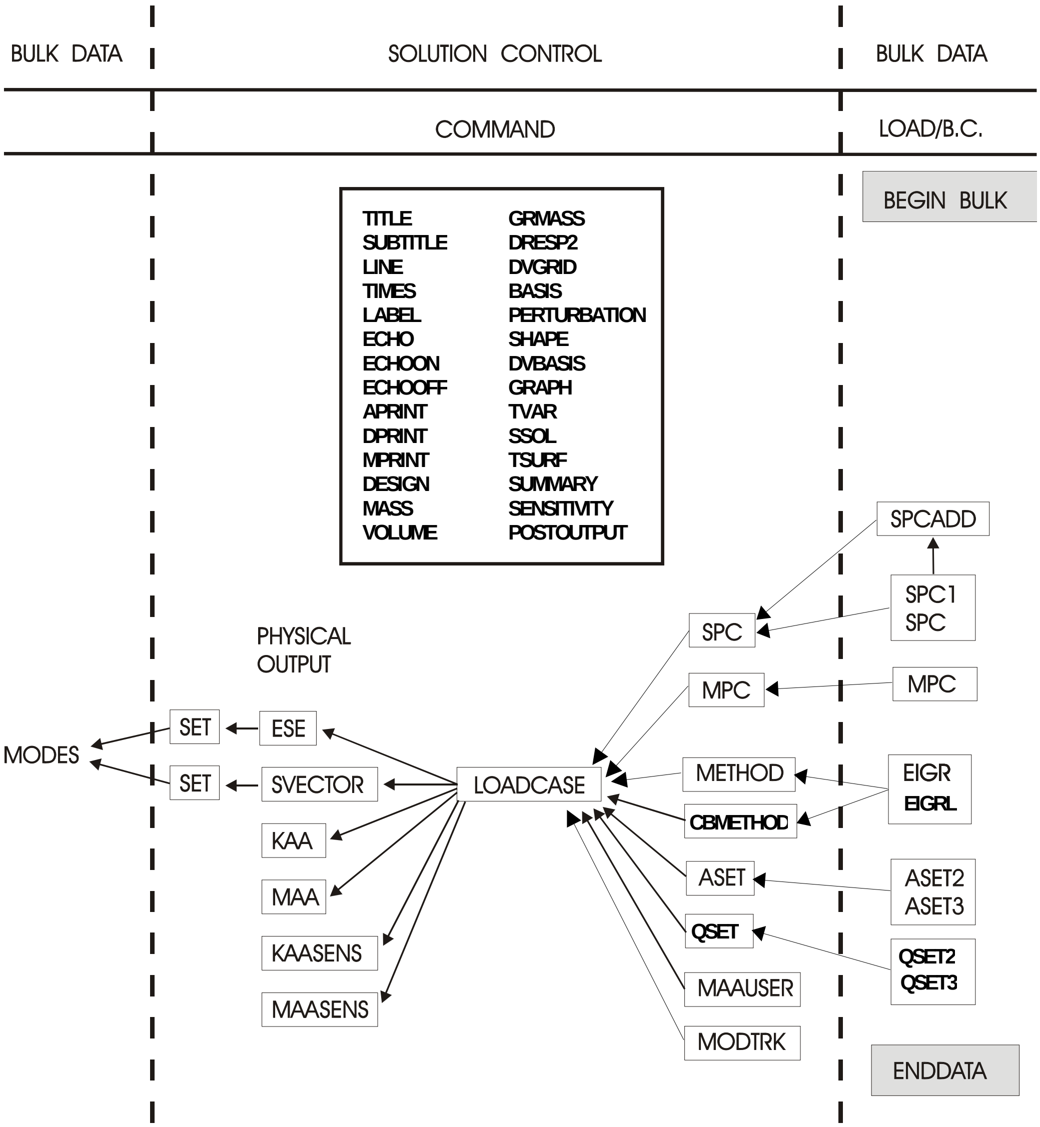


Figure 6-4 Solution Control for Normal Modes Analysis

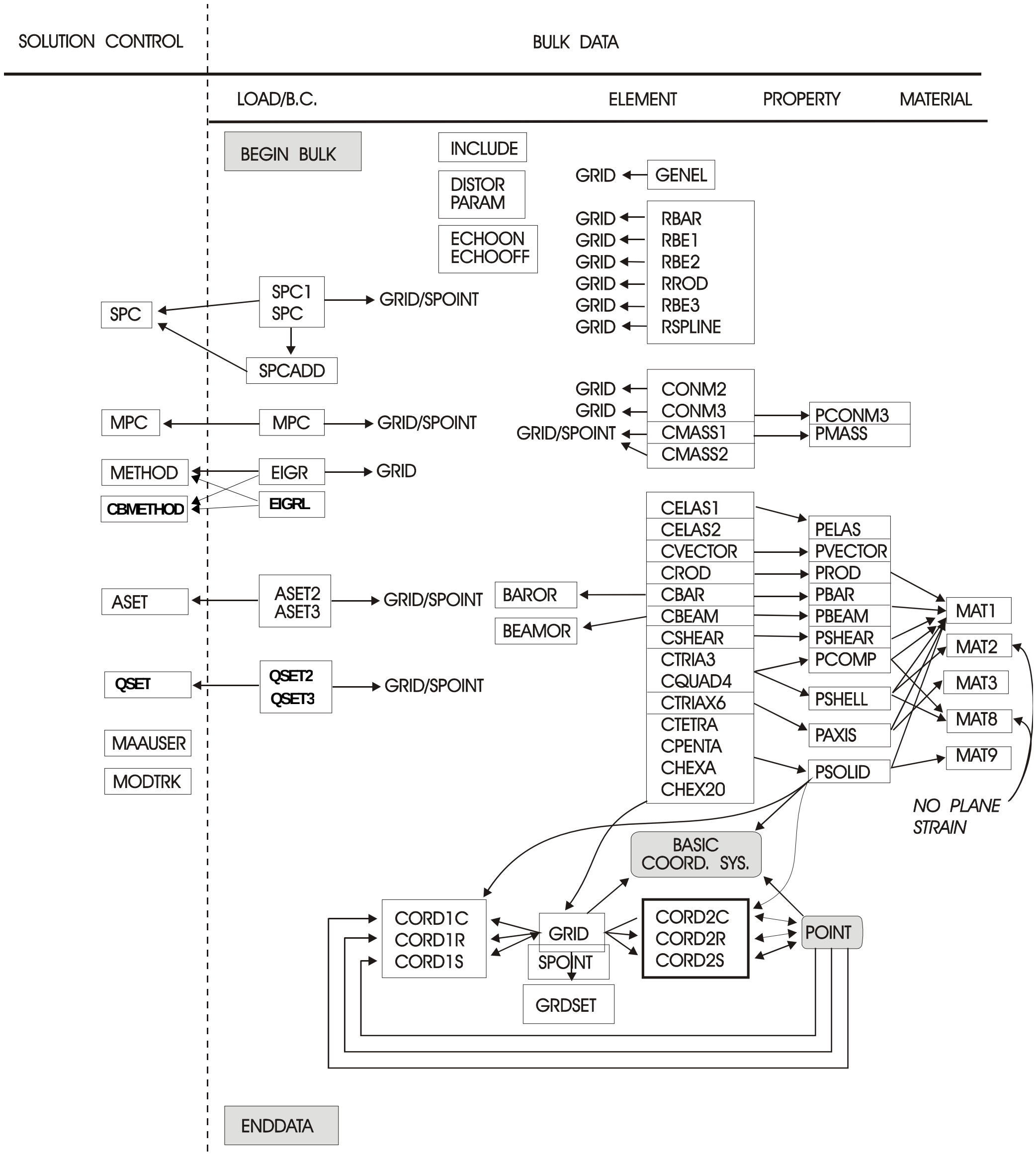


Figure 6-5 Bulk Data for Normal Modes Analysis

6.4 Thermal Analysis Data Relationships

The charts below show the basic relationships among the data for heat transfer analysis using *GENESIS*. This data may be included with structural analysis data to simultaneously perform structural and thermal analysis and design.

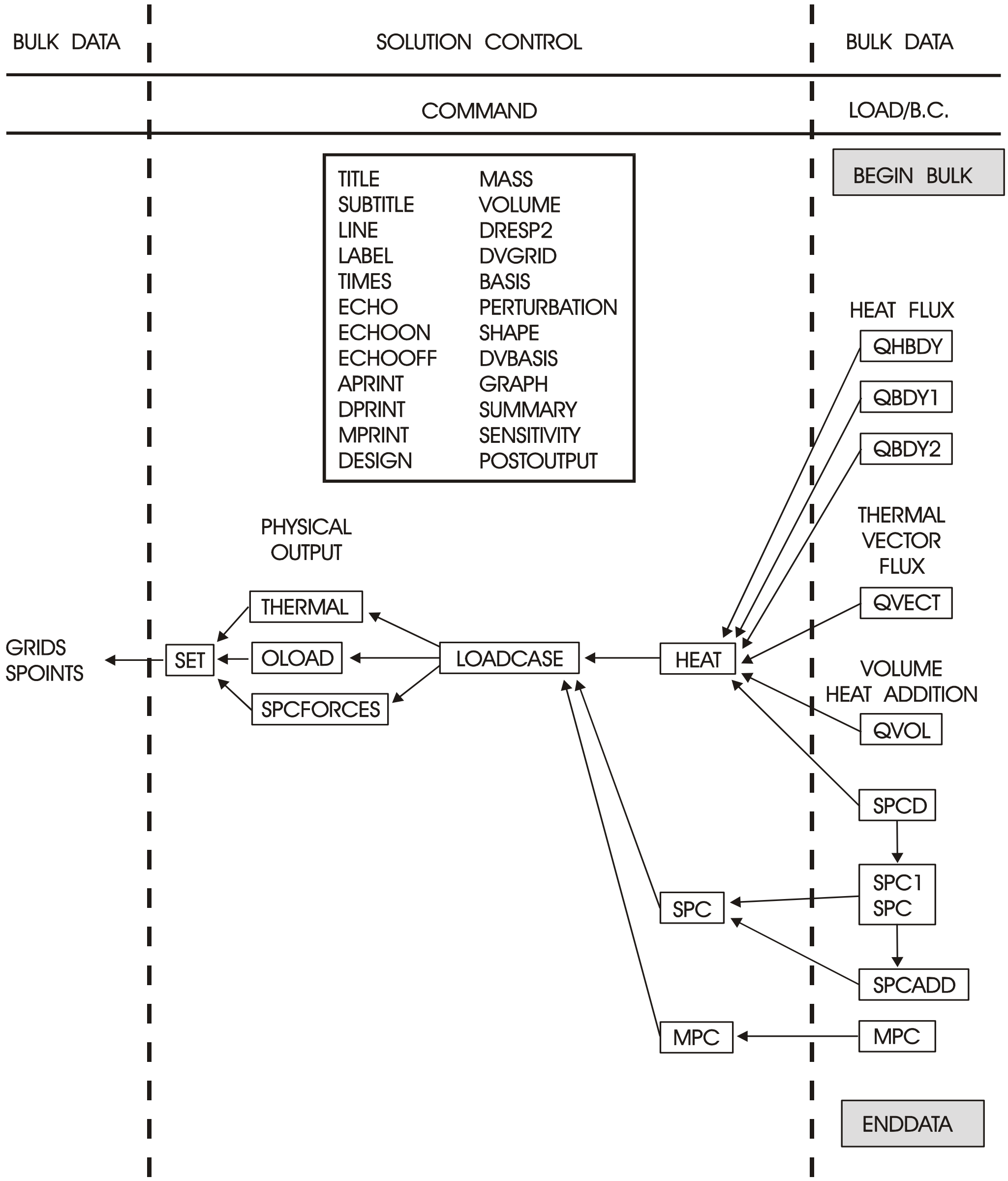


Figure 6-6 Solution Control for Thermal Analysis

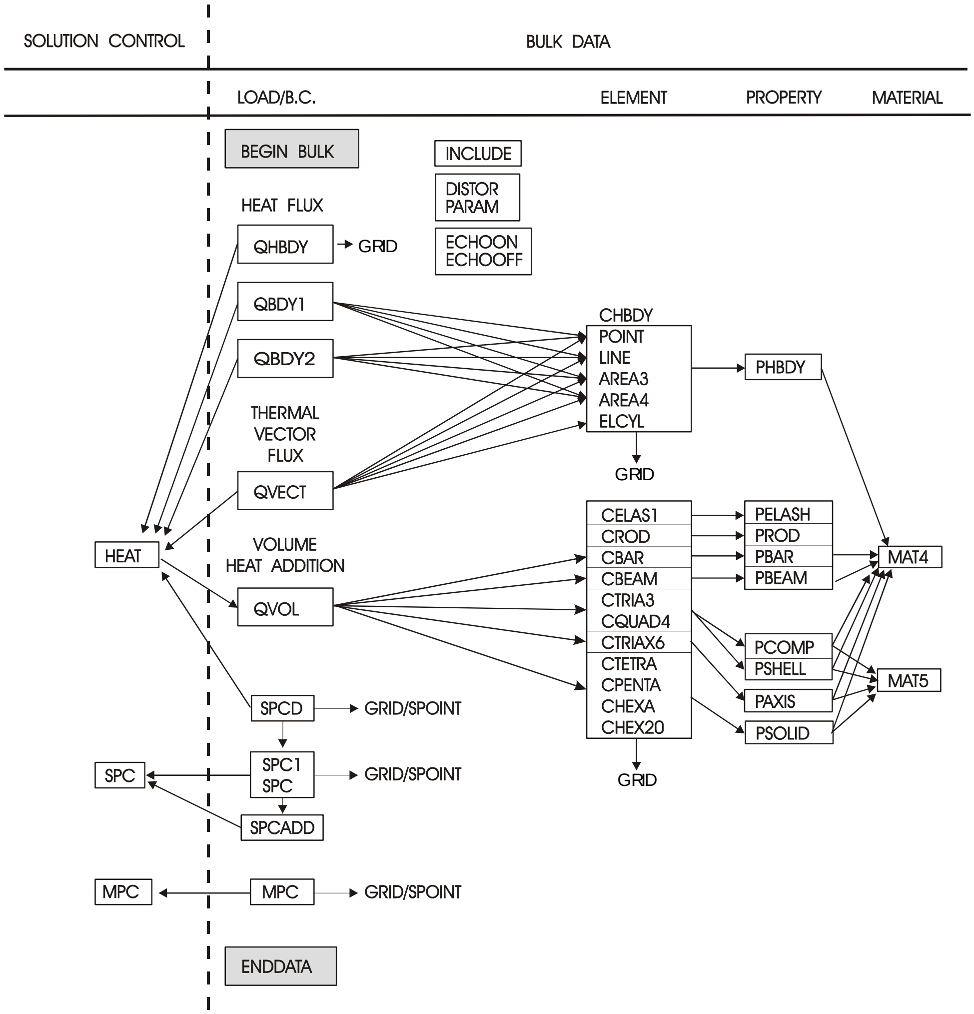


Figure 6-7 Bulk Data for Thermal Analysis

6.5 Frequency Response Analysis Data Relationships

The charts below show the basic relationships among the data statements for frequency response analysis using *GENESIS*. They include all commands that may be included in the input data file for solution control and bulk data.

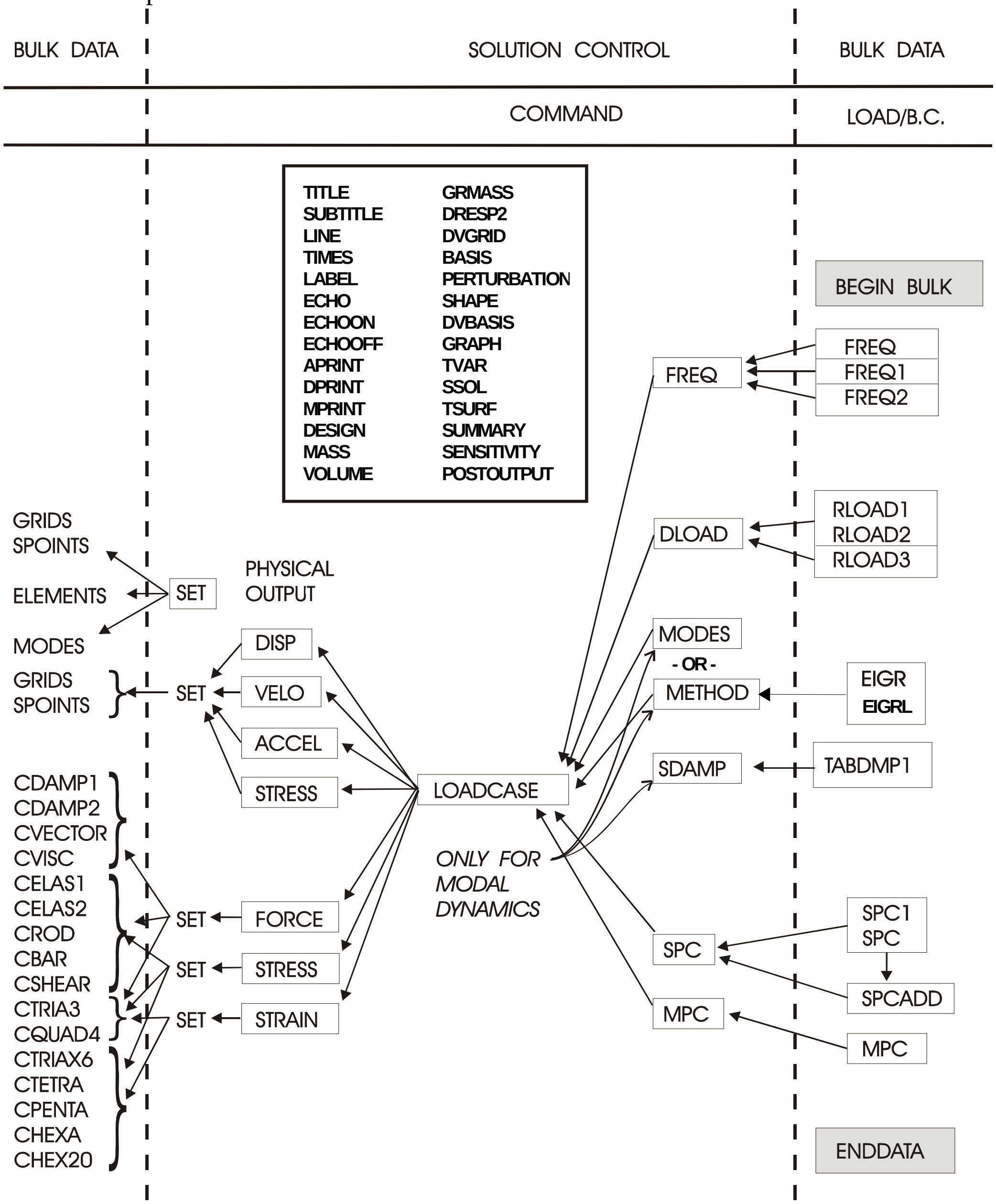


Figure 6-8 Solution Control for Dynamic Analysis

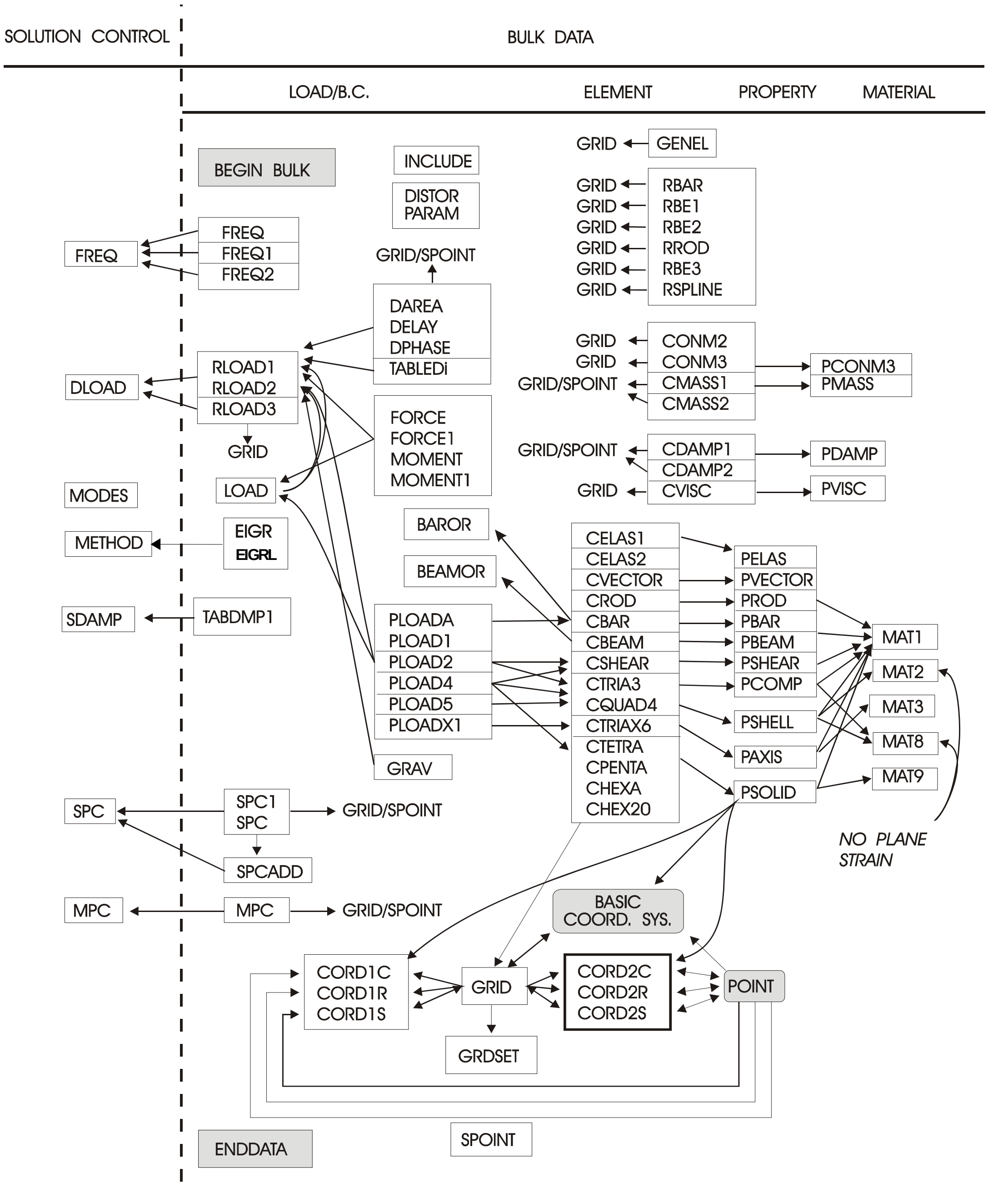


Figure 6-9 Bulk Data for Dynamic Analysis

6.6 Random Response Analysis Data Relationships

The chart below shows the basic relationships among the data statements for random response analysis. This chart includes all random data entris that may be included in the input data file in the solution control and bulk data areas for random response analysis (RANDOM, RANDPS, RANDT1 and TABRND1) as well as the relevant output contol commands (DISP, VELO, ACCE, STRESS, STRAIN and FORCE).

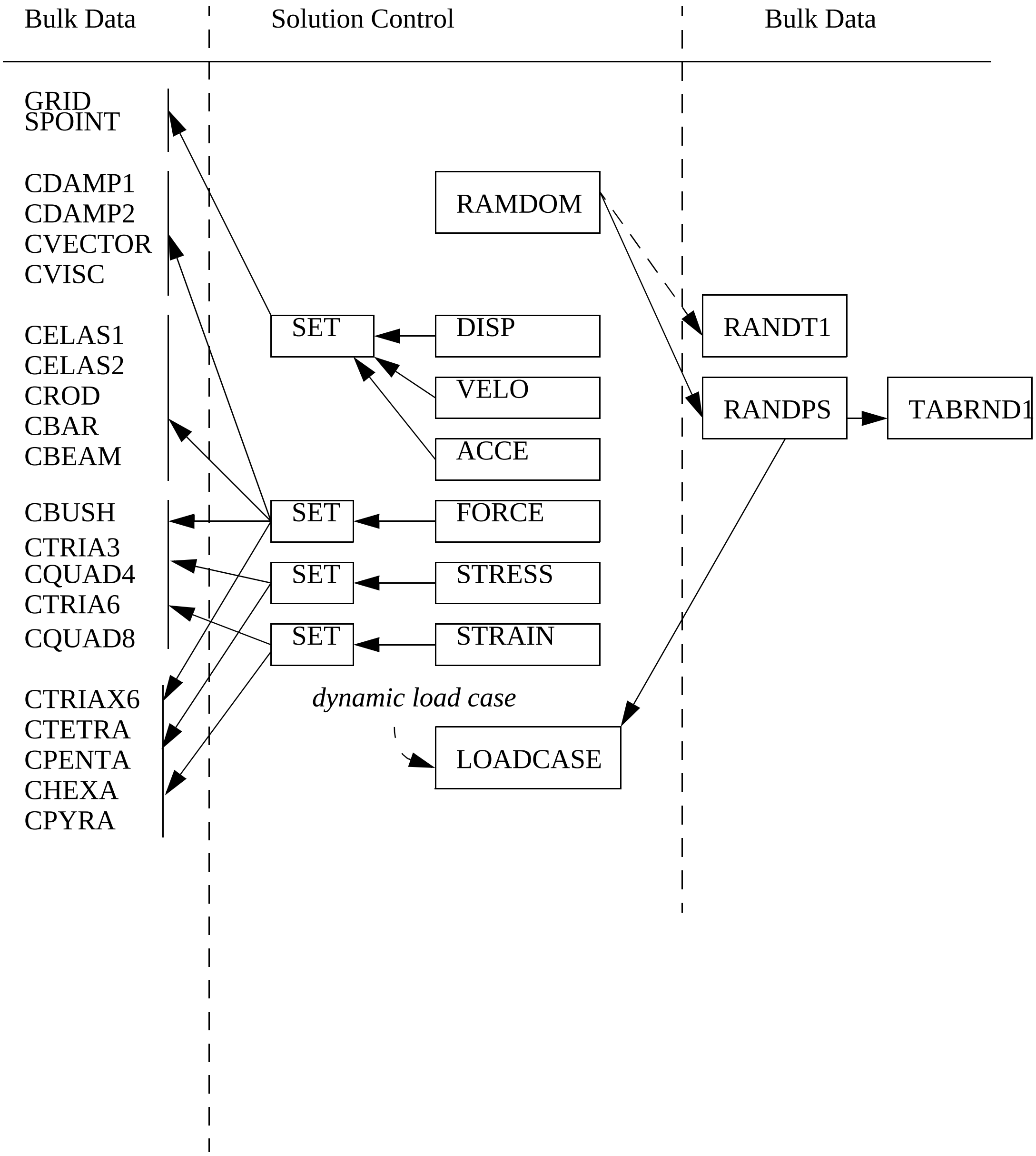


Figure 6-10 Basic Data for Random Analysis

6.7 Bulk Data

The bulk data for analysis using *GENESIS* is defined in this section. The data is given in alphabetical order.

6.7.1 $

Data Entry: **$** - Comment

Description: Enter a comment line.

Format:

$ Character data

Example:

$ This line is a comment.

6

6.7.2 ASET2

Data Entry: **ASET2 -** Degrees of Freedoms Selection

Description: Define a set of independent degrees of freedoms used in eigenvalue loadcases for Guyan reduction.

Format:

1	2	3	4	5	6	7	8	9	10
ASET2	SID	G1	C1	G2	C2	G3	C3		
+	G4	C4	G5	C5	-etc.-				

Example:

1	2	3	4	5	6	7	8	9	10
ASET2	10	1	123456	2	123456				

Field	Information	Description
2	SID	Set identification number of ASET (Integer>0).
3, 5, ...	Gi	**GRID** or **SPOINT** identification numbers (integer > 0).
4, 6, ...	Ci	Component number of Global Coordinate (any unique combination of the digits 1-6 (with no embedded blanks).

Remarks:

1. Degrees of freedoms specified on this data must not be constrained with SPC1, SPC, MPC, rigid elements or interpolation elements.
2. ASET sets must be selected in the Solution Control section (**ASET** = SID or **BOUNDARY** = SID) to be used.
3. The component numbers must be blank for SPOINTs.
4. There is no limit in the number of continuation lines.
5. Continuation data is optional.
6. ASET2 is an alternate format to the ASET3 data statement.
7. See **Guyan Reduction** (Sec. 2.13.1) for a general discussion.

6.7.3 ASET3

Data Entry: **ASET3 -** Degrees of Freedom Selection

Description: Define a set of independent degrees of freedoms used in eigenvalue loadcase for Guyan reduction.

Format:

1	2	3	4	5	6	7	8	9	10
ASET3	SID	C	G1	G2	G3	G4	G5	G6	
+	G7	G8	G9	-etc.-					

Example:

1	2	3	4	5	6	7	8	9	10
ASET3	10	123456	1	2	3	4	5	6	
+	7	8	9	10					

Alternate Format:

1	2	3	4	5	6	7	8	9	10
ASET3	SID	C	G1	THRU	G2				

Example:

	2	3	4	5	6	7	8	9	10
ASET3	10	123456	1	THRU	10				

Field	Information	Description
2	SID	Set identification number of ASET (Integer>0).
3	C	Component number of global coordinate (any unique combination of the digits 1-6,wiht no embedded blanks).
4, 5, ...	Gi	**GRID** or **SPOINT** identification numbers (integer > 0).

Remarks:

1. Degrees of freedoms specified on this data must not be constrained with SPC1, SPC, MPC, rigid elements or interpolation elements.
2. ASET sets must be selected in the Solution Control section (**ASET** = SID or **BOUNDARY** = SID) to be used.
3. The component number must be blank for SPOINTs.
4. There is no limit in the number of continuation lines.
5. Continuation data is optional.
6. ASET3 is an alternate format to the ASET2 data statement.
7. See **Guyan Reduction** (Sec. 2.13.1) for a general discussion.

6.7.4 BAROR

Data Entry: **BAROR** - Beam Element Orientation Default Values.

Description: Defines default values for fields 3 and 6 - 8 of the CBAR data.

Format:

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

Alternate Format:

1	2	3	4	5	6	7	8	9	10
BAROR		PID			X1	X2	X3		

Example:

1	2	3	4	5	6	7	8	9	10
BAROR		39			0.6	2.9	-5.87		

Field	Information	Description
3	PID	Identification number of **PBAR** property data (Integer > 0 or blank).
6-8	X1,X2,X3	Components of vector v, at end A, (see the figure below and the CBAR data description) measured at the offset point for end A, parallel to the components of the general coordinate system for GA, to determine (with the vector from offset end A to offset end B) the orientation of the element coordinate system for the beam element (Real or blank). See Remark 3.
6	GO	**GRID** point identification number to optionally supply X1, X2, X3 (Integer > 0). See Remarks 3 and 5.

Remarks:

1. The contents of fields on this data will be assumed for any CBAR data whose corresponding fields are blank.
2. For an explanation of bar element geometry, see **CBAR** (p. 393).
3. If Field 6 is integer, GO is used. If Field 6 is blank or real, then X1, X2, X3 is used. If X1 or GO is blank, X1=0.0 is assumed.
4. Only one BAROR entry may appear in the data entry section. A typical use of BAROR data is to define the orientation of all bar elements in a planar frame.
5. The orientation vector is ignored for heat transfer analysis.
6. If GO is specified, then the bar orientation vector is updated at each design cycle when shape optimization is being performed.

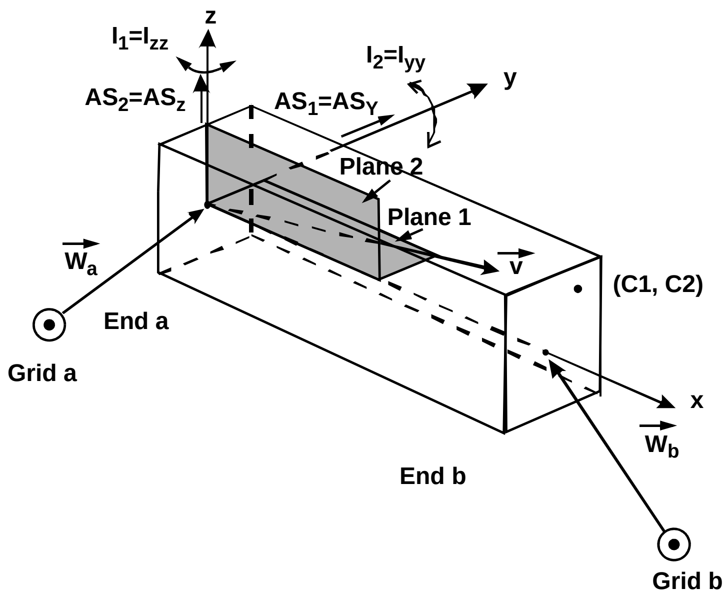

z
I_1=I_{zz}
I_2=I_{yy}
y
AS_2=AS_z
AS_1=AS_Y
Plane 2
Plane 1
$\vec{W_a}$
$\vec{v}$
(C1, C2)
End a
Grid a
x
$\vec{W_b}$
End b
Grid b

Figure 6-11

6.7.5 BCPADD

Data Entry: **BCPADD** - Contact Surface Pair Set Combination.

Description: Defines a new BCPAIR set as a union of BCPAIR entries.

Format:

1	2	3	4	5	6	7	8	9	10
BCPADD	BID	BID1	BID2	BID3	BID4	BID5	BID6	BID7	
+	BID8	BID9	-etc.-						

Example:

1	2	3	4	5	6	7	8	9	10
BCPADD	10	1	3	7					

Field	Information	Description
2	BID	Unique contact pair set Identification number (Integer > 0).
3, 4, 5, ...	BIDi	Contact pair set ID used by **BCPAIR** entries (Integer > 0).

Remarks:

1. Contact pair sets are not used unless selected in the solution control section (**BCONTACT** = BID).
2. The contact pair identification numbers (BIDi) must be unique.
3. **BCPAIR** sets defined by other BCPADD entries may not be referenced.
4. The **BCONTACT** set identification number defined by BCPADD must not also be used by **BCPAIR** bulk data entries.

6

6.7.6 BCPAIR

Data Entry: **BCPAIR** - Contact Surface Pair Definition.

Description: Defines a pair of contact surfaces and associated contact properties.

Format:

1	2	3	4	5	6	7	8	9	10
BCPAIR	BID	SSID	MSID	SRCHDIS	GPAD	PNLTY	MU1		

Example:

1	2	3	4	5	6	7	8	9	10
BCPAIR	5	10	19						

Field	Information	Description
2	BID	Identification number of contact pair set (Integer > 0).
3,4	SSID,MSID	Surface identification numbers defined by **BSURFE**, **BSURFM** or **BSURFP** (Integer > 0).
5	SRCHDIS	Search distance criteria for creating contact points. (Real $\geq$ 0.0. Default = 0.0). See remark 5.
6	GPAD	Padding of interface to account for additional layers. (Real or blank or IGNORE or ADJUST. Default = 0.0). See remark 6.
7	PNLTY	Penalty adjustment coefficient. Real > 0.0 or blank. (Default = 1.0). See remark 7.
8	MU1	Coefficient of static friction. (Real $\geq$ 0.0. Default = 0.0)

Remarks:

1. Contact pair sets are not used unless selected in the Solution Control Section (**BCONTACT** = BID) or referenced by a selected **BCPADD** data.
2. SSID and MSID must be unique.
3. The intersection of all element faces belonging to surface SSID and all element faces belonging to surface MSID surfaces must be null. This is checked during pre-processing and an error is reported if common element faces are present.
4. The direction of closing contact is determined by the normal directions of the element faces in surfaces SSID and MSID. In order for potential contact points to be created, the normals of faces in each surface should be pointed toward the opposite surface.

6

5. Contact points are created at faces where the normal distance to the closest opposite face is less than SRCHDIS. If SRCHDIS is 0.0, then any distance is acceptable. When calculating the normal distance to a surface face created by a CQUAD4/CQUAD8/CTRIA3/CTRIA6 element, if PARAM,**BCSRCH** is set to 1, the default, the face is located at the top surface of the element. If PARAM,**BCSRCH** is set to 0, the face is located at the reference plane of the element.
6. The initial offset of a contact point is the amount of relative displacement that can occur before the corresponding surface points are considered to be in contact. The initial offset of each contact point is calculated as the distance between the surface points minus GPAD. If GPAD is 'IGNORE', the distance between the contact surfaces is ignored, and the initial offset is set as zero at all contact points. If GPAD is 'ADJUST', then the initial offset is set equal to the calculated distance between surfaces except at points where that distance is less than the value of PARAMeter **BCADJUST**. Any contact point whose computed surface distance is less than BCADJUST has its initial offset set to zero.
7. The default maximum penalty stiffness is determined from the normal stiffness of surrounding elements at each contact point. This automatic penalty is multiplied by the value of PNLTY to set the actual maximum penalty stiffness. The penalties of each individual contact point start out relatively low, and are automatically increased during the solution process when excess penetration is measured, up to the maximum penalty stiffness. The default should be adequate for most problems. If the loading is unusually large, excessive penetration may exist in the final solution. In this case, it will be necessary to set PNLTY greater than 1.0. If PNLTY is set too large, however, this may cause numerical instability and prevent convergence.

6.7.7 BDYOR

Data Entry: **BDYOR** - Default Values for CHBDYG and CHBDYP.

Description: Defines the default values for **CHBDYG** and **CHBDYP** data.

Format:

1	2	3	4	5	6	7	8	9	10
BDYOR	TYPE						PID		
+		V1	V2	V3					

Example:

1	2	3	4	5	6	7	8	9	10
BDYOR	LINE						7		
+		1.0	0.0	0.0					

Field	Information	Description
2	TYPE	Type of area involved (must be one of “POINT”, “LINE”, “ELCYL”, “TUBE”, “AREA3”, “AREA4”, “AREA6” or “AREA8”).
8	PID	Identification number of a **PHBDY** entry (Integer > 0 or Blank).
3-5	V1,...,V3	Vector (in the basic coordinate system) used for element orientation (Real or Blank).

Remarks:

1. Only one BDYOR data may appear in the data entry section.
2. The TYPE, PID, V1, V2 and V3 fields can be blank.
3. BDYOR specifies the default values for CHBDYG and CHBDYP data. If a certain field in CHBDYG or CHBDYP is blank, the data in the corresponding field in BDYOR is used.

6.7.8 BEAMOR

Data Entry: **BEAMOR** - Beam Element Orientation Default Values.

Description: Defines default values for fields 3 and 6 - 8 of the CBEAM data.

Format:

1	2	3	4	5	6	7	8	9	10
BEAMOR		PID			GO				

Alternate Format:

1	2	3	4	5	6	7	8	9	10
BEAMOR		PID			X1	X2	X3		

Example:

1	2	3	4	5	6	7	8	9	10
BEAMOR		39			0.6	2.9	-5.87		

Field	Information	Description
3	PID	Identification number of **PBEAM** property data (Integer > 0 or blank).
6-8	X1,X2,X3	Components of vector v, at end A, (see the figure below and the CBEAM data description) measured at the offset point for end A, parallel to the components of the general coordinate system for GA, to determine (with the vector from offset end A to offset end B) the orientation of the element coordinate system for the beam element (Real or blank). See Remark 3.
6	GO	**GRID** point identification number to optionally supply X1, X2, X3 (Integer > 0). See Remarks 3 and 5.

6

Remarks:

1. The contents of fields on this data will be assumed for any CBEAM data whose corresponding fields are blank.
2. For an explanation of beam element geometry, see **CBEAM**.
3. If Field 6 is integer, GO is used. If Field 6 is blank or real, then X1, X2, X3 is used. If X1 or GO is blank, X1=0.0 is assumed.
4. Only one BEAMOR entry may appear in the data entry section. A typical use of BEAMOR data is to define the orientation of all beam elements in a planar frame.
5. The orientation vector is ignored for heat transfer analysis.
6. If GO is specified, then the beam orientation vector is updated at each design cycle when shape optimization is being performed.

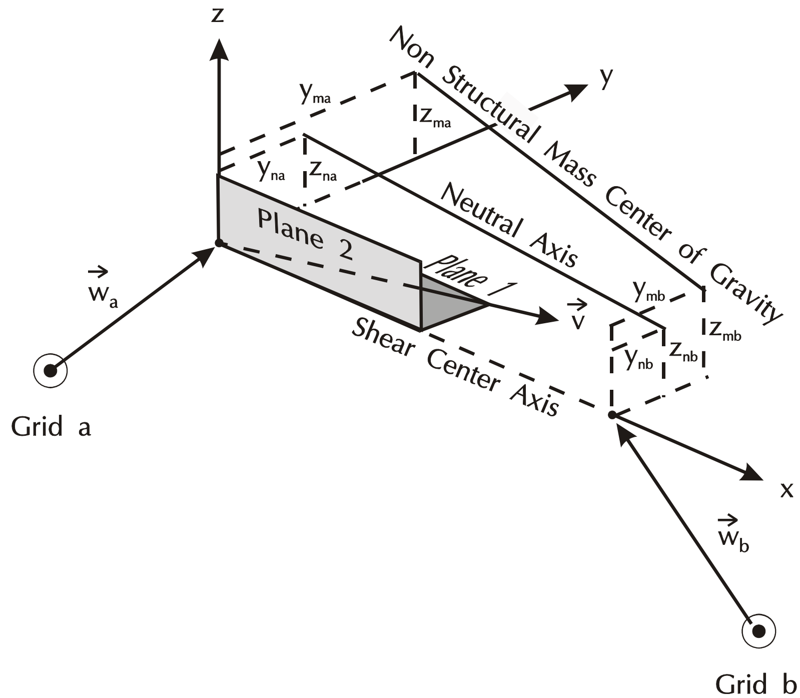

z
y
x
Non Structural Mass Center of Gravity
Neutral Axis
Shear Center Axis
Plane 2
Plane 1
y_{ma}
z_{ma}
y_{na}
z_{na}
y_{mb}
z_{mb}
y_{nb}
z_{nb}
$\vec{w}_a$
$\vec{w}_b$
$\vec{v}$
Grid a
Grid b

Figure 6-12

6.7.9 BOLT

Data Entry: **BOLT** - Bolt Interpolation Element.

Description: Defines MPC equations to connect top and bottom portions of a bolt with a bolt control grid.

Format:

1	2	3	4	5	6	7	8	9	10
BOLT	EID	GC							
+	"TOP"	GT1	GT2	GT3	GT4	GT5	GT6	GT7	
+		GT8	-etc.-						
+	"BOTTOM"	GB1	GB2	GB3	GB4	GB5	GB6	GB7	
+		GB8	-etc.-						

Example:

1	2	3	4	5	6	7	8	9	10
BOLT	100	1001							
+	TOP	11	12	13	14	15	16		
+	BOTTOM	111	112	113	114	115	116		

Field	Information	Description
2	EID	Identification number of the interpolation element (Integer > 0).
3	GC	**GRID** identification number of the bolt control grid (Integer > 0).
2	"TOP"	Indicates the start of the list of top grids. (Character).
3-9	GTi	GRID identification numbers of the top grids (Integer > 0).
2	"BOTTOM"	Indicates the start of the list of bottom grids. (Character).
3-9	GBi	GRID identification numbers of the bottom grids (Integer > 0).

Remarks:

1. The bolt control grid should not be connected to any other element.
2. The number of top grids must be equal to the number of bottom grids. Each grid pair (GTi, GBi) should be coincident.
3. Each grid pair (GTi, GBi) will be connected to the bolt control grid, GC, with the following MPC equations:

$$u_{bx} - u_{tx} - u_{cx} = 0$$
$$u_{by} - u_{ty} - u_{cy} = 0$$
$$u_{bz} - u_{tz} - u_{cz} = 0$$

4. All components of the bottom grids will be made dependent. Degrees of freedom specified as dependent may not be listed as dependent on other rigid or interpolation elements or **MPC**. Also, dependent degrees of freedom may not be listed on **SPC**, **SPC1**, **ASET2**, **ASET3** or **SUPORT1** data entries.
5. Element identification numbers must be unique with respect to all other element identification numbers.
6. Interpolation elements, unlike MPCs, are not selected through the Solution Control Section.
7. Forces of constraint are not recovered. **FORCE** will produce no output for this element.
8. Interpolation elements are ignored in heat transfer analysis.
9. See **BOLT Element** (Sec. 2.8.3) for a general discussion.

6

6.7.10 BPOINTG

Data Entry: **BPOINTG** - Point Collection Definition by Grid.

Description: Defines a collection of points by grid ID list.

Format:

1	2	3	4	5	6	7	8	9	10
BPOINTG	ID	GID1	GID2	GID3	GID4	GID5	GID6	GID7	
+	GID8	GID9	etc						

Example:

1	2	3	4	5	6	7	8	9	10
BPOINTG	11	1	2	3					

Field	Information	Description
2	ID	Unique point collection identification number (Integer > 0).
3,4,5..	GIDi	**GRID** identification number. At least one is required.

Remarks:

1. ID must be unique with respect to all other BPOINTG identification numbers.
2. Any number of continuation lines may be used to specify any number of grids.

6.7.11 BSURFE

Data Entry: **BSURFE** - Surface Definition by Element Faces.

Description: Defines a surface by element faces. The surface is the union of all specified element faces

Format:

1	2	3	4	5	6	7	8	9	10
BSURFE	ID	EID1	GA1	GB1	NORM1				
+		EID2	GA2	GB2	NORM2				
+		EID3	GA3	GB3	NORM3				
+		etc	etc	etc	etc				

Example:

1	2	3	4	5	6	7	8	9	10
BSURFE	100	121	19	32					
+		76	21	122					

Field	Information	Description
2	ID	Unique surface identification number (Integer > 0).
3	EIDi	Element identification number (Integer > 0). See Remark 2.
4	GAi	Blank for shell elements. For solid elements, GAi is the identification number of a **GRID** connected to a corner of element face (Integer > 0 or blank).
5	GBi	Blank for shell elements and triangular face of **CPENTA**. Identification number of the **GRID** connected to the corner diagonally opposite to GAi on the face of **CHEXA** or **CPENTA** element. For **CTETRA**, GBi is the indentification of the **GRID** located at the corner not on the face. For triangular faces of **CPYRA** elements, GBi is the identification of a GRID shared by both the triangular face and quadrilateral face. For quadrilateral face of CPYRA element, GBi is the identification number of the GRID connected to a corner diagonally opposite to GAi on the quadrilateral face(Integer > 0 or blank).
6	NORMi	Blank for solid elements. For shell elements, defines the normal of the surface face (Integer -1 or 1 or blank. Default = 1). See Remark 4.

6

Remarks:

1. ID must be unique with respect to all other BSURFE, BSURFP and BSURFM identification numbers.
2. BSURFE surfaces may be composed of **CHEXA**, **CTETRA**, **CPENTA**, **CPYRA**, **CQUAD4**, **CTRIA3**, **CQUAD8** and **CTRIA6** element faces only. CQUAD4, CQUAD8, CTRIA3 and CTRIA6 elements must not reference **PSKIN**.
3. Any number of conitinuation lines may be used to specify any number of elements.
4. If NORMi is -1 then the normal of the face is opposite to the normal of element EIDi, otherwise, the normal of the face is the same as the normal of the element. The normal of solid element faces always points outward from the solid element.

6.7.12 BSURFM

Data Entry: **BSURFM** - Surface Definition by Material.

Description: Defines a surface by materials. Elements with the specified materials are used to define a surface.

Format:

1	2	3	4	5	6	7	8	9	10
BSURFM	ID	MID1	MID2	MID3	MID4	MID5	MID6	MID7	
+	MID8	MID9	etc						

Example:

1	2	3	4	5	6	7	8	9	10
BSURFM	11	1	2	3					

Field	Information	Description
2	ID	Unique surface identification number (Integer > 0).
3,4,5..	MIDi	Material identification number. At least one is required.

Remarks:

1. ID must be unique with respect to all other BSURFM, BSURFE and BSURFP identification numbers.
2. BSURFM surfaces may be composed of elements whose properties reference **MAT1**, **MAT2**, **MAT8**, **MAT9** and **MAT11** materials only. Only faces of **CHEXA**, **CTETRA**, **CPENTA** and **CPYRA** elements and/or **CQUAD4**, **CTRIA3**, **CQUAD8** and **CTRIA6** elements referencing **PSHELL** will be used to create the surface. For solid elements, only those element faces on the exterior surface of the mesh will be used.
3. The surface normal is the underlying element normal.
4. Any number of continuation lines may be used to specify any number of materials.

6

6.7.13 BSURFP

Data Entry: **BSURFP** - Surface Definition by Property.

Description: Defines a surface by element property. Elements with the specified properties are used to define a surface.

Format:

1	2	3	4	5	6	7	8	9	10
BSURFP	ID	PTYPE	PID1		PID2		PID3		
+	PID4		etc						

Example:

1	2	3	4	5	6	7	8	9	10
BSURFP	5	PSOLID	19		70				

Field	Information	Description
2	ID	Unique surface identification number (Integer > 0).
3	PTYPE	**PSOLID** or **PSHELL** or **PCOMP**. Property type.
4,6,8...	PIDi	Identification number of properties. At least one is required. (Integer > 0)

Remarks:

1. ID must be unique with respect to all other BSURFP, BSURFM and BSURFE identification numbers.
2. BSURFP surfaces may be composed of elements with **PSOLID**, **PSHELL** and **PCOMP** properties only. Only faces of **CHEXA**, **CTETRA**, **CPENTA** and **CPYRA** elements and/or **CQUAD4**, **CTRIA3**, **CQUAD8** and **CTRIA6** elements will be used to create the surface. For solid elements, only those element faces on the exterior surface of the mesh will be used.
3. The surface normal is the underlying element normal.
4. Any number of continuation lines may be used to specify any number of element properties.

6

6.7.14 CBAR

Data Entry: **CBAR -** Bar Element Connection.

Description: Defines a uniform bar element of the structural and/or thermal model.

Format:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	X1	X2	X3		
+	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

Alternate Format:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	GO				
+	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

Examples:

1	2	3	4	5	6	7	8	9	10
CBAR	2	39	7	3	10.5	300.25	50.3		
+			5.13			3.0			

1	2	3	4	5	6	7	8	9	10
CBAR	2	39	7	3	13				
+			5.13			3.0			

6

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of **PBAR** or **PBARL** property data (default is EID if no BAROR data is present), (Integer >0 or blank).
4,5	GA,GB	**GRID** identification numbers of connection points (Integer > 0; $GA \neq GB$).
6-8	X1,X2,X3	Components of vector v, at end A (shown in the figure below), measured at the offset point for end A, parallel to the components of the general coordinate system for GA, to determine (with the vector from offset end A to offset end B) the orientation of the element coordinate system for the beam element (Real or blank. See the **BAROR** data for default options for fields 3 and 6 - 8. See remarks 2 and 6).
6	GO	Grid point identification number to optionally supply X1, X2, X3 (Integer > 0 or blank). See the BAROR data for default options for fields 3 and 6 - 8. See remark 2. Direction of orientation vector is GA to GO.
2,3	PA, PB	Pin flags for beam ends A and B respectively (Up to five of the unique digits 1 - 6 with no embedded blanks; integer > 0 or blank). Used to remove connections between the grid point and selected degrees of freedom of the beam. The degrees of freedom are defined in the element's coordinate system and the pin flags are applied at the offset ends of the beam (see the figure below). The bar must have stiffness associated with the pin flag. For example, if PA=4, the PBAR data must have a nonzero value for J, the torsional stiffness.
4-6	W1A,W2A, W3A	Components of the offset vector, measured in the general coordinate system at grid point a from the grid point to the end point of the neutral axis (Real or blank)
7-9	W1B,W2B, W3B	Components of the offset vector, measured in the general coordinate system at grid point b from the grid point to the end points of the neutral axis (Real or blank)

Remarks:

1. The bar element geometry is shown in the figure below.
2. If data in field 6 is integer, then GO is used. If field 6 is blank then values defined by the **BAROR** statement are used. Finally, if data in field 6 is real, then X1, X2 and X3 are used.
3. $GO \neq GA$ or GB.
4. If there are no pin flags and no offsets the continuation data may be omitted.
5. Element identification numbers must be unique with respect to all other element identification numbers.

6

6. If GO is specified, the beam orientation vector is updated at each design cycle when shape optimization is being performed.
7. Bar elements may be loaded using **PLOADA** or **PLOAD1** data.
8. Element forces are printed in the element coordinate system for ends a and b.
9. Stresses are recovered as indicated in PBAR data.
10. Strains are not recovered.
11. In heat transfer analysis, the orientation vector and pin flags are ignored. The offsets are considered as perfect conduction, i.e. the temperatures do not change in them.

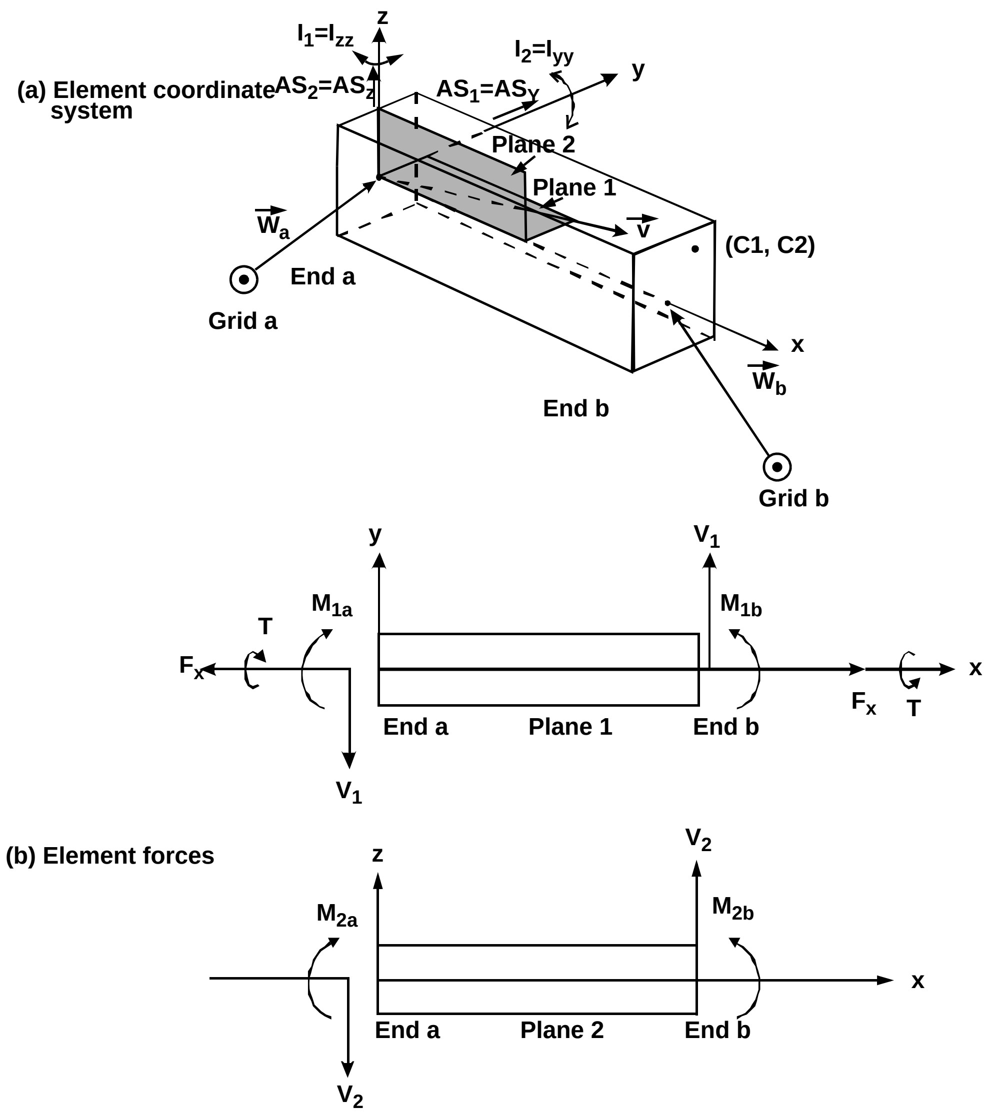


Figure 6-13

6.7.15 CBEAM

Data Entry: **CBEAM -** General Beam Element Connection.

Description: Defines a general beam element of the structural and/or thermal model.

Format:

1	2	3	4	5	6	7	8	9	10
CBEAM	EID	PID	GA	GB	X1	X2	X3		
+	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
+	SA	SB							

Alternate Format:

1	2	3	4	5	6	7	8	9	10
CBEAM	EID	PID	GA	GB	GO				
+	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
+	SA	SB							

Examples:

1	2	3	4	5	6	7	8	9	10
CBEAM	2	39	7	3	10.5	300.25	50.3		
+			5.13			3.0			
+	101	102							

1	2	3	4	5	6	7	8	9	10
CBEAM	2	39	7	3	13				
+			5.13			3.0			

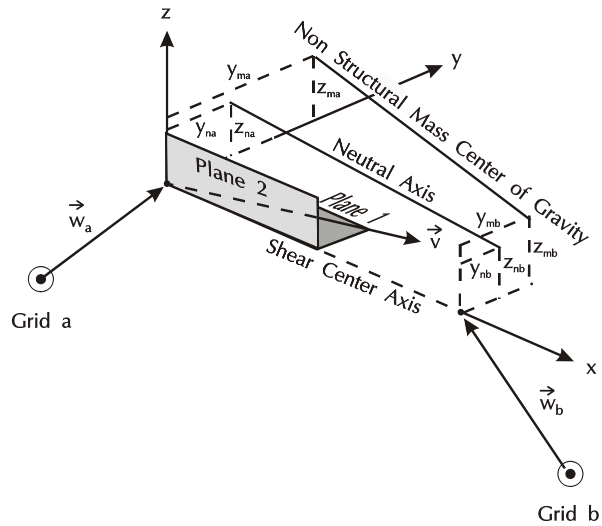
Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of **PBEAM** or **PBEAML** property data (default is EID if no BEAMOR data is present), (Integer >0 or blank).
4,5	GA,GB	**GRID** identification numbers of connection points (Integer > 0; $GA \neq GB$).
6-8	X1,X2,X3	Components of vector v, at end A (shown in the figure below), measured at the offset point for end A, parallel to the components of the general coordinate system for GA, to determine (with the vector from offset end A to offset end B) the orientation of the element coordinate system for the beam element (Real or blank. See the **BEAMOR** data for default options for fields 3 and 6 - 8. See remarks 2 and 6).
6	GO	Grid point identification number to optionally supply X1, X2, X3 (Integer > 0 or blank). See the BEAMOR data for default options for fields 3 and 6 - 8. See remark 2. Direction of orientation vector is GA to GO.
2,3	PA, PB	Pin flags for beam ends A and B respectively (Up to five of the unique digits 1 - 6 with no embedded blanks; integer > 0 or blank). Used to remove connections between the grid point and selected degrees of freedom of the beam. The degrees of freedom are defined in the element's coordinate system and the pin flags are applied at the offset ends of the beam (see the figure below). The beam must have stiffness associated with the pin flag. For example, if PA=4, the PBEAM data must have a nonzero value for J, the torsional stiffness.
4-6	W1A,W2A, W3A	Components of the offset vector, measured in the general coordinate system at grid point a from the grid point to the end point of the shear center axis (Real or blank)
7-9	W1B,W2B, W3B	Components of the offset vector, measured in the general coordinate system at grid point b from the grid point to the end points of the shear center axis (Real or blank)
2,3	SA, SB	**SPOINT** or **GRID** identification numbers for the warping degree of freedom, $d\theta/dx$ at ends A and B (Integer > 0 or Blank)

Remarks:

1. The beam element geometry is shown in the figure below.
2. If data in field 6 is integer, then GO is used. If field 6 is blank then values defined by the **BEAMOR** statement are used. Finally, if data in field 6 is real, then X1, X2 and X3 are used.
3. $GO \neq GA$ or GB.

6

4. The continuation lines may be omitted if there are no pin flags, offsets, or warping degrees of freedom.
5. If warping is included, the second continuation line must also be present, even if all fields are blank.
6. If SA and/or SB reference grid points, the warping degree of freedom is associated with component 1.
7. Element identification numbers must be unique with respect to all other element identification numbers.
8. If GO is specified, the beam orientation vector is updated at each design cycle when shape optimization is being performed.
9. Beam elements may be loaded using **PLOADA** or **PLOAD1** data.
10. Element forces are printed as in the figure below for end A and other stations as indicated on the **PBEAM** data.
11. Stresses are recovered as indicated in PBEAM data.
12. Strains are not recovered.
13. In heat transfer analysis, the orientation vector and pin flags are ignored. The offsets are considered as perfect conduction, i.e. the temperatures do not change in them.


z
y
x
Non Structural Mass Center of Gravity
Neutral Axis
Shear Center Axis
Plane 2
Plane 1
Grid a
Grid b

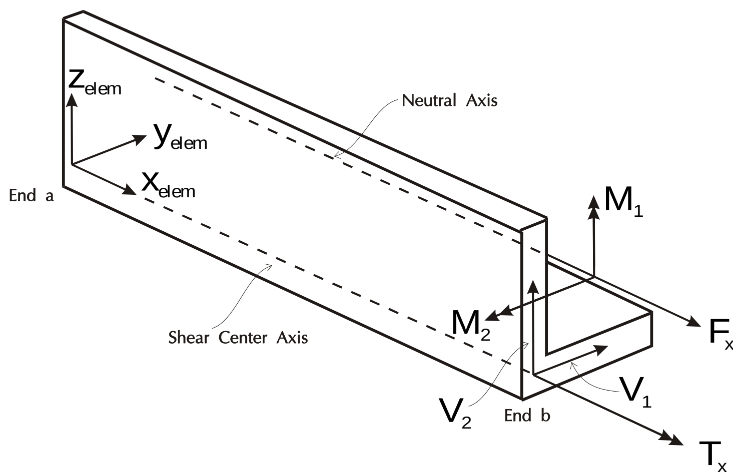

Neutral Axis
Shear Center Axis
End a
End b

Figure 6-14

6.7.16 CBUSH

Data Entry: **CBUSH** - Generalized Elastic Element Connection.

Description: Defines a generalized spring-damper element of the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH	EID	PID	GA	GB	X1	X2	X3	CID	
+	S	OCID	S1	S2	S2				

Alternate Format 1:

1	2	3	4	5	6	7	8	9	10
CBUSH	EID	PID	GA	GB	GO			CID	
+	S	OCID	S1	S2	S2				

Examples:

1	2	3	4	5	6	7	8	9	10
CBUSH	101	3001	206	406	10.5	300.25	50.3		
+	0.3								

1	2	3	4	5	6	7	8	9	10
CBUSH	102	3002	207	407				0	
+		0	2.0	4.5	-1.3				

6

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of **PBUSH** property data (default is EID), (Integer >0 or blank).
4	GA	**GRID** identification number of first connection point (Integer > 0).
5	GB	**GRID** identification numbers of second connection point (Integer > 0 or blank).
6-8	X1,X2,X3	Components of vector v, at end A (shown in the figure below), measured at the offset point for end A, parallel to the components of the general coordinate system for GA, to determine (with the vector from offset end A to offset end B) the orientation of the element coordinate system for the beam element (Real or blank). Only used if CID = blank.
6	GO	Grid point identification number to optionally supply X1, X2, X3 (Integer > 0 or blank). Direction of orientation vector is GA to GO. Only used if CID = blank.
9	CID	Coordinate system identification number defining the element coordinate system. (Integer ≥ 0 or blank). 0 indicates the basic coordinate system. Blank indicates to use Xi or GO along with the end locations to define the element coordinate system.
2	S	Location of the spring damper. (Real or blank: 0.0 ≤ S ≤ 1.0. Default = 0.5). Only used if OCID = -1.
3	OCID	Coordinate system identification number for the location of the spring-damper (Integer ≥ -1 or blank. Default = -1). -1 indicates to use S as the fraction along the line from GA to GB. 0 indicates the basic system.
4-6	Si	Components of the offset vector, measured in the OCID coordinate system located at end A, from GA to the location of the spring-damper (Real or blank)

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. If GA and GB are coincident or if GB is blank, CID must be specified.
3. If CID is not blank, GO/X1/X2/X3 are ignored.
4. If CID identifies a cylindrical or spherical coordinate system, then the coordinates of GA are used to locate the system.
5. If GA and GB are not coincident, then GO/X1/X2/X3 and CID may all be blank if only K1 and/or K4 are specified on the corresponding PBUSH. In this case, the line from GA to GB defines the element x-axis.
6. Element results are in the element coordinate system.

7. If OCID = -1 or blank then S is used and S1,S2,S3 are ignored. If OCID≥0 then S is ignored and S1,S2 and S3 are used.
8. The spring damper has zero length. Its ends are connected to grids GA and GB with rigid links.

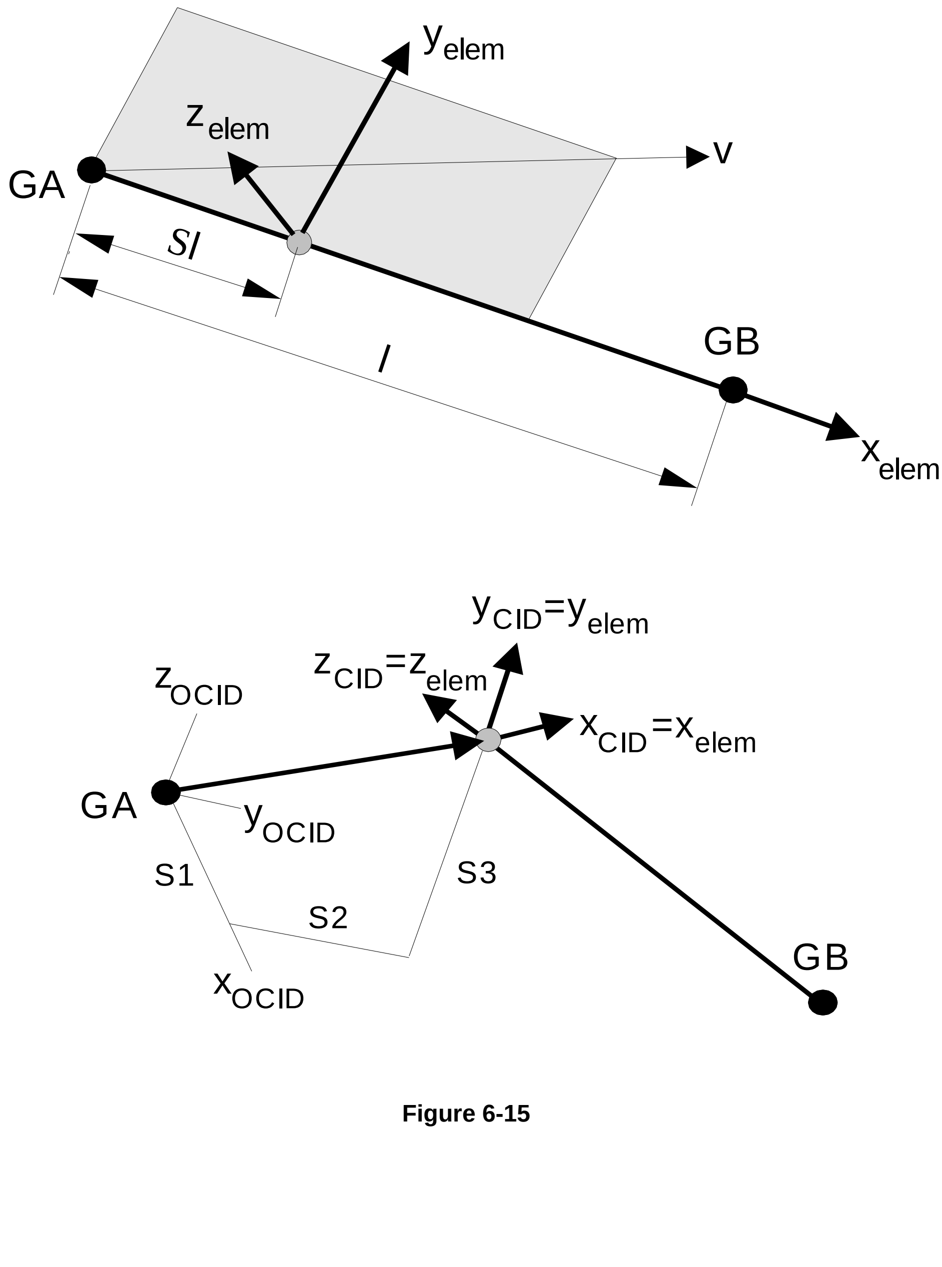


Figure 6-15

6.7.17 CDAMP1

Data Entry: **CDAMP1** - Scalar Viscous Damper Connection.

Description: Define a scalar damper element of the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP1	EID	PID	G1	C1	G2	C2			

Example:

1	2	3	4	5	6	7	8	9	10
CDAMP1	19	6	8	1	23	2			

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PDAMP** property data (Default is EID) (Integer > 0 or blank).
4	G1	**GRID** or **SPOINT** identification number (Integer $\geq$ 0 or blank).
5	C1	Component number in the general coordinate system (1 $\leq$ Integer $\leq$ 6 or blank).
6	G2	GRID or SPOINT identification number (Integer $\geq$ 0 or blank).
7	C2	Component number in the general coordinate system (1 $\leq$ Integer $\leq$ 6 or blank).

Remarks:

1. Scalar points may be used for G1 and/or G2 in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point whose displacement is constrained to zero.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.

6.7.18 CDAMP2

Data Entry: **CDAMP2** - Scalar Viscous Damper Connection.

Description: Define a scalar damper element of the structural model without referencing PDAMP.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP2	EID	B	G1	C1	G2	C2			

Example:

1	2	3	4	5	6	7	8	9	10
CDAMP2	19	16.0	8	1	23	2			

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	B	Scalar damper value (Real > 0.0)
4	G1	**GRID** or **SPOINT** identification number (Integer $\geq$ 0 or blank).
5	C1	Component number in the general coordinate system (1 $\leq$ Integer $\leq$ 6 or blank).
6	G2	GRID or SPOINT identification number (Integer $\geq$ 0 or blank).
7	C2	Component number in the general coordinate system (1 $\leq$ Integer $\leq$ 6 or blank).

6

Remarks:

1. Scalar points may be used for G1 and/or G2 in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point whose displacement is constrained to zero.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.

6.7.19 CDAMP3

Data Entry: **CDAMP3** - Scalar Viscous Damper Connection.

Description: Define a scalar damper element of the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP3	EID	PID	S1	S2					

Example:

1	2	3	4	5	6	7	8	9	10
CDAMP3	17	3	11	12					

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PDAMP** property data (Default is EID) (Integer > 0 or blank).
4	S1	**SPOINT** identification number (Integer $\geq$ 0 or blank).
5	S2	SPOINT identification number (Integer $\geq$ 0 or blank).

Remarks:

1. Zero or blank may be used to indicate a grounded terminal S1 or S2. A grounded terminal is a point whose displacement is constrained to zero.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. The two connection points S1and S2 must be distinct.

6.7.20 CDAMP4

Data Entry: **CDAMP4** - Scalar Viscous Damper Connection.

Description: Define a scalar damper element of the structural model without referencing PDAMP.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP4	EID	B	S1	S2					

Example:

1	2	3	4	5	6	7	8	9	10
CDAMP4	7	12.0	21	22					

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	B	Scalar damper value (Real > 0.0)
4	S1	**SPOINT** identification number (Integer $\geq$ 0 or blank).
5	S2	SPOINT identification number (Integer $\geq$ 0 or blank).

Remarks:

1. Zero or blank may be used to indicate a grounded terminal S1 or S2. A grounded terminal is a point whose displacement is constrained to zero.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. The two connection points S1 and S2 must be distinct.

6

6.7.21 CELAS1

Data Entry: **CELAS1 -** Scalar Spring Connection.

Description: Define a scalar spring element of the structural or thermal model.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
CELAS1	2	6	7	2	8	1			

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PELAS** property data for structural elements and **PELASH** property data for heat transfer elements (Default is EID) (Integer > 0).
4	G1	**GRID** or **SPOINT** identification number (Integer $\geq$ 0 or blank).
5	C1	Component number in the general coordinate system, blank for scalar points (1 $\leq$ Integer $\leq$ 6 or blank).
6	G2	GRID or SPOINT identification number (Integer $\geq$ 0 or blank).
7	C2	Component number in the general coordinate system, blank for scalar points (1 $\leq$ Integer $\leq$ 6 or blank). See Remark 6.

Remarks:

1. The two connection points (G1, C1) and (G2, C2) must be distinct.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. Either G1 or G2, but not both, may be equal to zero. In this case, the element is grounded and the component number is ignored.
4. The component numbers must be blank for heat transfer analysis.
5. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank.
6. A CELAS1 element may be used as a structural element or a heat transfer element, but not both. Structural elements reference PELAS data and heat transfer elements reference PELASH data.

6

6.7.22 CELAS2

Data Entry: **CELAS2** - Scalar Spring Connection.

Description: Define a scalar spring element of the structural model without referencing PELAS.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
CELAS2	25	10.0	14	3				1.0	

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	K	Elastic stiffness property (Real).
4	G1	**GRID** or **SPOINT** identification number (Integer $\geq$ 0 or blank).
5	C1	Component number in the general coordinate system, blank for scalar points (1 $\leq$ Integer $\leq$ 6 or blank).
6	G2	GRID or SPOINT identification number (Integer $\geq$ 0 or blank).
7	C2	Component number in the general coordinate system, blank for scalar points (1 $\leq$ Integer $\leq$ 6 or blank).
8	GE	Damping coefficient (Real or Blank).
9	SRC	Stress recovery coefficient. Stress = SRC*FORCE (Real or Blank).

Remarks:

1. The two connection points (G1, C1) and (G2, C2) must be distinct.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. Either G1 or G2, but not both, may be equal to zero. In this case, the element is grounded and the component number is ignored.
4. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank.
5. The user is cautioned to be careful using negative spring values.
6. For heat transfer analysis, use CELAS1 (or CELAS3) and PELASH.

6.7.23 CELAS3

Data Entry: **CELAS3** - Scalar Spring Connection.

Description: Define a scalar spring element of the structural or thermal model.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
CELAS3	14	2	23	24					

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PELAS** property data for structural elements and **PELASH** property data for heat transfer elements (Default is EID) (Integer > 0).
4	S1	**SPOINT** identification number (Integer $\geq$ 0 or blank).
5	S2	SPOINT identification number (Integer $\geq$ 0 or blank).

Remarks:

1. The two connection points S1 and S2 must be distinct.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. Either S1 or S2, but not both, may be equal to zero or blank. In this case, the element is grounded.

6

6.7.24 CELAS4

Data Entry: **CELAS4** - Scalar Spring Connection.

Description: Define a scalar spring element of the structural model without referencing PELAS.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
CELAS4	7	18.0	26	27					

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	K	Elastic stiffness property (Real).
4	S1	**SPOINT** identification number (Integer $\geq$ 0 or blank).
5	S2	SPOINT identification number (Integer $\geq$ 0 or blank).

Remarks:

1. The two connection points S1 and S2 must be distinct.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. Either S1 or S2, but not both, may be equal to zero or blank. In this case, the element is grounded.

6.7.25 CGAP

Data Entry: **CGAP** - Gap Element Definition.

Description: Defines a gap element of the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
CGAP	EID	PID	GA	GB	X1	X2	X3		

Alternate Format 1:

1	2	3	4	5	6	7	8	9	10
CGAP	EID	PID	GA	GB	GO				

Alternate Format 2:

1	2	3	4	5	6	7	8	9	10
CGAP	EID	PID	GA	GB				CID	

Examples:

1	2	3	4	5	6	7	8	9	10
CGAP	101	3001	206	406	10.5	300.25	50.3		

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of **PGAP** property data (default is EID), (Integer >0 or blank).
4	GA	**GRID** identification number of first connection point (Integer > 0).
5	GB	**GRID** identification numbers of second connection point (Integer > 0).
6-8	X1,X2,X3	Components of orientation vector v, at end A measured at end A using the general coordinate system for GA (Real or blank). Only used if CID = blank.
6	GO	Grid point identification number to optionally supply X1, X2, X3 (Integer > 0 or blank). Direction of orientation vector is GA to GO. Only used if CID = blank.
9	CID	Coordinate system identification number defining the element coordinate system. (Integer ≥ 0 or blank). 0 indicates the basic coordinate system. Blank indicates to use Xi or GO along with the end locations to define the element coordinate system.

6

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. If GA and GB are coincident, CID must be specified.
3. If CID is not blank, GO/X1/X2/X3 are ignored.
4. If CID identifies a cylindrical or spherical coordinate system, then the coordinates of GA are used to locate the system.
5. This element is a spring element with two possible stiffnesses, a “closed” stiffness (KA), and an “open” stiffness (KB). If nonlinearity is activated in a static loadcase with the **NLPARM** solution control command, then the spring stiffness is calculated from the displacements. See **Nonlinear Contact Element (CGAP)** (Sec. 2.4.11) for more details. In non-static loadcases, or in static loadcases without NLPARM, the stiffness is determined only by U0 from the referenced PGAP.

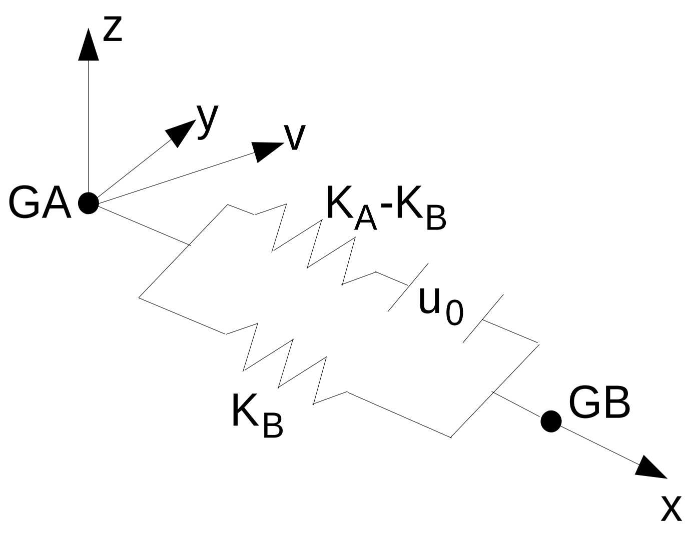


Figure 6-16

6.7.26 CGLUE

Data Entry: **CGLUE** - Glue Connection Definition.

Description: Defines a glue connection between a pair of surfaces.

Format:

1	2	3	4	5	6	7	8	9	10
CGLUE	GID	SSID	MSID	SRCHDIS		PNLTY	SHEAR		
+	“CYCLIC”	CID	AXIS	ANGLE					

Examples:

1	2	3	4	5	6	7	8	9	10
CGLUE	100	3001	4001						

1	2	3	4	5	6	7	8	9	10
CGLUE	600	8001	1001						
+	CYCLIC	9	Z	72.0					

Field	Information	Description
2	GID	Unique glue connection identification number (Integer > 0).
3,4	SSID,MSID	Surface identification numbers defined by **BSURFE**, **BSURFM** or **BSURFP** (Integer > 0).
5	SRCHDIS	Search distance criteria for creating glue points. (Real ≥ 0.0 or blank. Default = 0.0). See remark 4.
7	PNLTY	Penalty adjustment coefficient. (Real > 0.0 or blank. Default = 1.0). See remark 7.
8	SHEAR	Shear behavior of the connection. (STICK, SLIP or blank. Default = STICK). See remark 5.
2	“CYCLIC”	Keyword indicating that the continuation line contains data for cyclic periodicity. See remark 6.
3	CID	Coordinate system identification number defining the rotation for cyclic periodicity (Integer ≥ 0). 0 indicates the basic coordinate system.
4	AXIS	Rotation axis for cyclic periodicity (One of the characters X, Y or Z)
5	ANGLE	Rotation angle for cyclic periodicity in degrees. (Real)

Remarks:

1. SSID and MSID must be unique.

2. The intersection of all element faces belonging to surface SSID and all element faces belonging to surface MSID surfaces must be null. This is checked during pre-processing and an error is reported if common element faces are present.
3. The direction of relative displacement is determined by the normal directions of the element faces in surfaces SSID and MSID. In order for glue points to be created, the normals of faces in each surface should be pointed toward the opposite surface. For surfaces created by **CQUAD4**/**CQUAD8**/**CTRIA3**/**CTRIA6** elements, both the element normal and its opposite may be considered if SRCHDIS > 0.0.
4. Glue points are created at faces where the normal distance to the closest opposite face is less than SRCHDIS. If SRCHDIS is 0.0, then any distance is acceptable. When calculating the normal distance to a surface face created by a CQUAD4/CQUAD8/CTRIA3/CTRIA6 element, if PARAM,**GLSRCH** is set to 1, the default, the face is located at either the top or bottom surface of the element, depending on which side has the appropriately opposite pointing normal. If PARAM,**GLSRCH** is set to 0, the face is located at the reference plane of the element, and connections are only created if the normal points in the opposite direction of the other surface.
5. By default, the glue connection will bond each glue point in all three translation degrees of freedom. If SHEAR (field 8) is "SLIP", then the glue connection will only bond the direction normal to the faces at each glue point. The surfaces would be free to slide relative to each other in the in-plane directions.
6. The "CYCLIC" continuation data can optionally be used to connect the two radial faces of a wedge-shaped model in order to simulate an entire cyclic structure. If present, the faces of surface SSID will be rotated by ANGLE about the axis identified by AXIS of coordinate system CID for the purposes of determining the glue points connecting the surface degrees-of-freedom. The rotated surface SSID should be approximately coincident with surface MSID.
7. The default penalty stiffness is determined from the normal stiffness of surrounding elements at each glue point. This automatic penalty is multiplied by the value of PNLTY to set the actual penalty stiffness. The default should be adequate for most problems. If the loading is unusually large, excessive penetrations/separations may exist in the final solution. In this case, it will be necessary to set PNLTY greater than 1.0.
8. Glue connections defined by CGLUE act as perfect conductors in heat transfer analysis such that there is no difference between the temperatures of the two surfaces at each glue point.

6.7.27 CGLUE1

Data Entry: **CGLUE1 -** Glue Connection Definition.

Description: Defines a glue connection between a surface and a collection of points.

Format:

1	2	3	4	5	6	7	8	9	10
CGLUE1	GID	SPID	MSID	SRCHDIS			CM		

Example:

1	2	3	4	5	6	7	8	9	10
CGLUE1	100	3001	4001						

Field	Information	Description
2	GID	Unique glue connection identification number (Integer > 0).
3	SPID	Point collection identification number defined by **BPOINTG** (Integer > 0).
4	MSID	Surface identification number defined by **BSURFE**, **BSURFM** or **BSURFP** (Integer > 0).
5	SRCHDIS	Search distance criteria for creating connections. (Real ≥ 0.0 or blank. Default = 0.0). See remark 1.
8	CM	Components of grids in the point collection to be made dependent. Any of the digits 1, 2, ... 6 with no embedded blanks (Integer > 0 or blank. Default = 123). See remark 2.

Remarks:

1. A grid in the point collection is connected to the surface when the normal distance to the closest opposite face is less than SRCHDIS. If SRCHDIS is 0.0, then any distance is acceptable. If the normal projection of a grid to the closest opposite face falls far outside that face, that grid is not connected.
2. CGLUE1 internally generates RBE3 elements to connect grids in the point collection to the surface faces. Components CM of each grid in the point collection that is connected is set to be dependent on the generated data.

6.7.28 CHBDY

Data Entry: **CHBDY -** Heat Boundary Element Connection.

Description: Defines a boundary element for heat transfer analysis which is used for heat flux, thermal vector flux and/or convection.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDY	EID	PID	TYPE	G1	G2	G3	G4		
+	GA1	GA2	GA3	GA4	V1	V2	V3		

Example:

1	2	3	4	5	6	7	8	9	10
CHBDY	71	4	LINE	4	5				
+	9	10			1.0	0.0	0.0		

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PHBDY** entry (Integer > 0).
4	TYPE	Type of area involved (must be one of “POINT”, “LINE”, “AREA3”, “AREA4” or “ELCYL”).
5-8	G1,...,G4	**GRID** identification numbers or connection points (Integer > 0 or Blank).
2-5	GA1,..., GA4	GRID or **SPOINT** identification numbers of associated ambient points (Integer > 0 or Blank).
6-8	V1, V2, V3	Vector (in the basic coordinate system) used for element orientation (Real or Blank).

6

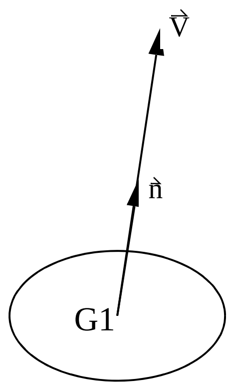
Remarks:

1. The continuation data is not required.
2. The five types have the following characteristics;
   a. The “POINT” type has one primary grid point, requires a property statement, and the normal vector V1,V2,V3 must be given if thermal vector flux is to be used.
   b. The “LINE” type has two primary grid points, requires a property statement, and the vector V1, V2, V3 is required if thermal vector flux is to be used.

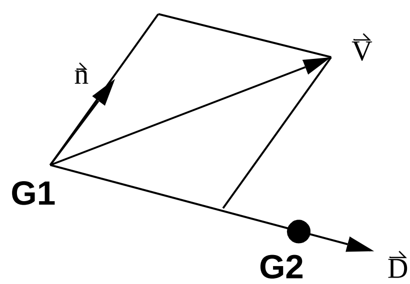
c. The “AREA3” and “AREA4” types have three and four primary grid points, respectively. These points define a triangular or quadrilateral surface and must be ordered to go around the boundary. A property statement is required for convection or thermal vector flux. The AREA4 element can be warped.

d. The “ELCYCL” type (elliptic cylinder) has two connected primary grid points. It requires a property statement and if thermal vector flux is used, the vector must be nonzero.

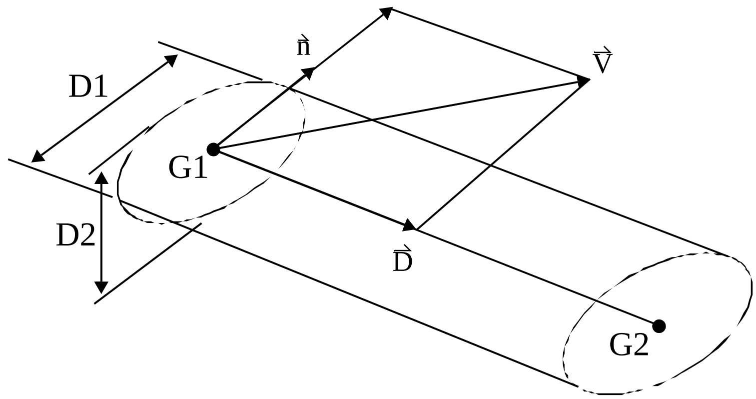
3. A property statement, PHBDY, is used to define the associated area factors, the absorptivity and the principal radii of the elliptic cylinder. The material coefficients used for convection are referenced by the PHBDY entry. See this entry description for details.
4. The associated points, GA1, GA2, etc., are used to define the ambient temperature for a convection field, and may be either grid or scalar points. These points correspond to the primary points G1, G2, etc., and the number of these depends on the TYPE option, but they need not be unique. Their values must be set in heat transfer with an SPCD statement and they may be connected to other elements. If any field is blank, the ambient temperature associated with that grid point is assumed to be zero.
5. Heat flux may be applied to this element with QBDY1 or QBDY2 data.
6. Thermal vector flux from a directional source may be applied to this element with a QVECT data statement.



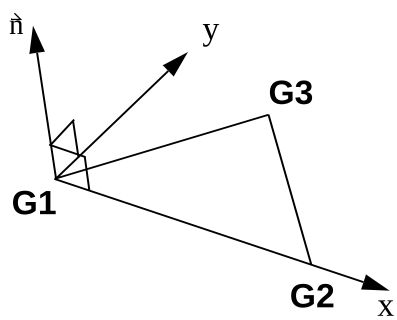
POINT



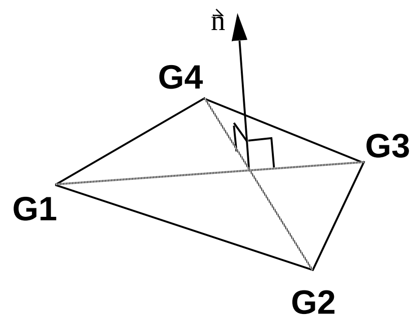
LINE



ELCYL



AREA3



AREA4

6

6.7.29 CHBDYE

Data Entry: **CHBDYE** - Heat Boundary Element Connection.

Description: Defines a boundary element for heat transfer analysis by referring to a heat conduction element.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYE	EID	EID2	SIDE						

Example:

1	2	3	4	5	6	7	8	9	10
CHBDYE	15	102	3						

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	EID2	Element identification number of a heat conduction element (Integer > 0). See Remark 2.
4	SIDE	Side identification number of element EID2. See Remark 3.

Remarks:

1. Heat boundary element connection CHBDYE is defined by referring to a heat conduction element. EID is the element identification number of the heat boundary element connection; EID2 is the element identification number of a heat conduction element.
2. EID2 could refer to a line element **CROD**, **CBAR**, **CBEAM** and **CTUBE**, a plane element **CQUAD4**, **CQUAD8**, **CTRIA3** and **CTRIA6**, or a solid element **CHEXA**, **CPENTA** and **CTETRA**.
3. The side identification number SIDE identifies which grids of the heat conduction element are used to define the CHBDYE heat boundary element.

6

a. For EID2 referring to a line element CROD, CBAR, CBEAM or CTUBE, the side identification number SIDE could be 1, 2 or 3. SIDE = 1 means the type of the CHBDYE element is LINE, the element has two primary grid points, with GA as the first point and GB as the second point; SIDE = 2 means the type of the CHBDYE element is POINT, and the element has one primary grid point GA; SIDE = 3 means the type of the CHBDYE element is POINT, and the element has one primary grid point GB. GA and GB are the leading and ending grids defined in the line element.

EID2 ELEMENT	SIDE	CHDBYE TYPE	GRID POINTS (IN ORDER)
CROD / CBAR / CBEAM / CTUBE	1	LINE	GA, GB
CROD / CBAR / CBEAM / CTUBE	2	POINT	GA
CROD / CBAR / CBEAM / CTUBE	3	POINT	GB

b. For EID2 referring to a plane element CQUAD4, CQUAD8, CTRIA3 or CTRIA6, the side identification number SIDE could be 1,...,5. The type of the CHBDYE element and the grid points in CHBDYE element for each SIDE number are tabulated below. G1,...,G4 are grids defined in the plane element and are following the order of the element.

EID2 ELEMENT	SIDE	CHDBYE TYPE	GRID POINTS (IN ORDER)
CQUAD4 / CQUAD8	1	AREA4	G1, G2, G3, G4
CQUAD4 / CQUAD8	2	LINE	G1, G2
CQUAD4 / CQUAD8	3	LINE	G2, G3
CQUAD4 / CQUAD8	4	LINE	G3, G4
CQUAD4 / CQUAD8	5	LINE	G4, G1

EID2 ELEMENT	SIDE	CHDBYE TYPE	GRID POINTS (IN ORDER)
CTRIA3 / CTRIA6	1	AREA3	G1, G2, G3
CTRIA3 / CTRIA6	2	LINE	G1, G2
CTRIA3 / CTRIA6	3	LINE	G2, G3
CTRIA3 / CTRIA6	4	LINE	G3, G1

c. For EID2 referring to a solid element CHEXA, CPENTA or CTETRA, the side identification number SIDE could be 1,...,6. The type of the CHBDYE element and the grid points involved in CHBDYE element for each SIDE number is tabulated below. G1,...,G8 are grids which are defined in the solid element and are following the order of the element.

EID2 ELEMENT	SIDE	CHDBYE TYPE	GRID POINTS (IN ORDER)
CHEXA	1	AREA4	G4, G3, G2, G1
CHEXA	2	AREA4	G1, G2, G6, G5
CHEXA	3	AREA4	G2, G3, G7, G6
CHEXA	4	AREA4	G3, G4, G8, G7
CHEXA	5	AREA4	G4, G1, G5, G8
CHEXA	6	AREA4	G5, G6, G7, G8

EID2 ELEMENT	SIDE	CHDBYE TYPE	GRID POINTS (IN ORDER)
CPENTA	1	AREA3	G3, G2, G1
CPENTA	2	AREA4	G1, G2, G5, G4
CPENTA	3	AREA4	G2, G3, G6, G5
CPENTA	4	AREA4	G3, G1, G4, G6
CPENTA	5	AREA3	G4, G5, G6

EID2 ELEMENT	SIDE	CHDBYE TYPE	GRID POINTS (IN ORDER)
CTETRA	1	AREA3	G3, G2, G1
CTETRA	2	AREA3	G1, G2, G4
CTETRA	3	AREA3	G2, G3, G4
CTETRA	4	AREA3	G3, G1, G4

4. CHBDYE generates equivalent **CHBDY** data. The sorted echo will show the data for the generated CHBDY.
5. Heat flux may be applied to this element with QBDY1 or QBDY2 data.
6. Thermal vector flux from a directional source may be applied to this element with a QVECT data statement.

6.7.30 CHBDYG

Data Entry: **CHBDYG** - Heat Boundary Element Connection.

Description: Defines a boundary element for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYG	EID		TYPE						
+	G1	G2	G3	G4					

Example:

1	2	3	4	5	6	7	8	9	10
CHBDYG	3		AREA3						
+	10	12	11						

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
4	TYPE	Type of area involved (must be one of “AREA3”, “AREA4”, “AREA6” or “AREA8”).
2-5	G1,...,G4	**GRID** identification numbers or connection points (Integer > 0 or Blank).

Remarks:

1. The types have the following characteristics;
   a. The “AREA3” and “AREA4” types have three and four primary grid points, respectively. These points define a triangular or quadrilateral surface and must be ordered to go around the boundary.
   b. The “AREA6” and “AREA8” types are treated as “AREA3” and “AREA4”, respectively. They have three and four primary grid points. The corner points are considered and the midpoints are ignored.
2. The default value for TYPE is specified in the BDYOR data entry.
3. CHBDYG generates equivalent **CHBDY** data. The sorted echo will show the data for the generated CHBDY.
4. Heat flux may be applied to this element with QBDY1 or QBDY2 data.
5. Thermal vector flux from a directional source may be applied to this element with a QVECT data statement.

6.7.31 CHBDYP

Data Entry: **CHBDYP** - Heat Boundary Element Connection.

Description: Defines a boundary element for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYP	EID	PID	TYPE			G1	G2		
+					V1	V2	V3		

Example:

1	2	3	4	5	6	7	8	9	10
CHBDYP	4	7	LINE			11	12		
+					1.0	0.0	0.0		

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PHBDY** entry (Integer > 0).
4	TYPE	Type of area involved (must be one of “POINT”, “LINE”, “ELCYL” or “TUBE”). See Remark 2.
7-8	G1, G2	**GRID** identification numbers or connection points (Integer > 0 or Blank).
6-8	V1, V2, V3	Vector (in the basic coordinate system) used for element orientation (Real or Blank).

6

Remarks:

1. The continuation data is not required.
2. The four types have the following characteristics;
   a. The “POINT” type has one primary grid point, requires a property statement, and the normal vector V1,V2,V3 must be given if thermal vector flux is to be used.
   b. The “LINE”, “ELCYCL” and “TUBE” type have two primary grid points, requires a property statement, and the vector V1, V2, V3 is required if thermal vector flux is to be used.
3. A property statement, PHBDY, is used to define the associated area factor and the principal radii of the elliptic cylinder. PHBDY has a special format for CHBDYP. See this entry description for details.
4. The default values for PID, TYPE, V1, V2 and V3 are specified in the BDYOR entry.

5. CHBDYP generates equivalent **CHBDY** data. The sorted echo will show the data for the generated CHBDY.
6. Heat flux may be applied to this element with QBDY1 or QBDY2 data.
7. Thermal vector flux from a directional source may be applied to this element with a QVECT data statement

6.7.32 CHEX20

Data Entry: **CHEX20 -** Connections of Solid Element with up to 21 Grid Points.

Description: Defines the connections of the HEX20 solid element.

Format:

1	2	3	4	5	6	7	8	9	10
CHEX20	EID	PID	G1	G2	G3	G4	G5	G6	
+	G7	G8	G9	G10	G11	G12	G13	G14	
+	G15	G16	G17	G18	G19	G20	G21		

Example:

1	2	3	4	5	6	7	8	9	10
CHEX20	71	4	3	4	5	6	7	8	
+	9	10		0	30	32	51	52	
+	53	54	61	62	63	65	66		

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PSOLID** property data (Integer > 0 or blank. Default = EID).
4-9	G1,...,G6	**GRID**identification numbers of connection points (Integer > 0).
2,3	G7, G8	GRID identification numbers of connection points (Integer > 0).
4-9	G9-G14	GRID identification numbers of connection points (Integer > 0 or blank).
2-8	G15-G21	GRID identification numbers of connection points (Integer $\geq$ 0 or blank).

Remarks:

1. **CHEXA** has the same capabilities, and is preferred over CHEX20.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. Grid points G1- G4 must be given in consecutive order about one face. G5-G8 are on the opposite face with G5 opposite G1, G6 opposite G2, etc.

6

4. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element in all cases.
5. A face of the HEX20 element can be loaded using **PLOAD4** data.
6. Components of stress, requested by the solution control command STRESS, are output in the material coordinate system defined on the PSOLID data.
7. Components of stress at grids of solid elements, requested by GSTRESS in the solution control are printed in the basic coordinate system.
8. If the referenced PSOLID has "PFLUID" in the FCTN field, then this CHEX20 defines a fluid element for a coupled fluid-structure problem. In this case, all referenced grid points must be fluid grids (have -1 in the CD field). Fluid elements have no stress or strain output, and may not be referenced by PLOAD4 data.

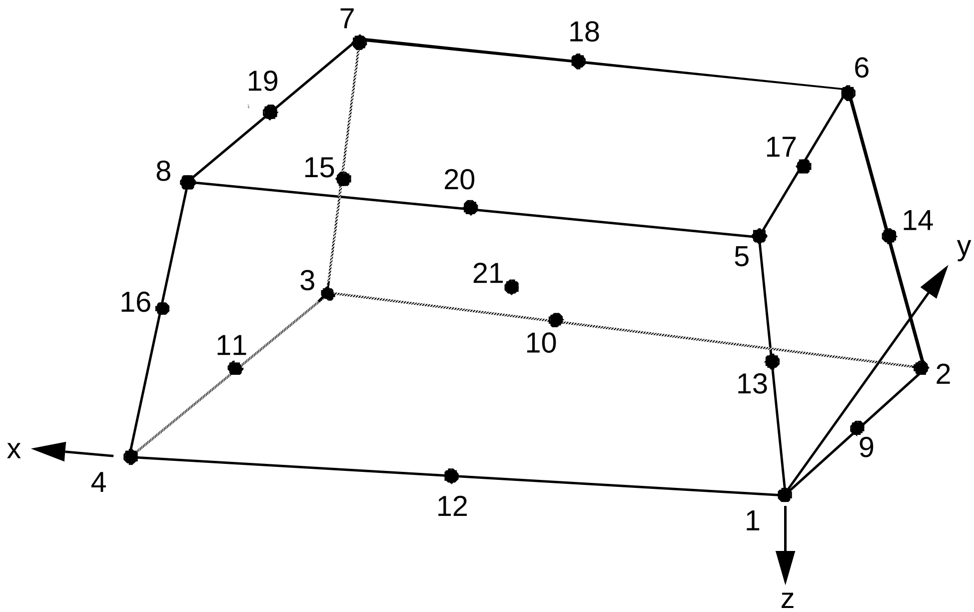


Figure 6-17

6

6.7.33 CHEXA

Data Entry: **CHEXA** - Six-sided Solid Element Connection.

Description: Defines the connections of the hexahedron solid or fluid 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	G21		

Example:

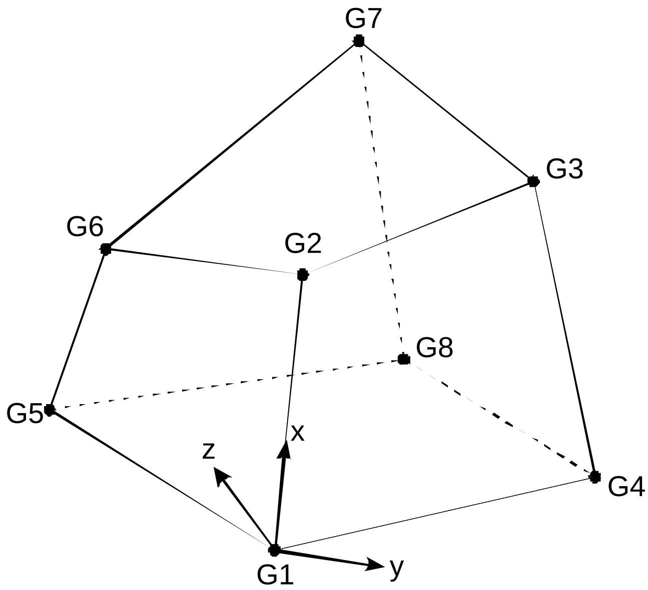
1	2	3	4	5	6	7	8	9	10
CHEXA	71	4	3	4	5	6	7	8	
+	9	10							

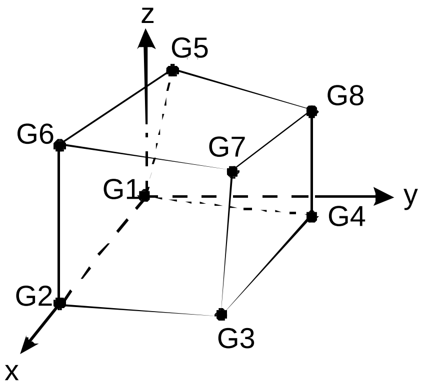
Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PSOLID** property data (Integer > 0 or blank. Default = EID).
4-9	G1,...,G6	**GRID** identification numbers of connection points (Integer > 0).
2,3	G7, G8	GRID identification numbers of connection points (Integer > 0).
4-9	G9,...,G14	GRID identification numbers of connection points (Integer > 0 or blank).
2-8	G15,...,G21	GRID identification numbers of connection points (Integer > 0 or blank).

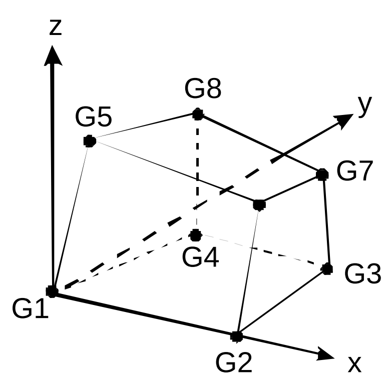
Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Grid points G1, ..., G4 must be given in consecutive order about one quadrilateral face. G5, ..., G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. The element coordinate system for the 8 noded HEXA element is defined as follows: The line from G1 to G2 defines the X axis. G4 is then used to define the X-Y plane. The Y axis is perpendicular to the X axis and lies in the X-Y plane. The Z axis is perpendicular to the X and Y axes.
4. A face of the HEXA element can be loaded using **PLOAD4** data.

5. Components of stress or strain, requested by STRESS or STRAIN in the solution control section, are output in the material coordinate system. The material coordinate system is defined on the PSOLID data.
6. Components of stress at grids of solid elements, requested by GSTRESS in the solution control, are printed in the basic coordinate system.
7. If the referenced PSOLID has “PFLUID” in the FCTN field, then this CHEXA defines a fluid element for a coupled fluid-structure problem. In this case, all referenced grid points must be fluid grids (have -1 in the CD field). Fluid elements have no stress or strain output, and may not be referenced by PLOAD4 data.







8. When any of the G9 through G21 are used, *GENESIS* will use the following connection of grids and coordinate system: For this case, *GENESIS* internally uses the HEX20 element.

Figure 6-18

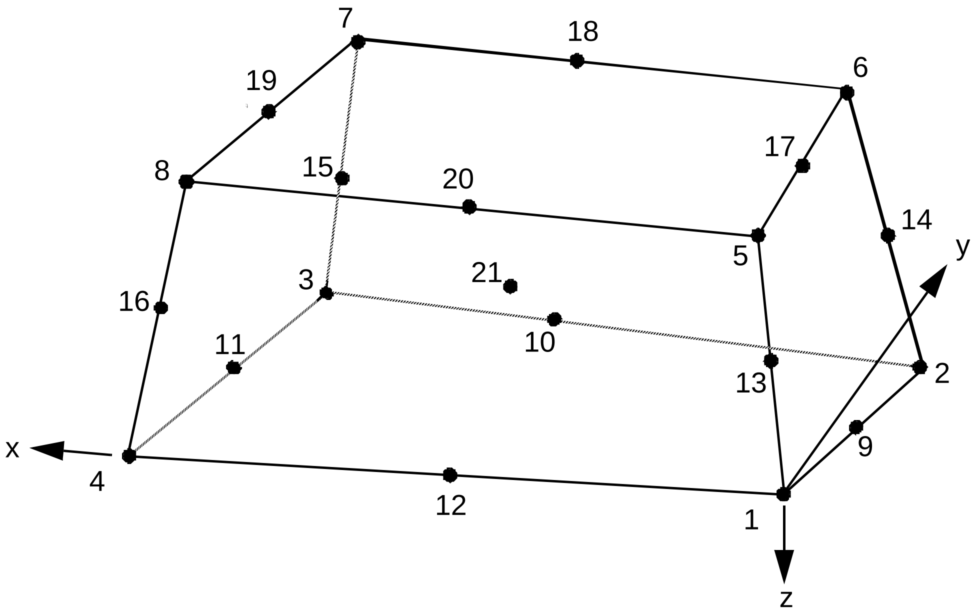


Figure 6-19

6.7.34 CMASS1

Data Entry: **CMASS1** - Scalar Mass Connection

Description: Defines a scalar mass element of the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS1	EID	PID	G1	C1	G2	C2			

Example:

1	2	3	4	5	6	7	8	9	10
CMASS1	32	6	2	1	23	2			

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PMASS** property entry (Default is EID) (Integer > 0).
4, 6	G1, G2	**GRID** or **SPOINT** identification number (Integer $\geq$ 0 or blank).
5, 7	C1, C2	Component number (0 $\leq$ Integer $\leq$ 6 or blank).

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point whose displacement is constrained to zero.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and G2, C2) must be distinct.

6

6.7.35 CMASS2

Data Entry: **CMASS2** - Scalar Mass Connection

Description: Defines a scalar mass element without referencing a PMASS.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS2	EID	M	G1	C1	G2	C2			

Example:

1	2	3	4	5	6	7	8	9	10
CMASS2	10	11.00	1001	1	1002	1			

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	M	Scalar mass value (Real > 0.0).
4, 6	G1, G2	**GRID** or **SPOINT** identification number (Integer ≥ 0 or blank).
5, 7	C1, C2	Component number (0 ≤ Integer ≤ 6 or blank).

Remarks:

1. Scalar points may be used for G1 and/or G2 in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point whose displacement is constrained to zero.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.

6

6.7.36 CMASS3

Data Entry: **CMASS3** - Scalar Mass Connection

Description: Defines a scalar mass element of the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS3	EID	PID	S1	S2					

Example:

1	2	3	4	5	6	7	8	9	10
CMASS3	7	3	2	1					

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PMASS** property entry (Default is EID) (Integer > 0).
4, 6	S1, S2	**SPOINT** identification number (Integer $\geq$ 0 or blank).

Remarks:

1. Zero or blank may be used to indicate a grounded terminal S1 or S2. A grounded terminal is a point whose displacement is constrained to zero.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. The two connection points S1 and S2 must be distinct.

6.7.37 CMASS4

Data Entry: **CMASS4 -** Scalar Mass Connection

Description: Defines a scalar mass element without referencing a PMASS.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS4	EID	M	S1	S2					

Example:

1	2	3	4	5	6	7	8	9	10
CMASS4	18	15.00	101	102					

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	M	Scalar mass value (Real > 0.0).
4, 6	S1, S2	**SPOINT** identification number (Integer ≥ 0 or blank).

Remarks:

1. Zero or blank may be used to indicate a grounded terminal S1 or S2. A grounded terminal is a point whose displacement is constrained to zero.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. The two connection points S1 and S2 must be distinct.

6

6.7.38 CONM2

Data Entry: **CONM2 -** Concentrated Mass Element Connection.

Description: Defines a concentrated mass at a grid point of the structural model.

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:

1	2	3	4	5	6	7	8	9	10
CONM2	4	4		1000.0	0.0	0.0	0.0		
+	0.0	0.0	0.0	0.0	0.0	0.0			

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	G	**GRID** identification number (Integer > 0).
4	CID	Coordinate system identification number (Integer $\geq$ 0 or blank or -1). A value of 0 or blank implies the basic coordinate system). See remark 4.
5	M	Mass value (Real $\geq$ 0.0).
6-8	X1,X2,X3	Offset distances from the grid point to the center of gravity of the mass in the coordinate system defined in field 4, unless CID = -1, in which case X1, X2, X3 are the coordinates, not offsets, of the center of gravity of the mass in the basic coordinate system (Real or blank. Default=0.0).
2-7	Iij	Mass moments of inertia measured at the mass center of gravity in the coordinate system defined by field 4 (The I11, I22 and I33 terms can be Real $\geq$ 0.0. The rest may be real or blank). If CID = -1, the basic coordinate system is used.

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The CONM2 information cannot be updated in the structural optimization because it has no property data. To use masses as design elements, use the CONM3 data.
3. The continuation may be omitted. This implies zero values for the mass moments of inertia.

4. If CID = -1 in field 4, offsets are internally computed as the difference between the grid point location and X1, X2, X3. In this case, the values of Iij are in a coordinate system that parallels the basic coordinate system. The grid point location may be defined in a nonbasic coordinate system.
5. The form of the inertia matrix about its center of gravity is taken as:

$$\begin{bmatrix} M & & & & & \\ & M & & & \text{SYM} & \\ & & M & & & \\ & & & I11 & & \\ & & & -I21 & I22 & \\ & & & -I31 & -I32 & I33 \end{bmatrix}$$

where:

$$M = \int \rho dV$$

$$I11 = \int \rho (x_2^2 + x_3^2) dV$$

$$I22 = \int \rho (x_1^2 + x_3^2) dV$$

$$I33 = \int \rho (x_1^2 + x_2^2) dV$$

$$I21 = \int \rho x_1 x_2 dV$$

$$I31 = \int \rho x_1 x_3 dV$$

$$I32 = \int \rho x_2 x_3 dV$$

and x_1, x_2, x_3 are components of distance from a point in the mass to the center of gravity of the mass. These coordinates are measured in the coordinate system defined in field 4. The negative signs for the off-diagonal terms are supplied by the program.

6. If lumped mass is used (PARAM, COUPMASS, NO) then the mass offset and I_{21}, I_{31} and I_{32} terms will be ignored. If a mass offset is specified, a warning message will be issued.
7. CONM2 elements are ignored in heat transfer analysis.

6.7.39 CONM3

Data Entry: **CONM3 -** Concentrated Mass Element Connection, Separate Property Data Form.

Description: Defines a concentrated mass at a grid point of the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
CONM3	EID	PID	G						

Example:

1	2	3	4	5	6	7	8	9	10
CONM3	32	5	107						

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PCONM3** property entry (Integer > 0).
4	G	**GRID** identification number (Integer > 0).

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. CONM3 elements are ignored in heat transfer analysis.

6

6.7.40 CONV

Data Entry: **CONV -** Convection Boundary Condition.

Description: Defines a convection condition on a heat boundary element connection CHBDYE, CHBDYG or CHBDYP.

Format:

1	2	3	4	5	6	7	8	9	10
CONV	EID	PCONID			GA1	GA2	GA3	GA4	

Example:

1	2	3	4	5	6	7	8	9	10
CONV	7	2			10	11	12		

Field	Information	Description
2	EID	Element identification number of a CHBDYE, CHBDYG or CHBDYP element (Integer > 0).
3	PCONID	Identification number of a **PCONV** entry (Integer > 0).
6	GA1	**GRID** or **SPOINT** identification numbers of associated ambient points (Integer > 0). See Remark 3.
7-9	GA2,GA3,GA4	**GRID** or **SPOINT** identification numbers of associated ambient points (Integer > 0 or Blank, Default = GA1). See Remark 3.

Remarks:

1. CONV is used with **CHBDYE**, **CHBDYG** or **CHBDYP** elements.
2. A property statement, **PCONV**, is used to define the convective film coefficient. See this entry description for details.
3. The associated points, GA1, GA2, etc., are used to define the ambient temperature for a convection field, and may be either grid or scalar points. These points correspond to the primary points G1, G2, etc., and the number of these depends on the referring CHBDYE, CHBDYG or CHBDYP element, but they need not be unique. Their values must be set in heat transfer with an SPCD statement and they may be connected to other elements.

6.7.41 CORD1C

Data Entry: **CORD1C** - Cylindrical Coordinate System Definition.

Description: Defines a cylindrical coordinate system by reference to three points. These points must be defined in coordinate systems whose definition does not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin. The points are defined by GRID data statements.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1C	CID	G1	G2	G3	CID	G1	G2	G3	

Example:

1	2	3	4	5	6	7	8	9	10
CORD1C	2	15	21	17					

Field	Information	Description
2, 6	CID	Coordinate system identification number (Integer > 0).
3-5 7-9	G1,G2,G3	**GRID** identification number (Integer > 0; $G1 \neq G2 \neq G3$).

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C and CORD2S data must all be unique
2. The three points G1, G2, G3 must be noncollinear.
3. The location of a grid point (P in the sketch) in this coordinate system is given by (R, θ, Z) where θ is measured in degrees.
4. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_z).
5. Points on the z-axis may not have their displacement directions defined in this coordinate system since an ambiguity results.
6. One or two coordinate systems may be defined on a single data entry.
7. The coordinate system is <u>not</u> changed during shape optimization, even if G1, G2, G3 move.
8. The points that define the coordinate system do not have to be GRIDs connected to the structure.

6

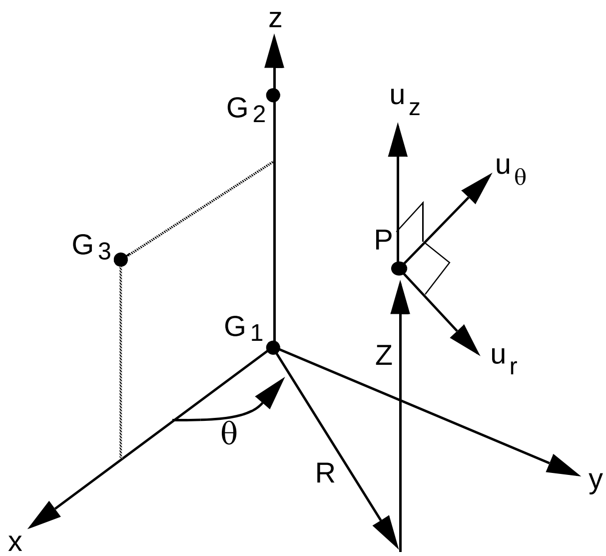


Figure 6-20

6.7.42 CORD1R

Data Entry: **CORD1R** - Rectangular Coordinate System Definition.

Description: Defines a rectangular coordinate system by reference to three points. These points must be defined in coordinate systems whose definition does not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the x-z plane. The points are defined by GRID data statements.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1R	CID	G1	G2	G3	CID	G1	G2	G3	

Example:

1	2	3	4	5	6	7	8	9	10
CORD1R	2	10	11	17	4	60	150	128	

Field	Information	Description
2,6 3-5	CID	Coordinate system identification number (Integer > 0).
7-9	G1,G2,G3	**GRID** identification number (Integer > 0; $G1 \neq G2 \neq G3$).

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C and CORD2S data must all be unique.
2. The three points G1, G2, G3 must be noncollinear.
3. The location a of grid point (P in the sketch) in this coordinate system is given by (X, Y, Z).
4. The displacement coordinate directions at P are shown above by (u_x, u_y, u_z).
5. One or two coordinate systems may be defined on a single data entry.
6. The coordinate system is not changed during shape optimization, even if G1, G2, G3 move.
7. The points that define the coordinate system do not have to be GRIDs connected to the structure.

6

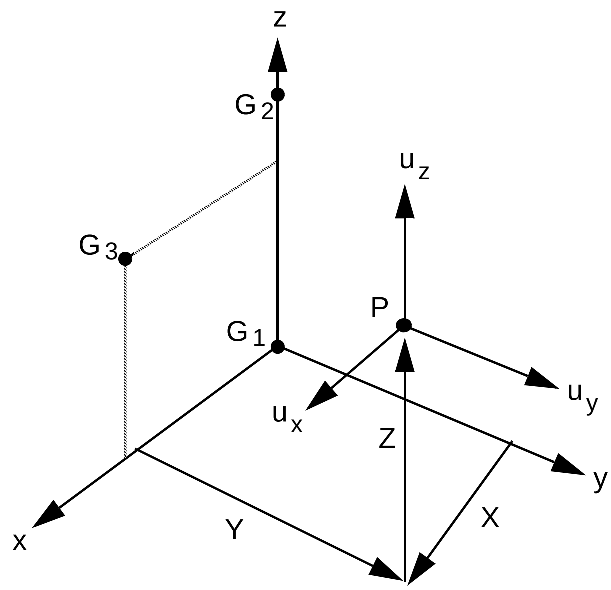


Figure 6-21

6.7.43 CORD1S

Data Entry: **CORD1S** - Spherical Coordinate System Definition.

Description: Defines a spherical coordinate system by reference to three points. These points must be defined in coordinate systems whose definition does not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin. The points are defined by GRID data statements.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1S	CID	G1	G2	G3	CID	G1	G2	G3	

Example:

1	2	3	4	5	6	7	8	9	10
CORD1S	2	1	10	20	3	200	210	150	

Field	Information	Description
2,6,..	CID	Coordinate system identification number (Integer > 0).
3-5,7-9	G1,G2,G3	**GRID** identification number (Integer > 0; $G1 \neq G2 \neq G3$).

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C and CORD2S data must all be unique.
2. The three points G1, G2, G3 must be noncollinear.
3. The location of a grid point (P in the sketch) in this coordinate system is given by (ρ, θ, ϕ) where θ and ϕ are measured in degrees.
4. The displacement coordinate directions at P are dependent on the location of P as shown above by $(u_\rho, u_\theta, u_\phi)$.
5. Points on the polar axis may not have their displacement directions defined in this coordinate system since an ambiguity results.
6. One or two coordinate systems may be defined on a single data entry.
7. The coordinate system is **not** changed during shape optimization, even if G1, G2, G3 move.
8. The points that define the coordinate system do not have to be GRIDs connected to the structure.

6

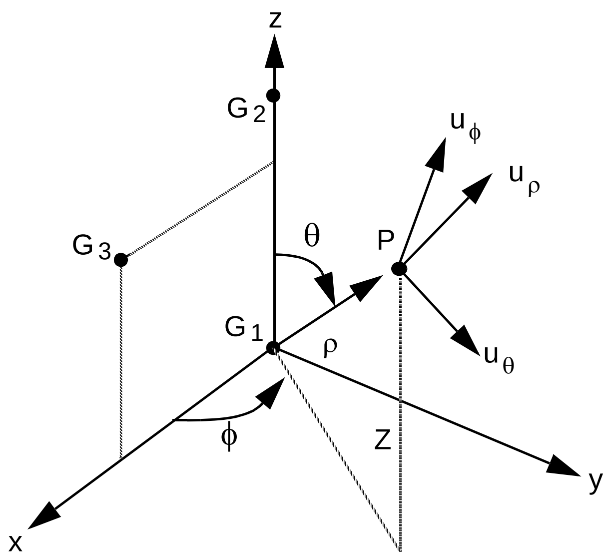


Figure 6-22

6.7.44 CORD2C

Data Entry: **CORD2C** - Cylindrical Coordinate System Definition.

Description: Defines a cylindrical coordinate system by reference to the coordinate of three points. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
CORD2C	3	15	-2.1	0.0	1.0	3.1	1.0	0.0	
+	4.2	2.0	-3.1						

Field	Information	Description
2	CID	Coordinate system identification number (Integer > 0).
3	RID	Reference to a coordinate system which is defined independently of new coordinate system (Integer $\geq$ 0 or blank).
4-6 7-9 2-4	A1,A2,A3 B1,B2,B3 C1,C2,C3	Coordinates of three points in coordinate system defined in field 3 (Real or blank. Default=0.0).

Remarks:

1. Continuation data must be present.
2. The three points (A1, A2, A3), (B1, B2, B3), (C1, C2, C3) must be unique and noncollinear.
3. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, and CORD2S data must all be unique.
4. An RID of zero references the basic coordinate system.
5. The location of a grid point (P in the sketch) in this coordinate system is given by (R, θ, Z) where θ is measured in degrees.
6. The displacement coordinate directions at P are dependent on the location of P as shown above by (u_r, u_θ, u_z).

Points on the z-axis may not have their displacement direction defined in this coordinate system since an ambiguity results.

6

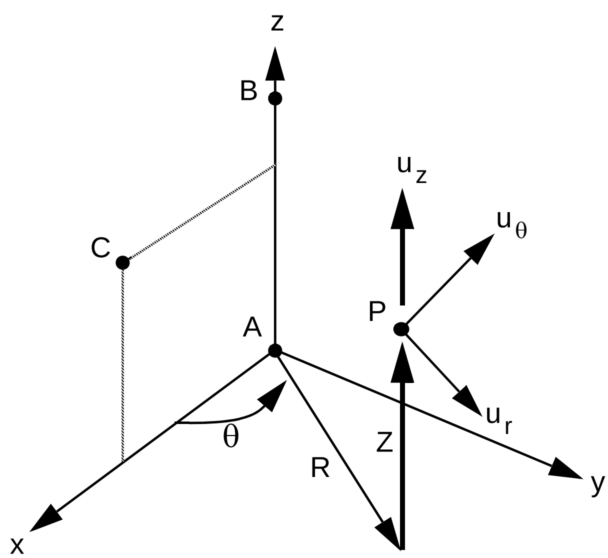


Figure 6-23

6.7.45 CORD2R

Data Entry: **CORD2R** - Rectangular Coordinate System Definition.

Description: Defines a rectangular coordinate system by reference to the coordinates of three points. The first point defines the origin. The second defines the direction of the z-axis,. The third point defines a vector which, with the z-axis, defines the x-z plane. The reference coordinate system must be independently defined.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
CORD2R	2	19	-1.3	1.0	0.0	3.1	0.0	1.0	
+	2.1	1.0	0.0						

Field	Information	Description
2	CID	Coordinate system identification number (Integer > 0).
3	RID	Reference to a coordinate system which is defined independently of new coordinate system (Integer $\geq$ 0 or blank).
4-6 7-9 2-4	A1,A2,A3 B1,B2,B3 C1,C2,C3	Coordinates of three points in coordinate system defined in field 3 (Real or blank. Default=0.0).

Remarks:

1. Continuation data must be present.
2. The three points (A1, A2, A3), (B1, B2, B3), (C1, C2, C3) must be unique and noncollinear.
3. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, and CORD2S data must all be unique.
4. An RID of zero references the basic coordinate system.
5. The location of a grid point (P in the sketch) in this coordinate system is given by (X, Y, Z).

6

The displacement coordinate directions at P are shown by (u_x, u_y, u_z).

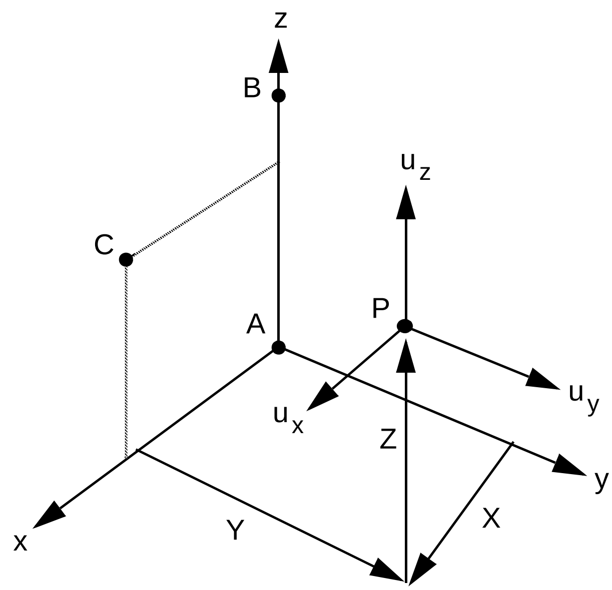


Figure 6-24

6.7.46 CORD2S

Data Entry: **CORD2S** - Spherical Coordinate System Definition.

Description: Defines a spherical coordinate system by reference to the coordinates of three points. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
CORD2S	3	14	-1.3	1.	0.0	3.8	0.0	1.0	
+	4.9	1.0	-2.7						

Field	Information	Description
2	CID	Coordinate system identification number (Integer > 0).
3	RID	Reference to a coordinate system which is defined independently of new coordinate system (Integer > 0 or blank).
4-6 7-9 2-4	A1,A2,A3 B1,B2,B3 C1,C2,C3	Coordinates of three points in coordinate system defined in field 3 (Real or blank. Default=0.0).

Remarks:

1. Continuation data must be present.
2. The three points (A1, A2, A3), (B1, B2, B3), (C1, C2, C3) must be unique noncollinear.
3. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C and CORD2S data must all be unique.
4. An RID of zero references the basic coordinate system.
5. The location of a grid point (P in the sketch) in this coordinate system is given by (ρ, θ, ϕ) where θ and ϕ are measured in degrees.
6. The displacement coordinate directions at P are shown above by (u_ρ, u_θ, u_ϕ).
7. Points on the polar axis may not have their displacement directions defined in this coordinate system since an ambiguity results.

6

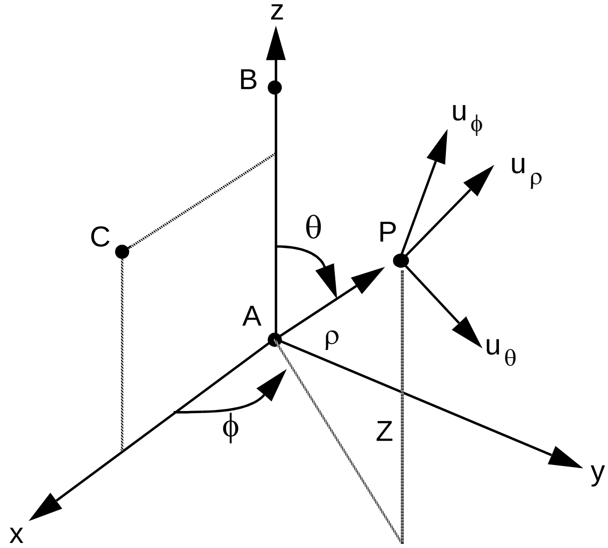


Figure 6-25

6.7.47 CPENTA

Data Entry: **CPENTA** - Five-sided Solid Element Connection with 6 or 15 nodes.

Description: Defines the connections of the CPENTA 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:

1	2	3	4	5	6	7	8	9	10
CPENTA	57	3	7	12	51	92	16	6	

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PSOLID** property data (Integer > 0 or blank. Default = EID).
4-9	G1 - G6	**GRID** identification numbers of connected points (Integer > 0).
2-9	G7 - G14	**GRID** identification numbers of connected points (Integer > 0 or blank).
2	G15	**GRID** identification numbers of connected points (Integer > 0 or blank).

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Grid points G7 through G15 are optional. If any grid points, G7 through G15 exist, all must be specified. The midside grids are recommended to be located within the middle third of edge.
3. The topology of **Figure 6-26** and **Figure 6-27** must be preserved, i.e., G1, G2, G3 define a triangular face. G4, G5 and G6 lie on the opposite face with G4 opposite G1, G5 opposite G2 and G6 opposite G3.
4. The element coordinate system of the 6 node linear element is defined with reference to **Figure 6-26** as follows. The origin is located at G1 and the x-axis lies on the G1-G2 edge. The y-axis lies in the G1-G2-G3 plane and is perpendicular to the x-axis, with the positive y-axis on the same side of the G1-G2 edge as node G3. The z-axis is orthogonal to the x and y axes.

5. The element coordinate system of the 15 node quadratic element is defined with reference to **Figure 6-27** as follows. The midplane is defined as the plane throught the three midpoints of straight line segments G1-G4, G2-G5 and G3-G6. The origin, O, is located at the midpoint of G1-G4. The z-axis points towards the top triangular face, and is oriented somewhere between a line joining the centroids of the bottom and top triangular faces and the normal to the midplane. The x and y-axes are perpendicular to the z-axis and point in directions toward, but not necessarily intersecting the edges G2-G11-G5 and G3-G12-G6.
6. A face of the PENTA element can be loaded using **PLOAD4** data.
7. Components of stress, requested by the solution control command STRESS, are output in the material coordinate system which is defined on the PSOLID data.
8. Components of grid stresses, requested by GSTRESS in the solution control, are printed in the basic coordinate system.
9. If the referenced PSOLID has “PFLUID” in the FCTN field, then this CPENTA defines a fluid element for a coupled fluid-structure problem. In this case, all referenced grid points must be fluid grids (have -1 in the CD field). Fluid elements have no stress or strain output, and may not be referenced by PLOAD4 data.

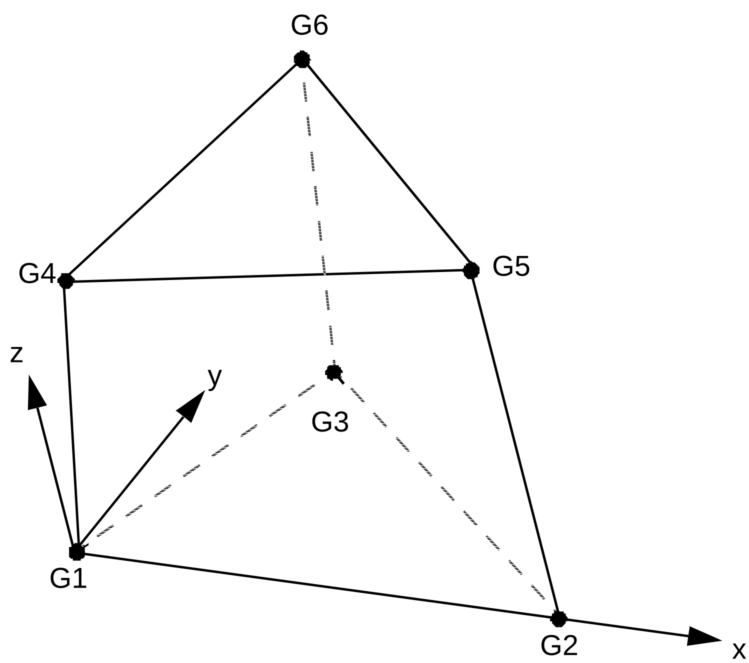


Figure 6-26

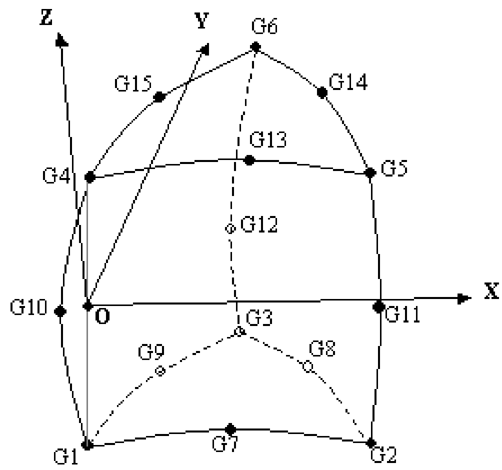


Figure 6-27

6.7.48 CPYRA

Data Entry: **CPYRA -** Five-sided Solid Element Connection with 5or 13 nodes.

Description: Defines the connections of the CPYRA element.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
CPYRA	57	3	7	12	51	92	16		

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PSOLID** property data (Integer > 0 or blank. Default = EID).
4-8	G1 - G5	**GRID** identification numbers of connected points (Integer > 0).
9,2-8	G6 - G13	**GRID** identification numbers of connected points (Integer > 0 or blank).

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Grid points G6 through G13 are optional. If any grid points, G6 through G13 exist, all must be specified. The midside grids are recommended to be located within the middle third of edge.
3. The topology of **Figure 6-28** and **Figure 6-29** must be preserved, i.e., G1, G2, G3, G4 define a quadrilateral face and G5 is the vertex of the pyramid.
4. The element coordinate system of the 5 node linear element is defined with reference to **Figure 6-28** as follows. The origin is located at G1 and the x-axis lies on the G1-G2 edge. The y-axis lies in the G1-G2-G3 plane and is perpendicular to the x-axis, with the positive y-axis on the same side of the G1-G2 edge as node G3. The z-axis is orthogonal to the x and y axes.
5. The element coordinate system of the 13 node quadratic element is defined with reference to **Figure 6-29** as follows. The origin is located at the center of the quadrilateral face. The z-axis is along a line connecting the center to grid G5. The y-axis is orthogonal to the z-axis and a line connecting the mid points of edges G1-G4 and G2-G3. The x-axis is orthogonal to the y- and z-axis.

6. A face of the PYRA element can be loaded using **PLOAD4** data.
7. Components of stress, requested by the solution control command STRESS, are output in the material coordinate system which is defined on the PSOLID data.
8. Components of grid stresses, requested by GSTRESS in the solution control, are printed in the basic coordinate system.
9. If the referenced PSOLID has “PFLUID” in the FCTN field, then this CPYRA defines a fluid element for a coupled fluid-structure problem. In this case, all referenced grid points must be fluid grids (have -1 in the CD field). Fluid elements have no stress or strain output, and may not be referenced by PLOAD4 data.

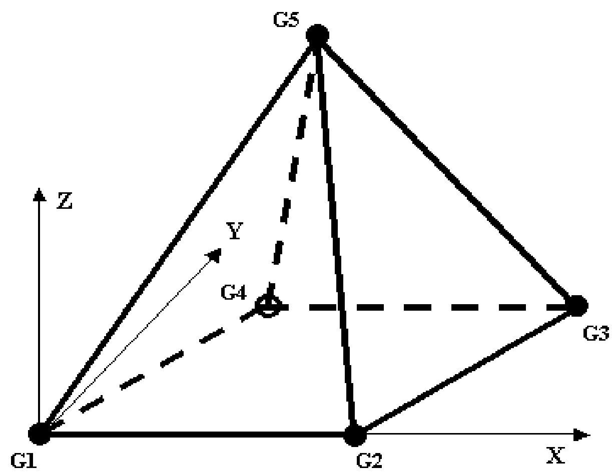


Figure 6-28

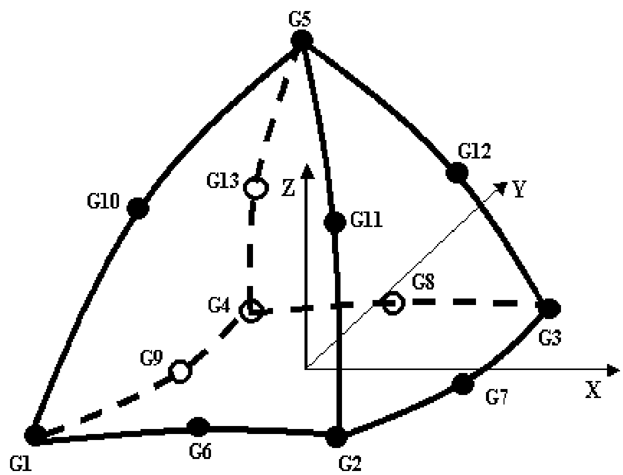


Figure 6-29

6.7.49 CQUAD4

Data Entry: **CQUAD4** - Quadrilateral Element Connection.

Description: Defines a quadrilateral plate/shell element (QUAD4) of the structural model. This is an isoparametric membrane-bending element of uniform thickness. Also defines a conductive element for heat transfer analysis. Alternatively, when used with PSKIN, CTRIA4 defines non-structural skin elements.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
CQUAD4	95	34	20	65	23	21	1.2	0.5	

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PSHELL**, **PCOMP**, **PCOMPG** or **PSKIN** property data (Integer > 0 or blank, default is EID).
4-7	G1-G4	**GRID** identification numbers of connection point (Integers > 0, all unique).
8	THETA	Material property orientation specification (Real or blank; or Integer $\geq$ 0). If Real, specifies the material property orientation angle in degrees. The sketch below gives the sign convention for THETA. If Integer, the orientation of the material x-axis is along the projection onto the plane of the element of the x-axis of the coordinate system specified by the integer value. See remark 8 & 9.
9	ZOFFS	Offset from the surface of grid points to the element reference plane (see Remark 4 & 8) (Real or blank. Default=0.0).

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All the interior angles must be less than 180 degrees.

6

4. Elements may be offset from the grid point surface by means of ZOFFS. Other data, such as material matrices and stress fiber location are given relative to the reference plane. Positive offset implies that the element reference plane lies above the grid points in the sketch. If the ZOFFS field is used, the MID2 field should be specified on the PSHELL entry referenced by the PID.
5. QUAD4 elements that reference PSHELL or PCOMP/PCOMPG can be loaded using **PLOAD2**, **PLOAD4** or **PLOAD5** data.
6. For QUAD4 elements that reference PSHELL data, forces, stresses and strains can be output in the element coordinate system, material coordinate system, basic or any defined coordinate system. The choice can be done in the PSHELL data statement (default is the element coordinate system).
7. For QUAD4 elements that reference PCOMP/PCOMPG data, stresses and strains are printed, if requested, in the layer coordinate system at the middle of the layer (half of its thickness). Forces are printed in the element coordinate system at the reference plane.
8. QUAD4 elements that reference **PSKIN** data are known as skin elements. Skin elements do not contribute stiffness, mass, conduction, load or anything to the structural or heat transfer problems. THETA or ZOFFS are ignored for skin elements. Skin elements can be used with the design data DTGRID to define a topography region.
9. The analysis parameter **THETA** provides a default value for THETA. See page 565.

This is a plate element with inplane rotational stiffness.

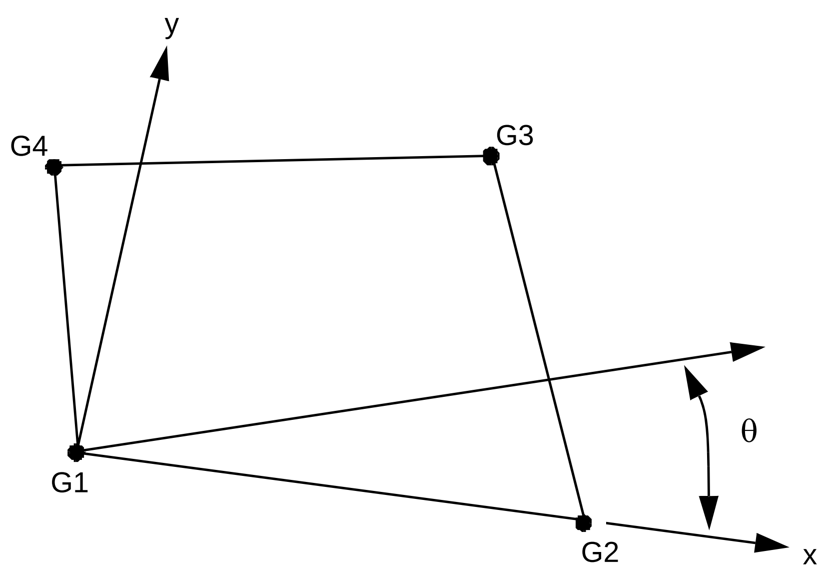


Figure 6-30

6.7.50 CQUAD8

Data Entry: **CQUAD8** - Curved Quadrilateral Shell Element Connection.

Description: Defines a curved quadrilateral shell element (QUAD8) with eight grid points. This is a general purpose shell element suitable for thick and thin shell analysis. Also defines a plane strain element and a conductive element for heat transfer analysis. Alternatively, when used with PSKIN, CQUAD8 defines non-structural skin elements.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD8	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8					THETA	ZOFFS	

Example:

1	2	3	4	5	6	7	8	9	10
CQUAD8	64	20	12	45	46	54	71	77	84
	91	95							

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PSHELL**, **PCOMP**, **PCOMPG** or **PSKIN** property data (Integer > 0 or blank, default is EID).
4-9,12-13	G1-G8	**GRID** identification numbers of connection point (Integers > 0, all unique).
18	THETA	Material property orientation specification (Real or blank; or Integer $\geq$ 0). If Real, specifies the material property orientation angle in degrees. See **Figure 6-31**. If Integer, the orientation of the material x-axis is along the projection onto the plane of the element of the x-axis of the coordinate system specified by the integer value.
19	ZOFFS	Offset from the surface of grid points to the element reference plane (see Remark 5) (Real or blank. Default=0.0).

Remarks:

1. Element identification numbers must be unique with respect to <u>all</u> other element identification numbers.
2. Grid points G1 through G8 must be ordered as shown in **Figure 6-31**.
3. The orientation of the material property coordinate system is defined locally at each integration point by THETA, which is the angle between $x_{material}$ and η.

6

4. It is recommended that the midside grid points be located within the middle third of edge.
5. Elements may be offset from the grid point surface by means of ZOFFS. Other data, such as material matrices and stress fiber location are given relative to the reference plane. Positive offset implies that the element reference plane lies above the grid points in the sketch below.
6. QUAD8 elements that reference PSHELL or PCOMP/PCOMPG can be loaded using **PLOAD2**, **PLOAD4** or **PLOAD5** data.
7. For QUAD8 elements that reference PSHELL data, forces, stresses and strains can be output in the element coordinate system, material coordinate system, basic or any defined coordinate system. The choice can be done in the PSHELL data statement (default is the element coordinate system).
8. For QUAD8 elements that reference PCOMP/PCOMPG data, stresses and strains are printed, if requested, in the layer coordinate system at the middle of the layer (half of its thickness). Forces are printed in the element coordinate system at the reference plane.
9. QUAD8 requires the specification of MID1, MID2 and MID3 on PSHELL card since it is a general shell element suitable for thick and thin shell analysis.When used with PCOMP/PCOMPG, the user should ensure that the resulting equivalent material properties for the stack of lamina has non-zero shear properties.
10. QUAD8 elements that reference **PSKIN** data are known as skin elements. Skin elements do not contribute stiffness, mass, conduction, load or anything to the structural or heat transfer problems. THETA or ZOFFS are ignored for skin elements. Skin elements can be used with the design data DTGRID to define a topography region.
11. The analysis parameter **THETA** provides a default value for THETA.

This is a shell element with inplane rotational stiffness.

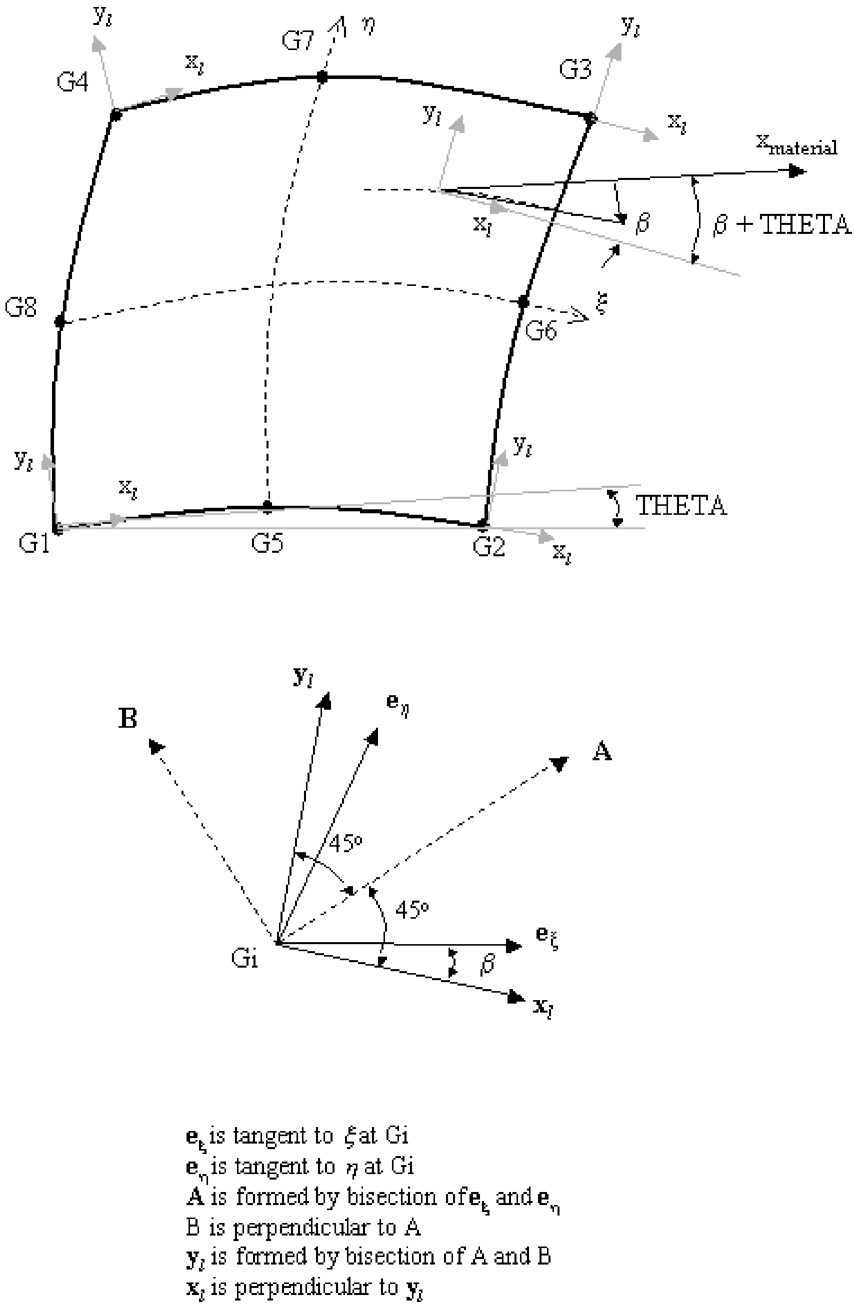

y_l
G7
η
G4
x_l
G3
x_material
β
β + THETA
G8
G6
ξ
THETA
G1
G5
G2
y_l
e_η
B
A
45°
45°
e_ξ
Gi
β
x_l
e_ξ is tangent to ξ at Gi
e_η is tangent to η at Gi
A is formed by bisection of e_ξ and e_η
B is perpendicular to A
y_l is formed by bisection of A and B
x_l is perpendicular to y_l

Figure 6-31

6.7.51 CROD

Data Entry: **CROD** - Rod Element Connection.

Description: Defines a tension-compression element (ROD) of the structural model. Also defines a conductive element for heat transfer analysis.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
CROD	17	10	63	91					

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PROD** property data (Default is EID) (Integer > 0 or blank).
4,5	G1,G2	**GRID** identification numbers of connection points (Integer > 0; $G1 \neq G2$).

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.

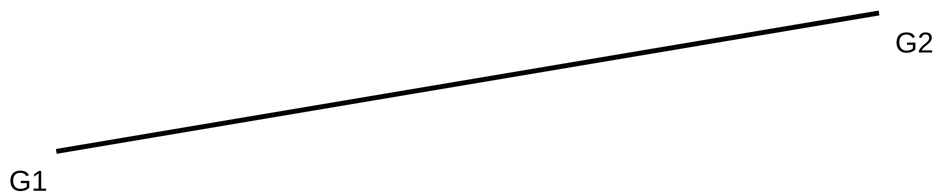


Figure 6-32

6.7.52 CSHEAR

Data Entry: **CSHEAR** - Shear Panel Element Connection.

Description: Defines a shear panel element (CSHEAR) of the structural model.

Format:

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

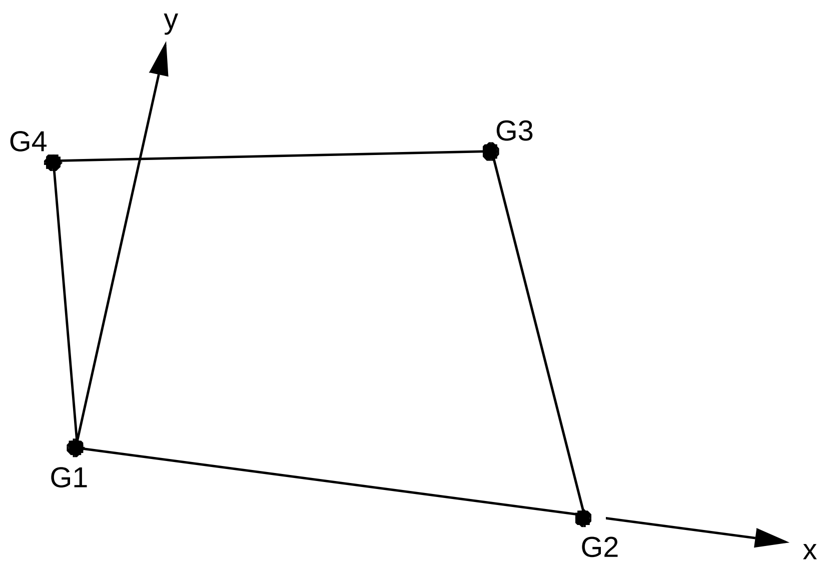
Example:

1	2	3	4	5	6	7	8	9	10
CSHEAR	22	3	15	18	4	17			

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PSHEAR** property data (Default is EID) (Integer > 0 or blank).
4-7	G1-G4	**GRID** identification numbers of connection points (Integer > 0).

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All interior angles must be less than 180^{o}.


y
G4
G3
G1
G2
x

6.7.53 CTETRA

Data Entry: **CTETRA** - Four sided Solid Element with 4 or 10 grid points.

Description: Defines the connections of the CTETRA 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:

1	2	3	4	5	6	7	8	9	10
CTETRA	122	3	15	18	4	17			

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PSOLID** property data (Default is EID) (Integer > 0 or blank).
4-9, 2-5	G1-G10	**GRID** identification numbers of connection points (Integer > 0).

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The element coordinate system is defined as follows: The origin is located at G1 and the x-axis lies on the G1-G2 edge. The y-axis lies in the G1-G2-G3 plane and is perpendicular to the x-axis, with the positive y-axis on the same side of the G1-G2 edge as node G3. The z-axis is orthogonal to the x and y axes.
3. A face of the TETRA element can be loaded using **PLOAD4** data.
4. Components of stress and strain, requested by the solution control commands STRESS and STRAIN, are output in the material coordinate system defined on the PSOLID data
5. Components of stress at grids of solid elements, requested by GSTRESS in the solution control, are printed in the basic coordinate system.
6. Grid points G5 through G10 are optional. If any grid points, G5 through G10 exist, all must be specified.
7. If the referenced PSOLID has “PFLUID” in the FCTN field, then this CTETRA defines a fluid element for a coupled fluid-structure problem. In this case, all referenced grid points must be fluid grids (have -1 in the CD field). Fluid elements have no stress or strain output, and may not be referenced by PLOAD4 data.

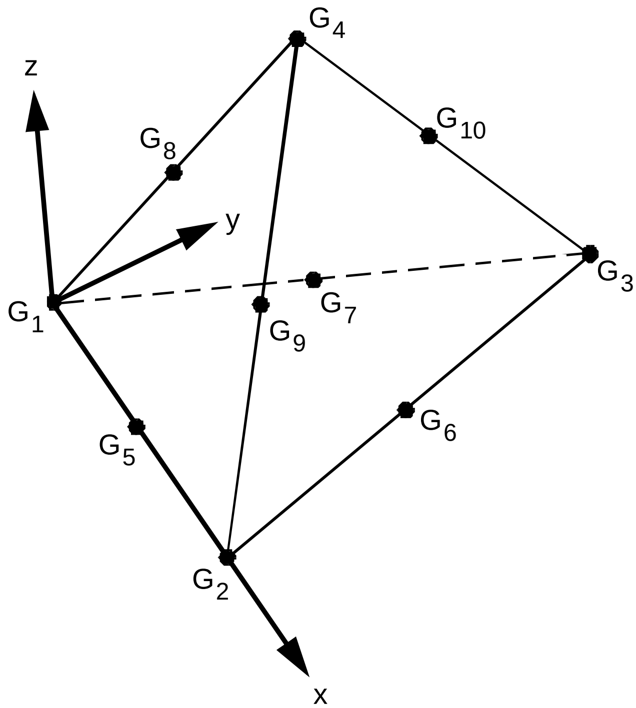


Figure 6-33

6

6.7.54 CTRIA3

Data Entry: **CTRIA3** - Triangular Element Connection.

Description: Defines a three node triangular plate element (TRIA3) of the structural model. This is an isoparametric membrane-bending element. Also defines a conductive element for heat transfer analysis. Alternatively, when used with PSKIN, CTRIA3 defines non-structural skin elements.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
CTRIA3	100	8	150	152	68				

Field	Information	Description
2	EID	Unique element identification number (integer > 0).
3	PID	Identification number of a **PSHELL**, **PCOMP**, **PCOMPG** or **PSKIN** property data (Integer > 0 or blank, default is EID).
4-6	G1,G2,G3	**GRID** identification numbers of connection points (Integers > 0, all unique).
7	THETA	Material property orientation specification (Real or blank; or Integer ≥ 0). If Real, specifies the material property orientation angle in degrees. The sketch below gives the sign convention for THETA. If Integer, the orientation of the material x-axis is along the projection onto the plane of the element of the x-axis of the coordinate system specified by the integer value. See remarks 7 & 8.
8	ZOFFS	Offset from the surface of the grid points to the element reference plane. See Remark 2 and 7. (Real or blank. Default=0.0).

Remarks:

1. Element identification numbers must be unique with respect to <u>all</u> other element identification numbers.
2. Elements may be offset from the grid point surface by means of ZOFFS. Other data, such as material matrices and stress fiber locations are given relative to the reference plane. Positive offset implies that the element reference plane lies above the grid points in the sketch.
3. The element coordinate system is shown in the figure below.

6

4. TRIA3 elements that reference PSHELL or PCOMP/PCOMPG can be loaded using **PLOAD2**, **PLOAD4** or **PLOAD5** data.
5. For TRIA3 elements that reference PSHELL data, forces, stresses and strains can be printed in the element coordinate system, material coordinate system, basic or any defined system. The choice can be done in the PSHELL data statement (default is the element coordinate system).
6. For CTRIA3 elements that reference PCOMP/PCOMPG data, stresses and strains are printed, if requested, in the layer coordinate system at the middle of the layer (half of its thickness). Forces are printed in the element coordinate system at the reference plane.
7. CTRIA3 elements that reference **PSKIN** data are known as skin elements. Skin elements do not contribute to stiffness, mass, conduction, load or anything to the structural or heat transfer problems. THETA or ZOFFS are ignored for skin elements. Skin elements can be used with the design data DTGRID to define a topography region.
8. The analysis parameter **THETA** provides a default value for THETA. See page 565.

This is a plate element with inplane rotational stiffness.

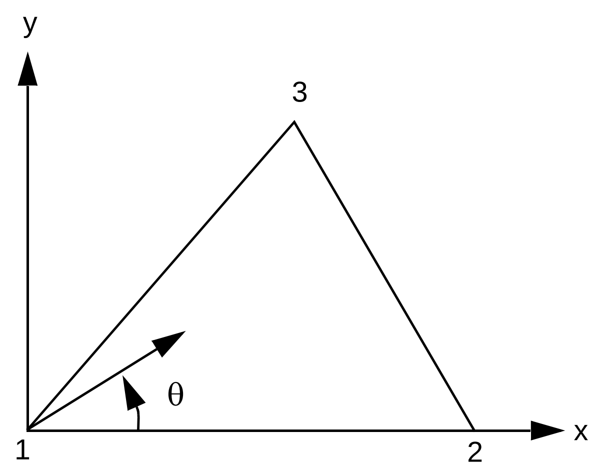


Figure 6-34

6.7.55 CTRIA6

Data Entry: **CTRIA6** - Curved Triangular Shell Element Connection.

Description: Defines a curved triangular shell element (TRIA6) with six grid points. This is a general purpose shell element suitable for thick and thin shell analysis. Also defines a plane strain element and a conductive element for heat transfer analysis. Alternatively, when used with PSKIN, CTRIA6 defines non-structural skin elements.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIA6	EID	PID	G1	G2	G3	G4	G5	G6	
	THETA or MCID	ZOFFS							

Example:

1	2	3	4	5	6	7	8	9	10
CTRIA6	100	8	150	152	68	112	130	142	

Field	Information	Description
2	EID	Unique element identification number (integer > 0).
3	PID	Identification number of a **PSHELL**, **PCOMP**, **PCOMPG** or **PSKIN** property data (Integer > 0 or blank, default is EID).
4-9	G1,G2,G3,G4, G5,G6	**GRID** identification numbers of connection points (Integers > 0, all unique).
12	THETA	Material property orientation specification (Real or blank; or Integer ≥ 0). If Real, specifies the material property orientation angle in degrees. **Figure 6-35** below shows the sign convention for THETA. If Integer, the orientation of the material x-axis is along the projection onto the plane of the element of the x-axis of the coordinate system specified by the integer value.
13	ZOFFS	Offset from the surface of the grid points to the element reference plane. See Remark 5 & 10. (Real or blank. Default=0.0).

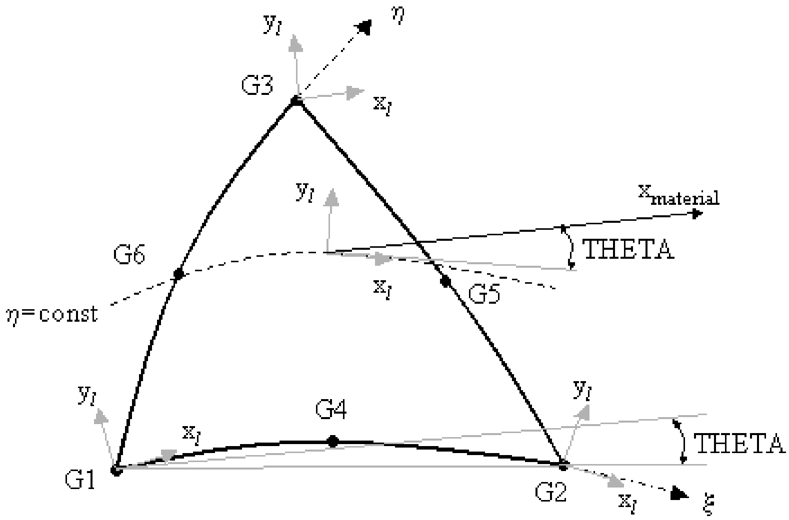
Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Grid points G1 to G6 must be numbered as shown in **Figure 6-35**
3. The orientation of the material property coordinate system is defined locally at each interior integration point by THETA, which is the angle between $x_{material}$ and the line of constant η.

6

4. It is recommended that midside grid points be located within the middle third of the edge.
5. Elements may be offset from the grid point surface by means of ZOFFS. Other data, such as material matrices and stress fiber locations are given relative to the reference plane. Positive offset implies that the element reference plane lies above the grid points in the sketch below.
6. TRIA6 elements that reference PSHELL or PCOMP/PCOMPG can be loaded using **PLOAD2**, **PLOAD4** or **PLOAD5** data.
7. For TRIA6 elements that reference PSHELL data, forces, stresses and strains can be printed in the element coordinate system, material coordinate system, basic or any defined system. The choice can be done in the PSHELL data statement (default is the element coordinate system).
8. For TRIA6 elements that reference PCOMP/PCOMPG data, stresses and strains are printed, if requested, in the layer coordinate system at the middle of the layer (half of its thickness). Forces are printed in the element coordinate system at the reference plane
9. TRIA6 requires the specification of MID1, MID2 and MID3 on PSHELL entries since it is a general purpose shell element suitable for thick and thin shell analysis. When used with PCOMP/PCOMPG, the user should ensure that the resulting equivalent material properties for the stack of lamina has non-zero shear properties
10. TRIA6 elements that reference **PSKIN** data are known as skin elements. Skin elements do not contribute stiffness, mass, conduction, load or anything to the structural or heat transfer problems. THETA or ZOFFS are ignored for skin elements. Skin elements can be used with the design data DTGRID to define a topography region.
11. The analysis parameter **THETA** provides a default value for THETA.

This is a shell element with inplane rotational stiffness.



y_l
η
G3
x_l
y_l
$x_{material}$
G6
THETA
x_l
G5
η=const
y_l
y_l
G4
x_l
THETA
G1
G2
x_l
ξ

Figure 6-35

6

6.7.56 CTRIAX6

Data Entry: **CTRIAX6** - Six Node Axisymmetric Element Connection.

Description: Defines an isoparametric and axisymmetric triangular cross section ring element (TRIAX6) with midside grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIAX6	EID	PID	G1	G2	G3	G4	G5	G6	
	TH								

Example:

1	2	3	4	5	6	7	8	9	10
CTRIAX6	100	888	9	10	11	31	32	42	
	8.0								

Field	Information	Description
2	EID	Unique element identification number (integer > 0).
3	PID	Identification number of a **PAXIS** property data (Integer > 0).
4-9	G1 thru G6	**GRID** identification numbers of connection points (Integers > 0, all unique).
3	TH	Material property orientation angle (Real in degrees, default = 0.0)

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. The grid points must lie in the x-z plane of the basic coordinate system with $x \geq 0$. The grid points must be listed consecutively, beginning at a vertex and proceeding around the perimeter in a counter-clockwise direction.
3. The continuation data is not required.
4. Concentrated loads on grid circles for this element must be computed for 360^o, i.e., multiply load per unit length by $2\pi r$.
5. Stress and Strain, if requested in the Solution Control, are printed in the element coordinate system. Grid stress, if requested in the Solution Control, is printed in the basic coordinate system.

6

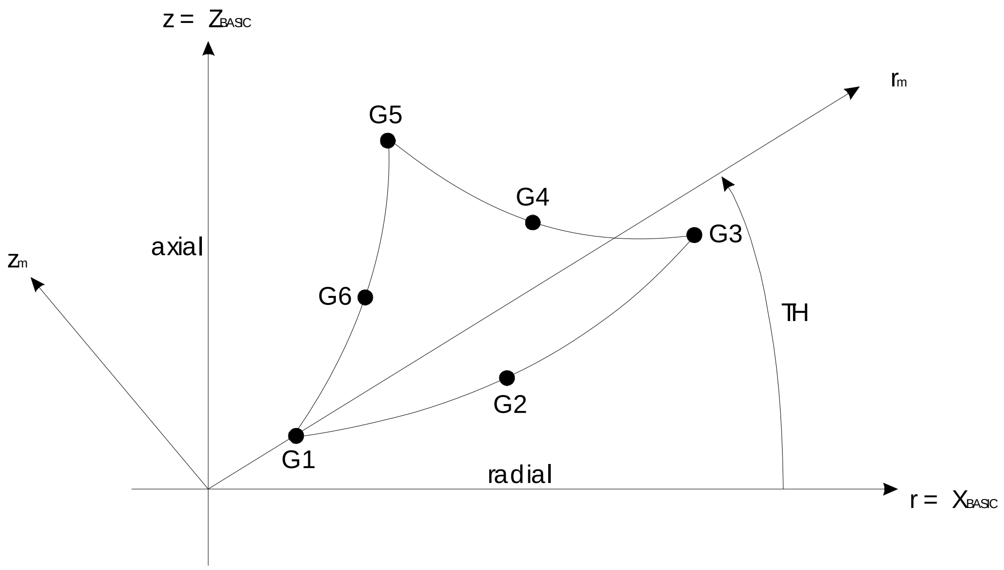


Figure 6-36

6.7.57 CTUBE

Data Entry: **CTUBE** - Tube (Rod) Element Connection.

Description: Defines a tension-compression element rod of the structural model. Also defines a conductive element for heat transfer analysis.

Format:

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

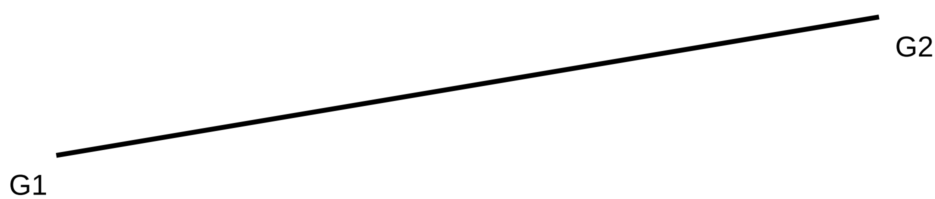
Example:

1	2	3	4	5	6	7	8	9	10
CTUBE	17	10	63	91					

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PTUBE** property data (Default is EID) (Integer > 0 or blank).
4,5	G1,G2	**GRID** identification numbers of connection points (Integer > 0; $G1 \neq G2$).

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.



2. CTUBE data will be converted to CROD data. This data entry is used for compatability with other codes. CROD data is a preferred data entry.

6.7.58 CVECTOR

Data Entry: **CVECTOR** - Vector Spring Element Connection.

Description: Defines a vector spring element of the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
CVECTOR	EID	PID	GA	GB	X1	X2	X3		

Alternate Format:

1	2	3	4	5	6	7	8	9	10
CVECTOR	EID	PID	GA	GB	GO				

Examples:

1	2	3	4	5	6	7	8	9	10
CVECTOR	2	39	7	3	10.5	300.25	50.3		

1	2	3	4	5	6	7	8	9	10
CVECTOR	2	39	7	3	13				

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of **PVECTOR** property data (default is EID), (Integer >0 or blank).
4,5	GA,GB	**GRID** identification numbers of connection points (Integer > 0; $GA \neq GB$, GB may be blank).
6-8	X1,X2,X3	Components of vector v, at end A (shown in the figure below), measured at end A, parallel to the components of the general coordinate system for GA, to determine (with the vector from end A to end B) the orientation of the element coordinate system for the vector element (Real or blank. See remarks 2 and 7).
6	GO	GRID identification number to optionally supply X1, X2, X3 (Integer > 0 or blank). See remark 2. Direction of orientation vector is GA to GO.

Remarks:

1. The vector element geometry is shown in the figure below.
2. If data in field 6 is integer, then GO is used. Otherwise, if data in field 6 is real, then X1, X2 and X3 are used.
3. $GO \neq GA$ or GB.
4. Element identification numbers must be unique with respect to all other element identification numbers.
5. If GO is specified, the vector orientation vector is updated at each design cycle when shape optimization is being performed.
6. Element forces are printed in the element coordinate system for ends a and b.
7. If the orientation vector is not provided, then the element coordinate system corresponds to the output coordinate system defined at grid A.
8. If the element is grounded (grid B is blank), then fields 6 to 8 must be blank.
9. The element coordinate system (V is not blank) is shown below.

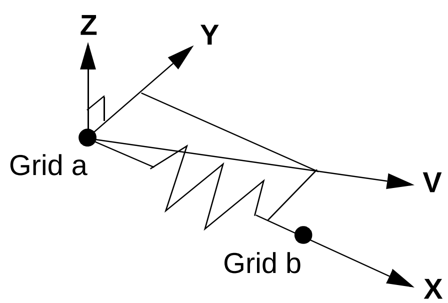


Figure 6-37

10. The force-displacement relationship is calculated by the following expression:

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \begin{bmatrix} K & -K \\ -K & K \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \quad (6\text{-}1)$$

where

$$F_1 = \begin{Bmatrix} F_{x1} \\ F_{y1} \\ F_{z1} \\ M_{x1} \\ M_{y1} \\ M_{z1} \end{Bmatrix} \qquad F_2 = \begin{Bmatrix} F_{x2} \\ F_{y2} \\ F_{z2} \\ M_{x2} \\ M_{y2} \\ M_{z2} \end{Bmatrix} = -F_1$$

$$K = \begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16} \\ & K_{22} & K_{23} & K_{24} & K_{25} & K_{26} \\ & & K_{33} & K_{34} & K_{35} & K_{36} \\ & & & K_{44} & K_{45} & K_{46} \\ & \mathrm{SYM} & & & K_{55} & K_{56} \\ & & & & & K_{66} \end{bmatrix}$$

u_1 and u_2 correspond to the displacements in the element coordinate system at the connection grids.

6.7.59 CVISC

Data Entry: **CVISC** - Viscous Damper Connection

Description: Defines a viscous damper element (VISC) of the structural model.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
CVISC	21	6327	29	31					

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PVISC** property entry (Default is EID) (Integer > 0 or blank).
4, 5	G1, G2	**GRID** identification number (Integer > 0; $G1 \neq G2$).

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Only one CVISC element may be defined on a single entry.

6

6.7.60 CWELD

Data Entry: **CWELD** - Weld Element Connection.

Description: Defines a weld or fastener element connecting two patches (PARTPAT, ELPAT, ELEMID, GRIDID) or a patch to point (ELEMID, GRIDID) or two points (ALIGN) .

Format PARTPAT:

1	2	3	4	5	6	7	8	9	10
CWELD	EID	PID	GS	PARTPAT	GA	GB			
	PIDA	PIDB							
	XS	YS	ZS						

Example:

CWELD	200	6	137	PARTPAT					
	1	2							

Alternate format ELPAT:

1	2	3	4	5	6	7	8	9	10
CWELD	EID	PID	GS	ELPAT	GA	GB			
+	SHIDA	SHIDB							
+	XS	YS	ZS						

Example:

CWELD	201	6	137	ELPAT					
	30	31							

Alternate Format ELEMID:

1	2	3	4	5	6	7	8	9	10
CWELD	EID	PID	GS	ELEMID	GA	GB			
+	SHIDA	SHIDB							

Example:

CWELD	202	7	101	ELEMID					
	101	221							

Alternate Format GRIDID:

1	2	3	4	5	6	7	8	9	10
CWELD	EID	PID	GS	GRIDID	GA	GB	SPTYP		
+	GA1	GA2	GA3	GA4					
+	GB1	GB2	GB3	GB4					

Example:

CWELD	301	6	121	GRIDID			QT		
+	12	23	58	61					
+	101	143	172						

Alternate Format ALIGN:

1	2	3	4	5	6	7	8	9	10
CWELD	EID	PID		ALIGN	GA	GB			

Example:

CWELD	401	7		ALIGN	143	209			

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
3	PID	Identification number of a **PWELD** property data (Integer > 0).
4	GS	**GRID** identification number of point which defines the location of connector (Integer > 0 or blank).
5	PARTPAT	Character string indicating that the connectivity of surface patch A to surface patch B is defined by two **PSHELL**, **PCOMP** or **PCOMPG** property identification numbers PIDA and PIDB. Up to 2x2 elements per patch are connected. See remark 3
6,7	GA,GB	**GRID** identification numbers of points piercing surfaces A and B. (Integer > 0; $GA \neq GB$).
2,3	PIDA,PIDB	Property identification numbers of **PSHELL**, **PCOMP** or **PCOMPG** entries defining surfaces A and B. (Integer > 0; $PIDA \neq PIDB$)
2-4	XS,YS,ZS	Coordinates of spot weld location in basic coordinate system (Real > 0 or blank).

6

Field	Information	Description
Alternate format ELPAT		
5	ELPAT	Character string indicating the connectivity of surface patch A to surface patch B is defined by two shell element identification numbers SHIDA and SHIDB. Up to 2x2 elements per patch are connected. See remark 3.
2-3	SHIDA,SHIDB	Element identification numbers of **CTRIA3**, **CQUAD4**, **CTRIA6** or **CQUAD8** elements on patch A and B (Integer > 0)
Alternate format ELEMID		
5	ELEMID	Character string indicating the connectivity of surface patch A to surface patch B is defined by two shell element identification numbers SHIDA and SHIDB. Only one shell element per patch is connected. See remark 5
2-3	SHIDA,SHIDB	Element identification numbers of **CTRIA3**, **CQUAD4**, **CTRIA6** or **CQUAD8** elements on patch A and B (Integer > 0)
Alternate format GRIDID		
5	GRIDID	Character string indicating the connectivity of surface patch A to surface patch B is defined by two sequences of grid point identification numbers GAi and GBi. The surface of any element can be connected. See remark 6
8	SPTYP	Character string indicating types of surface patches A and B. SPTYP = “QQ”, “TT”, “QT”, “TQ”, “Q” OR “T”. See remark 7
2-5	GAi,GBi	**GRID** identification numbers of surface patches A and B respectively. (Integer > 0).
Alternate format ALIGN		
	ALIGN	Character string indicating the connectivity of surface A to surface B is defined by two shell vertex grid points GA and GB respectively.
6,7	GA,GB	Shell vertex **GRID** identification numbers (Integer > 0).

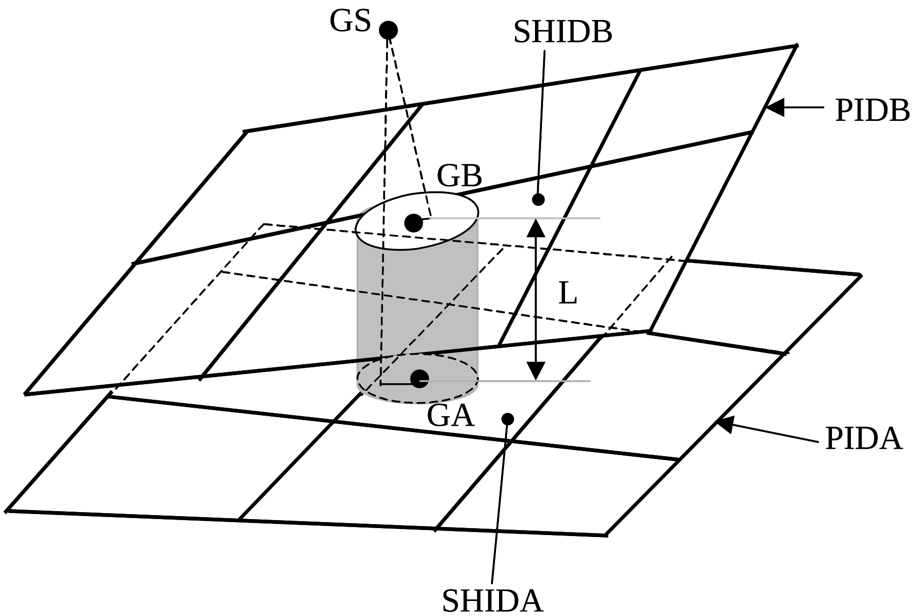


Figure 6-38 Patch-to-Patch connection defined with format PARTPAT or ELPAT

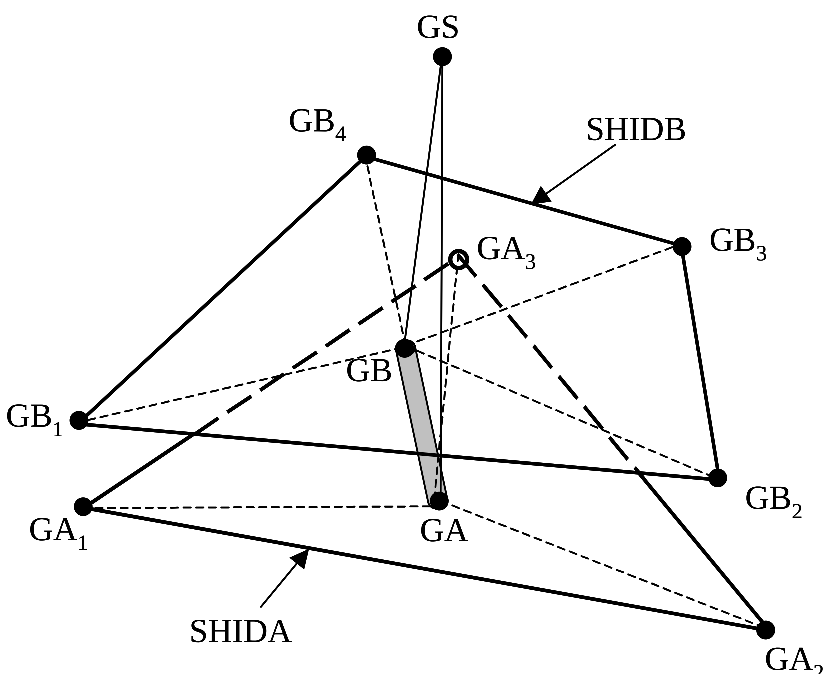


Figure 6-39 Patch-to-Patch connection defined with format ELEMID or GRIDID

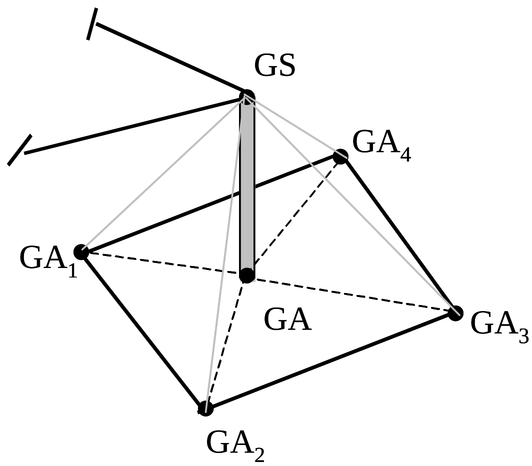


Figure 6-40 Point-to-Patch connection defined with format ELEMID or GRIDID

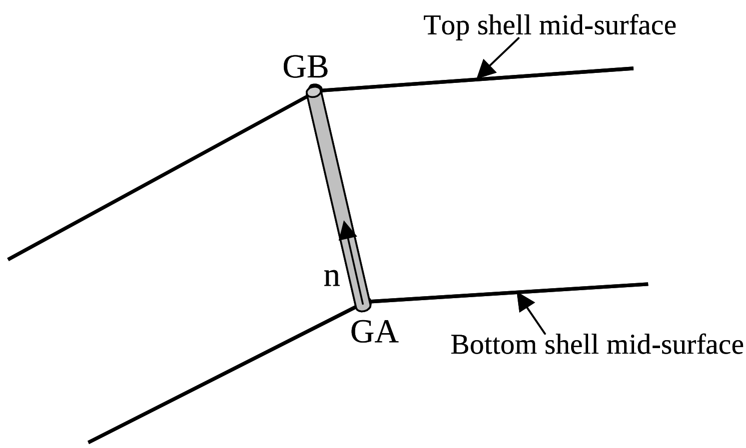


Figure 6-41 Point-to-Point connection defined with format ALIGN

Remarks:

1. Grid point GS defines the approximate location of the connector in space. GA and GB are the projections of GS onto surface patches A and B respectively. GS must have a normal projection on the surface patches. GS is ignored for format ALIGN. If GS is not specified, GA is used instead. For formats PARTPAT and ELPAT, if GA and GB are not specified, then XS, YS and ZS must be specified.
2. Connectivity between grid points on surface A and surface B is generated automatically using the location of GS and area of the cross-section of the connector. The user can print out connectivity information and change the default search and projection parameters with the **SWLDPRM** bulk data entry.

3. For the PARTPAT and ELPAT formats, depending on the location of the piercing points GA, GB and the diameter D defined on the **PWELD** entry, the number of elements per patch ranges from a single element up to 3x3 elements (see **Figure 6-38**).
4. The definition of piercing grid points GA and GB are optional for all formats except ALIGN. If GA and GB are defined, GS is ignored. If GA and GB are not specifed, they are generated from the normal projection of GS on surface patches A and B. If GA and GB are specified, they must lie on or at least have a projection on surface patches A and B respectively. The locations of GA and GB are corrected to lie on the surface patches. The length of the connector is the distance between GA and GB.
5. Format ELEMID defines a patch to patch connection with a single element per patch regardless of the diameter of the connector (see **Figure 6-39**). In addition, format ELEMID can define a point to patch connection if SHIDB is left blank. Then grid GS is connected to shell SHIDA (see **Figure 6-40**).
6. Format GRIDID defines a connection of two patches with sequences of grid points GAi and GBi (see **Figure 6-39**). GAi and GBi do not have to belong to shell elements. In addition, the format GRIDID can define a point to patch connection if all GBi fields are left blank. Then grid GS is connected to grids GAi (see **Figure 6-40**).
7. GAi are required for format GRIDID. Atleast 3 and at most 4 grid IDs may be specified for GAi and GBi respectively. SPTYP defines the type of surface patches to be connected and is required for format GRIDID to identify quadrilateral and triangular patches. The combinations are

SPTYP	Description
QQ	Connects quadrilateral surface patch A (Q4/Q8) with a quadrilateral surface patch B (Q4/Q8)
QT	Connects quadrilateral surface patch A (Q4/Q8) with a triangular surface patch B (T3/T6)
TT	Connects triangular surface patch A (T3/T6) with a triangular surface patch B (T3/T6)
TQ	Connects triangular surface patch A (T3/T6) with a quadrilateral surface patch B (Q4/Q8)
Q	Connects a shell vertex grid GS with a quadrilateral surface patch A (Q4/Q8)
T	Connects a shell vertex grid GS with a triangular surface patch A (T3/T6)

8. Forces and moments are output in the element coordinate system defined as follows: The element x-axis points from GA to GB. The element y-axis lies on the plane created by the element x-axis and the axis of basic coordinate system making the largest angle with the element x-axis, and is orthogonal to the element x-axis. The z-axis is the cross product of the element x-axis and the element y-axis.

9. The output format of the forces and moments, including sign convention, is identical to the CBAR element.

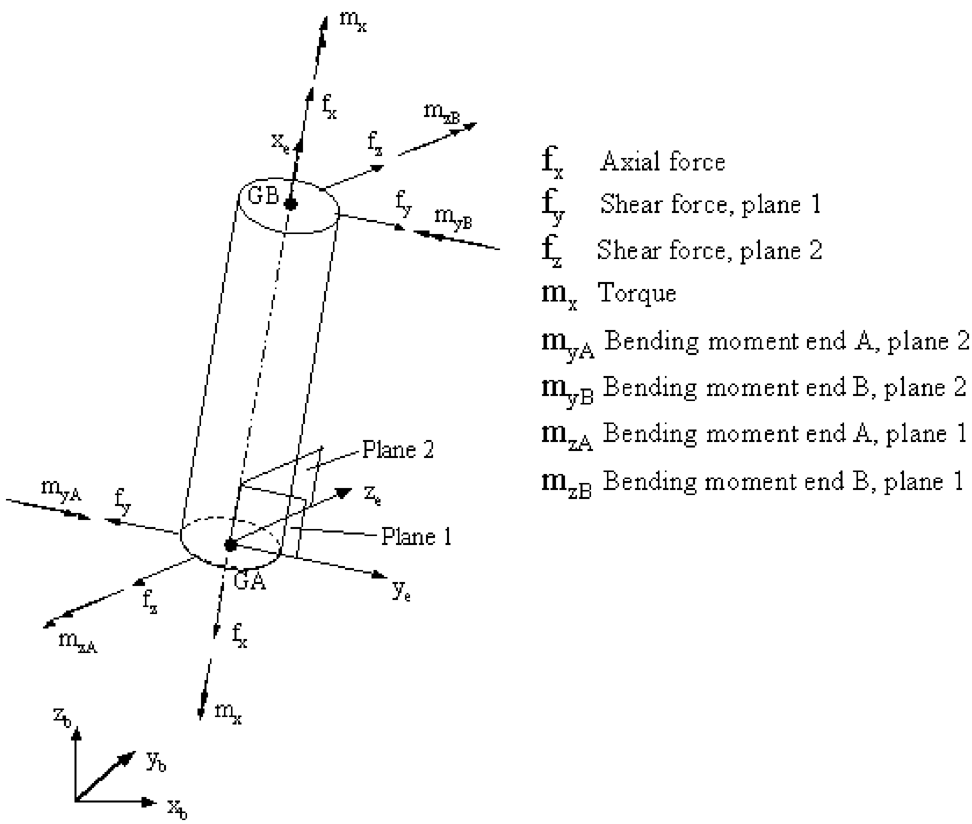


Figure 6-42 CWELD Element Forces

6.7.61 DAREA

Data Entry: **DAREA** - Dynamic Load Scale Factor

Description: This data is used in conjunction with the RLOAD1 or RLOAD2 data entries, and defines the point where the dynamic load is to be applied with the scale (area) factor, A

Format:

1	2	3	4	5	6	7	8	9	10
DAREA	SID	P	C	A	P	C	A		

Example:

1	2	3	4	5	6	7	8	9	10
DAREA	3	6	2	8.2	15	1	10.1		

Field	Information	Description
2	SID	Identification number of DAREA set (Integer > 0).
3, 6	P	**GRID** or **SPOINT** identification number (Integer > 0).
4, 7	C	Component number (Integer 1-6 for grid point; blank or 0 for scalar point).
5, 8	A	Scale (area) factor, A, for the designated coordinate (Real).

Remarks:

1. One or two scale factors may be defined on a single entry.
2. Refer to **RLOAD1** or **RLOAD2** data for the formulas which define the scale factor, A.
3. Component numbers refer to global coordinates.
4. DAREA entries may be replaced by, or used with, FORCEi, MOMENTi, PLOADi and GRAV data.

6.7.62 DEFORM

Data Entry: **DEFORM** - Initial axial deformation on CROD, CBAR and CBEAM elements.

Description: Defines an initial non-elastic axial deformation for CROD, CBAR and CBEAM elements.

Format:

1	2	3	4	5	6	7	8	9	10
DEFORM	SID	EID	DEF	EID	DEF	EID	DEF		

Example:

1	2	3	4	5	6	7	8	9	10
PLOAD2	5	101	2.5	102	-2.5				

Field	Information	Description
2	SID	Deformation set identification number (Integer > 0).
3,5,7	EID	Element identification number of a **CROD**, **CBAR**, or **CBEAM** element (Integer > 0).
4,6,8	DEF	Enforced initial deformation (Real, a positive value indicates elongation).

Remarks:

1. Deform sets can be selected on static load cases with the solution control command (**DEFORM**=SID)
2. Only one deformation per element per deform set is allowed.
3. All elements referenced must exist.
4. One to three element deformations may be defined per entry.

6

6.7.63 DELAY

Data Entry: **DELAY** - Dynamic Load Time Delay

Description: This data is used in conjunction with the RLOAD1 or RLOAD2 data entries, and defines the time delay term, τ, in the equation of the loading function.

Format:

1	2	3	4	5	6	7	8	9	10
DELAY	SID	P	C	T	P	C	T		

Example:

1	2	3	4	5	6	7	8	9	10
DELAY	5	21	6	4.25	7	6	8.1		

Field	Information	Description
2	SID	Identification number of DELAY set (Integer > 0).
3, 6	P	**GRID** or **SPOINT** identification number (Integer > 0).
4, 7	C	Component number (Integer 1-6 for grid point; blank or 0 for scalar point).
5, 8	T	Time delay term, τ, for the designated coordinate (Real).

Remarks:

1. One or two dynamic load time delays may be defined on a single entry.
2. Refer to **RLOAD1** or **RLOAD2** data for the formulas which define the manner in which the time delay term, τ, is used.
3. **DAREA** data should exist for the same grid point and component.

6

6.7.64 DISTOR

Data Entry: **DISTOR -** Overrides Element Distortion Parameter

Description: Override internal limits for warnings and errors of distortion parameters.

Format:

1	2	3	4	5	6	7	8	9	10
DISTOR	ELEMENT	TYPE	ERR/WAR	OPTION	V1	V2			

Examples:

1	2	3	4	5	6	7	8	9	10
DISTOR	TRIA3	ARATIO	ERROR	1		20000.0			
DISTOR	QUAD4	SKEW	WARNING	2		0.01			
DISTOR	QUAD4	ANGLE	WARNING	1	10.0	178.0			

Field	Information	Description
2	ELEMENT	Element type: TRIA3, QUAD4, TRIA6, QUAD8, SHEAR, TRIAX6, TETRA, TET10, PENTA, PYRA, HEXA or HEX20 (Word).
3	TYPE	Element characteristic: ARATIO, SKEW, TAPER, WARP, TWIST, EDGE, COLLAP, ANGLE, HOENOR or HOETAN (Word).
4	ERR/WAR	ERROR or WARNING (Word).
5	OPTION	Options are: 1 to enter limit values or 2 to interpolate value between default value and maximum value (Integer 1 or 2).
6	V1	Lower limit value (OPTION=1) or scalar value to interpolate to find lower limit value (OPTION=2) (REAL for OPTION=1 or $0 \leq$ REAL ≤ 1 for OPTION=2).
7	V2	Upper limit value (OPTION=1) or scalar value to interpolate to find Upper limit value (OPTION=2) (REAL for OPTION=1 or $0 \leq$ REAL ≤ 1 for OPTION=2).

6

Remarks:

1. When option 2 is used, the limit values are calculated using the expression LIMIT = DLIMIT + (MLIMIT - DLIMIT)*VI, where DLIMIT represents the default limit and MLIMIT is the maximum limit allowed. When DLIMIT is equal to MLIMIT, the limits are not changed
2. The default bound values and maximum bound values are listed below.
3. A summary of the bound values can be printed using ECHO=SORT or ECHO=BOTH.
4. For a detailed definition of the distortion parameters, see **Element Verification** (Sec. 2.20).
5. Shape checking is performed on both regular elements and DOMAIN elements.

TRIA3 Default and Maximum Bound Values

Concept	TYPE	Default limit for Warning Message		Default limit for Error Message		Maximum Limit	
		LL	UL	LL	UL	LL	UL
Aspect ratio	ARATIO	-	50.0	-	500.0	1.0	1.0E5
Skew angle	SKEW	-	75.0	-	90.0	0.0	90.0

QUAD4 and SHEAR Default and Maximum Bound Values

Concept	TYPE	Default limit for Warning Message		Default limit for Error Message		Maximum Limit	
		LL	UL	LL	UL	LL	UL
Aspect ratio	ARATIO	-	100.0	-	1000.0	1.0	1.0E5
Skew angle	SKEW	-	60.0	-	75.0	0.0	90.0
Taper	TAPER	-	1.0	-	1.0	0.0	1.0
Warp angle	WARP	-	30.0	-	45.0	0.0	90.0
Interior angles	ANGLES	15.0	165.0	0.0	180.0	0.0	180.0

TRIA6 Default and Maximum Bound Values

Concept	TYPE	Default limit for Warning Message		Default limit for Error Message		Maximum Limit	
		LL	UL	LL	UL	LL	UL
Aspect ratio	ARATIO	1.0	50.0	1.0	500.0	1.0	1.0E5
Skew angle	SKEW	0.0	75.0	0.0	90.0	0.0	90.0
Hoe Normal Offset	HOENOR	0.0	0.3	0.0	0.60	0.0	1.0E5
Hoe Tangent Offset	HOETAN	0.0	0.2	0.0	0.24	0.0	0.25

QUAD8 Default and Maximum Bound Values

Concept	TYPE	Default limit for Warning Message		Default limit for Error Message		Maximum Limit	
		LL	UL	LL	UL	LL	UL
Aspect ratio	ARATIO	-	100.0	-	1000.0	1.0	1.0E5
Skew angle	SKEW	-	60.0	-	75.0	0.0	90.0
Taper	TAPER	-	1.0	-	1.0	0.0	1.0
Warp angle	WARP	-	30.0	-	45.0	0.0	90.0
Interior angles	ANGLES	15.0	165.0	-	180.0	0.0	180.0
Hoe normal offset	HOENOR	-	0.30	-	0.60	0.0	1.0E5
Hoe tangent offset	HOETAN	-	0.2	-	0.24	0.0	0.25

TRIAX6 Default and Maximum Bound Values

Concept	TYPE	Default limit for Warning Message		Default limit for Error Message		Maximum Limit	
		LL	UL	LL	UL	LL	UL
Aspect ratio	ARATIO	-	50.0	-	500.0	1.0	1.0E5
Skew angle	SKEW	-	75.0	-	90.0	0.0	90.0
Hoe Normal Offset	HOENOR	-	0.3	-	0.60	0.0	1.0E5
Hoe Tangent Offset	HOETAN	-	0.25	-	0.50	0.0	0.50

TETRA and TET10 Default and Maximum Bound Values

Concept	TYPE	Default limit for Warning Message		Default limit for Error Message		Maximum Limit	
		LL	UL	LL	UL	LL	UL
Aspect ratio	ARATIO	0.001	100.0	0.001	1000.0	0.0	1.0E5
Skew angle	SKEW	-	75.0	-	90.0	0.0	90.0
Collapse	COLLAP	0.001	100.0	0.0	100.0	0.0	1000.0
Edge angle	EDGE	-	75.0	-	90.0	0.0	90.0
Hoe normal offset	HOENOR	-	0.30	-	0.60	0.0	1.0E5
Hoe tangent offset	HOETAN	-	0.25	-	0.50	0.0	0.50

PENTA (6 node) Default and Maximum Bound Values

Concept	TYPE	Default limit for Warning Message		Default limit for Error Message		Maximum Limit	
		LL	UL	LL	UL	LL	UL
Aspect ratio	ARATIO	-	1000.0	-	1000.0	1.0	1.0E5
Skew angle	SKEW	-	60.0	-	75.0	0.0	90.0
Taper angle	TAPER	-	1.0	-	1.0	0.0	1.0
Warp angle	WARP	-	30.0	-	30.0	0.0	90.0
Twist angle	TWIST	-	30.0	-	75.0	0.0	90.0
Edge angles	EDGE	-	75.0	-	90.0	0.0	90.0

PENTA (15 node) Default and Maximum Bound Values

Concept	TYPE	Default limit for Warning Message		Default limit for Error Message		Maximum Limit	
		LL	UL	LL	UL	LL	UL
Aspect ratio	ARATIO	-	1000.0	-	1000.0	1.0	1.0E5
Skew angle	SKEW	-	60.0	-	75.0	0.0	90.0
Taper angle	TAPER	-	1.0	-	1.0	0.0	1.0
Warp angle	WARP	-	30.0	-	60.0	0.0	90.0
Twist angle	TWIST	-	30.0	-	75.0	0.0	90.0
Edge angles	EDGE	-	75.0	-	88.0	0.0	90.0
Hoe Normal offset	HOENOR	-	0.3	-	0.6	0.0	1.0E5
Hoe Tangent offset	HOETAN	-	0.2	-	0.24	0.0	0.25

PYRA (5 node) Default and Maximum Bound Values

Concept	TYPE	Default limit for Warning Message		Default limit for Error Message		Maximum Limit	
		LL	UL	LL	UL	LL	UL
Aspect ratio	ARATIO	1.0	100.0	1.0	1000.0	1.0	1.0E5
Skew angle	SKEW	0.0	60.0	0.0	75.0	0.0	90.0
Warp angle	WARP	0.0	30.0	0.0	60.0	0.0	90.0
Face Vertex angle	ANGLES	15.0	165.0	0.0	180.0	0.0	180.0

PYRA (13 node) Default and Maximum Bound Values

Concept	TYPE	Default limit for Warning Message		Default limit for Error Message		Maximum Limit	
		LL	UL	LL	UL	LL	UL
Aspect ratio	ARATIO	1.0	100.0	1.0	1000.0	1.0	1.0E5
Skew angle	SKEW	0.0	60.0	0.0	75.0	0.0	90.0
Warp angle	WARP	0.0	30.0	0.0	60.0	0.0	90.0
Face Vertex angle	ANGLES	15.0	165.0	0.0	180.0	0.0	180.0
Hoe Normal offset	HOENOR	0.0	0.3	0.0	0.6	0.0	1.0E5
Hoe Tangent offset	HOETAN	0.0	0.2	0.0	0.24	0.0	0.25

HEXA (8 node*) Default and Maximum Bound Values

Concept	TYPE	Default limit for Warning Message		Default limit for Error Message		Maximum Limit	
		LL	UL	LL	UL	LL	UL
Aspect ratio	ARATIO	-	100.0	-	1000.0	1.0	1.0E5
Face Skew angle	SKEW	-	60.0	-	90.0	0.0	90.0
Face taper	TAPER	-	1.0	-	1.0	0.0	1.0
Face Warp angle	WARP	-	30.0	-	45.0	0.0	90.0
Twist angle	TWIST	-	30.0	-	90.0	0.0	90.0
Edge angles	EDGE	-	60.0	-	90.0	0.0	90.0

* For HEXA elements with 9 or more nodes, see next table.

HEX20 or HEXA (When 9-21 nodes are used) Default and Maximum Bound Values

Concept	TYPE	Default limit for Warning Message		Default limit for Error Message		Maximum Limit	
		LL	UL	LL	UL	LL	UL
Aspect ratio	ARATIO	-	100.0	-	1000.0	1.0	1.0E5
Face Skew angle	SKEW	-	60.0	-	75.0	0.0	90.0
Face taper	TAPER	-	1.0	-	1.0	0.0	1.0
Face Warp angle	WARP	-	30.0	-	45.0	0.0	90.0
Twist angle	TWIST	-	30.0	-	75.0	0.0	90.0
Edge angles	EDGE	-	60.0	-	90.0	0.0	90.0
Hoe normal offset	HOENOR	-	0.30	-	0.60	0.0	1.0E5
Hoe tangent offset	HOETAN	-	0.25	-	0.50	0.0	0.50

6.7.65 DLOAD

Data Entry: **DLOAD** - Dynamic Load Set Combination.

Description: Defines a new dynamic load set as a linear combination of dynamic loads from RLOAD1, RLOAD2 and RLOAD3 entries.

Format:

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	SID1	S2	SID2	S3	SID3	
+	S4	SID4	-etc.-						

Example:

1	2	3	4	5	6	7	8	9	10
DLOAD	10	1.0	4.0	1	2.0	2	-1.0	3	
+	5.0	4							

Field	Information	Description
2	SID	Load set identification number (Integer > 0).
3	S	Overall scale factor (Real or Blank. Default = 1.0).
4, 6, 8 2, 4, ..	Si	Scale factor for dynamic load set SIDi (Real).
5, 7, 9 3, 5, ..	SIDi	Load set ID used by **RLOAD1**, **RLOAD2** and/or **RLOAD3** entries (Integer > 0).

Remarks:

1. The load vector is created as follows:

$$F = S \sum_i S_i F_{SIDi}$$

2. Dynamic load sets can be selected in the Solution Control Section (**DLOAD** = SID).
3. The load set IDs (SIDi) must be unique.
4. Load sets defined on other DLOAD entries may not be referenced.

6

6.7.66 DMIG

Data Entry: **DMIG** - Direct Matrix Input Using Grid/Component.

Description: Defines a matrix using GRID IDs and components to specify degrees of freedom. The matrix may be added to the global stiffness matrix, the global mass matrix or the static load vectors.

The matrix is specified by one Header Entry followed by a Column Entry for every column in the matrix. The Header and Column entries are linked by using the same NAME in field 2.

Header Format:

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	0	FORM	TYPE		POLAR		NCOL	

Column Format for FORM = 6:

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	GJ	CJ		G1	C1	A1	B1	
+	G2	C2	A2	B2	G3	C3	A3	B3	
+	G4	C4	A4	B4	-etc.-				

Column Format for FORM = 9:

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	COLJ			G1	C1	A1		
+	G2	C2	A2		G3	C3	A3		
+	G4	C4	A4		-etc.-				

Example:

1	2	3	4	5	6	7	8	9	10
DMIG	M2	0	6	1					
DMIG	M2	101	1		101	1	1.234-3		
DMIG	M2	101	2		101	2	1.234-3		
DMIG	M2	101	3		101	3	1.234-3		
DMIG	M2	101	4		101	1	2.345-1		
+	101	2	-3.456-1		101	4	4.567+1		
DMIG	M2	101	5		101	1	3.456-1		
+	101	3	-5.678-1		101	5	6.789+1		
DMIG	M2	101	6		101	2	5.678-1		
+	101	3	7.890-1		101	6	8.901+1		

6

Field	Information	Description
2	NAME	Name of the Matrix (Character).
3	FORM	Form of the matrix (Integer 6 or 9) 6 = Symmetric 9 = Rectangular.
4	TYPE	Type of the matrix (Integer 1-4) 1 or 2 = Real 3 or 4 = Complex.
7	POLAR	Complex number entry style flag (Integer 0 or 1 or blank) (Default = 0). 0 = Ai,Bi are the Real, Imaginary components, respectively. 1 = Ai,Bi are the Magnitude and Phase (Bi in degrees).
9	NCOL	Number of columns in the matrix. Only used if FORM = 9 (Integer > 0 or blank).
3,4	GJ, CJ	**GRID** or **SPOINT** identification number and component code specifying the degree of freedom of the column (GJ: Integer > 0; CJ: Integer 1-6 or blank).
3	COLJ	Column number (0 < Integer ≤ NCOL).
6,7;2,3	Gi, Ci	GRID or SPOINT identification number and component code specifying the degree of freedom of the row (Gi: Integer > 0; Ci: Integer 1-6 or blank).
8,9;4,5	Ai, Bi	Value of the matrix term in row (Gi,Ci) of column (GJ,CJ) or COLJ (Real).

Remarks:

1. DMIG entries are not used unless selected by solution control commands **K2GG**, **K2PP**, **M2GG** or **P2G**. Matrices selected by K2GG, M2GG or P2G must be real (TYPE = 1 or 2). Matrices selected by K2PP may be real or complex.
2. The header entry is identified by a 0 in field 3. Exactly one header entry must be used for each different matrix. Column entries are required for every non-null column in the matrix.
3. If the matrix is real (TYPE = 1 or 2), the Bi fields must be blank.
4. If the matrix is rectangular (FORM=9), the matrix must be real (TYPE = 1 or 2).
5. If the matrix is symmetric (FORM=6), an off-diagonal term may be input into either the upper or lower triangle of the matrix, and the corresponing term in the other triangle will be automatically generated.
6. If multiple values are specifed for the same row-column term of the matrix, then the values will be summed into the matrix location.
7. It is recommended to use double field format for the bulk data entries to allow the input to use the most number of significant digits.

6.7.67 DPHASE

Data Entry: **DPHASE** - Dynamic Load Phase Lead

Description: This data is used in conjunction with the RLOAD1 or RLOAD2 data entries, and defines the phase lead term, θ, in the equation of the loading function.

Format:

1	2	3	4	5	6	7	8	9	10
DPHASE	SID	P	C	TH	P	C	TH		

Example:

1	2	3	4	5	6	7	8	9	10
DPHASE	4	21	6	2.3	8	6	7.2		

Field	Information	Description
2	SID	Identification number of DPHASE set (Integer > 0).
3, 6	P	**GRID** or **SPOINT** identification number (Integer > 0).
4, 7	C	Component number (Integer 1-6 for grid point; blank or 0 for scalar point).
5, 8	TH	Phase lead, θ, (in degrees) for the designated coordinate (Real).

Remarks:

1. One or two dynamic load time delays may be defined on a single entry.
2. Refer to **RLOAD1** or **RLOAD2** data for the formulas which define the manner in which the phase lead, θ, is used.
3. **DAREA** data should exist for the same grid point and component.

6.7.68 EIGR

Data Entry: **EIGR** - Eigenvalue Calculation Data.

Description: Defines data needed to perform real eigenvalue analysis, used in frequency or buckling analysis.

Format:

1	2	3	4	5	6	7	8	9	10
EIGR	SID	METHOD	V1	V2		ND	NORM		
+	MODE1	G1	C1		MODE2	G2	C2		
+	MODE3	G3	C3		-etc.-				

Example 1:

1	2	3	4	5	6	7	8	9	10
EIGR	1	SMS		250.0					

Example 2:

1	2	3	4	5	6	7	8	9	10
EIGR	1	LAN				2	POINT		
+	1	100	1		2	100	2		

Field	Information	Description
2	SID	Set identification number (Unique integer > 0).
3	METHOD	Method of eigenvalue calculation. One of the words SUB, LAN or SMS or Integer > 0. See remarks 2, 3 and 8.
4,5	V1,V2	Value range of interest. Used if METHOD = LAN, SMS or integer (Real, V1 < V2 or blank). If referenced by a frequency loadcase, values are frequencies. If referenced by a buckling loadcase, values are eigenvalues. For the SMS method, V2 must be nonblank.
7	ND	Desired number of eigenvalues to be calculated (Integer > 0 or Blank). See remark 9.
8	NORM	Mode shape normalization scheme. In frequency calculations one of the norms MAX, MAX0, MASS, POINT or blank (Default = MASS. In buckling calculations, one of the norms MAX, STIFF, POINT or blank (Default = STIFF).
2, 6, ...	MODEi	Mode number used for point norm only (Integer > 0 or Blank).
3, 7, ...	GI	**GRID** or **SPOINT** identification number used for point norm only (Integer > 0 or Blank).

6

4, 8, ...	CI	Component number in the general coordinate system used for point norm only (1 ≤ Integer ≤ 6 or Blank)

Remarks:

1. In frequency calculation loadcases, the mass matrix can be either a consistent mass matrix or a lumped (diagonal) mass matrix. The default is a coupled mass matrix. To specify a lumped mass matrix, set the parameter COUPMASS = NO in a PARAM data statement.
2. METHOD = SUB specifies the subspace iteration method. LAN specifies the Lanczos method. SMS specifies the SMS approximation method. If METHOD is integer, the value must match *mthid* defined on an **EIGMETHOD** executive control command. SMS and external EIGMETHOD are only applicable to frequency calculation loadcases.
3. METHOD=LAN, SMS or integer can only be used at installations that have the sparse matrix solver. For METHOD = LAN, SMS or integer, the analysis parameter SOLVER must be 1.
4. In frequency calculation loadcases, damping is not considered.
5. If the eigenvector component specified in POINT normalization is zero, then the component of MAX NORM is used for normalization and a warning message is printed in the output file.
6. In frequency calculation loadcases, it is recommended to avoid MAX norm when eigenvector components are selected with DRESP1 data. The reason for this is that this norm is, in general, discontinuous because the component that has the maximum value will often change as the design changes.
7. If POINT normalization is selected and normalization information is not given for some of the modes, then MAX0 normalization is used for those unspecified modes.
8. In frequency calculation loadcases, when Guyan reduction is requested via ASET data, the method requested is ignored. Instead, the modified Givens method is used.

6

9. In all buckling loadcases, and frequency calculation loadcases using METHOD=SUB, the default for ND is 1. If METHOD=LAN, SMS or integer, modes are calculated according to the table below. Modes are ordered by value (absolute value for buckling) of the eigenvalue, from smallest to largest.

V1	V2	ND	Modes Calculated
V1	V2	ND	At most ND modes between V1 and V2
V1	V2	Blank	All modes between V1 and V2
V1	Blank	ND	First ND modes greater than or equal to V1
V1	Blank	Blank	First one mode greater than or equal to V1
Blank	V2	ND	At most ND modes less than or equal to V2
Blank	V2	Blank	All modes less than or equal to V2
Blank	Blank	ND	First ND modes
Blank	Blank	Blank	First one mode

6.7.69 EIGRL

Data Entry: **EIGRL -** Eigenvalue Calculation Data.

Description: Defines data needed to perform real eigenvalue analysis using the Lanczos or SMS method . This is used in frequency or buckling analysis.

Format:

1	2	3	4	5	6	7	8	9	10
EIGRL	SID	V1	V2	ND		MAXSET		NORM	

Example 1:

1	2	3	4	5	6	7	8	9	10
EIGRL	1		250.						

Example 2:

1	2	3	4	5	6	7	8	9	10
EIGRL	1	0.0		10				MAX	

Field	Information	Description
2	SID	Set identification number (Unique integer > 0).
3,4	V1,V2	Value range of interest. (Real, V1 < V2 or blank). If referenced by a frequency loadcase, values are frequencies. If referenced by a buckling loadcase, values are eigenvalues.
5	ND	Desired number of eigenvalues to be calculated. (Integer > 0 or Blank). See remark 7.
7	MAXSET	Block size for Lanczos calculations (0<Integer≤15 or Blank). See remark 6.
9	NORM	Mode shape normalization scheme. In frequency calculations one of the norms MAX, MASS or blank (Default = MASS). In buckling calculations, one of the norms MAX, STIFF or blank (Default = STIFF).

Remarks:

1. In frequency calculation loadcases, the mass matrix can be either a consistent mass matrix or a lumped (diagonal) mass matrix. The default is a coupled mass matrix. To specify a lumped mass matrix, set the parameter COUPMASS = NO in a PARAM data statement.
2. This data can only be used at installations that have the sparse matrix solver. The analysis parameter SOLVER must be 1.
3. In frequency calculation loadcases, damping is not considered.

4. In frequency calculation loadcases, it is recommended to avoid MAX norm when eigenvector components are selected with DRESP1 data. The reason for this is that this norm is, in general, discontinuous because the component that has the maximum value will often change as the design changes.
5. In frequency calculation loadcases, when Guyan reduction is requested via ASET data, the Lanczos method is not used. Instead, the modified Givens method is used.
6. The MAXSET value can be used to fine tune the performance of the Block Lanczos algorithm. It should be greater than or equal to the highest multiplicity of any eigenvalue. A value of 2 generally gives better performance than 1. Values of 10 or greater typically only give good performance on machines with a large I/O bandwidth (e.g., Cray). Memory limitations may reduce the value used. The default is 6.
7. In buckling loadcases the default for ND is 1. Modes are calculated according to the table below. Modes are ordered by value (absolute value for buckling) of the eigenvalue, from smallest to largest.

V1	V2	ND	Modes Calculated
V1	V2	ND	At most ND modes between V1 and V2
V1	V2	Blank	All modes between V1 and V2
V1	Blank	ND	First ND modes greater than or equal to V1
V1	Blank	Blank	First one mode greater than or equal to V1
Blank	V2	ND	At most ND modes less than or equal to V2
Blank	V2	Blank	All modes less than or equal to V2
Blank	Blank	ND	First ND modes
Blank	Blank	Blank	First one mode

8. In frequency calculation loadcases, when V2 is specified, the SMS method is used. Otherwise, the Lanczos method is used. To force use of the Lanczos method, use the **EIGR** entry with METHOD = LAN.

6.7.70 ENDDATA

Data Entry: **ENDDATA** - Mark the End of Bulk Data.

Description: Marks the end of the Bulk Data section of the input file.

Format:

1	2	3	4	5	6	7	8	9	10
ENDDATA									

Example:

1	2	3	4	5	6	7	8	9	10
ENDDATA									

Remarks:

1. An ENDDATA entry is required.
2. Any entries following the ENDDATA entry are ignored.

6

6.7.71 ERPPNL

Data Entry: **ERPPNL** - Equivalent Radiated Power Panel.

Description: Defines panels for calculation of equivalent radiated power.

Format:

1	2	3	4	5	6	7	8	9	10
ERPPNL	NAME1	SID1	NAME2	SID2	NAME3	SID3	NAME4	SID4	
+	NAME5	SID5	-etc.-						

Example:

1	2	3	4	5	6	7	8	9	10
ERPPNL	LEFT	1	TOP	7					

Field	Information	Description
2, 4, 6, 8	NAMEi	Unique panel name (Character).
3, 5, 7, 9	SIDi	Element set defined by a **SET** solution control entry (Integer > 0). See remark 2.

Remarks:

1. Calculated equivalent radiated power values can be output using the **ERP** solution control command.
2. The element sets listed on ERPPNL can only contain **CQUAD4**, **CTRIA3**, **CQUAD8** and/or **CTRIA6** elements that reference **PSHELL** or **PCOMP** properties.
3. See **Equivalent Radiated Power** (Sec. 2.16.2) for a discussion of the equivalent radiated power calculation.

6

6.7.72 FINDEX

Data Entry: **FINDEX** - Failure Index Equation Data.

Description: Defines a user-supplied equation for composite failure index calculation as a function of stresses and material constants.

Format:

1	2	3	4	5	6	7	8	9	10
FINDEX	EQID	EQUATION							
+	EQUATION (Cont.)								

Example 1:Hill Failure

1	2	3	4	5	6	7	8	9	10
FINDEX	2	F(S1,S2,S12,XT,XC,YT,YC,S,F12) =(S1/XC)**2-S1*S2/XC**2+(S2/YC)**2							
+	+(S12/S)**2+(XT+YT+F12)*0.0								

Example 2: Hill Failure and Layered equation.

1	2	3	4	5	6	7	8	9	10
FINDEX	3	F(S1,S2,S12,XT,XC,YT,YC,S,F12) = (S1/XC)**2; G=S1*S2/XC**2;							
+	H = F-G+(S2/YC)**2+(S12/S)**2+(XT+YT+F12)*0.0								

Field	Information	Description
2	EQID	Unique equation identification number with respect to FINDEX and FINDEXN. (Integer > 0).
3-...	EQUATION	Equation. See Remark 10.

Remarks:

1. FINDEX equations can be referenced by field 6 of **PCOMP** or **PCOMPG** data entries.
2. The EQUATION consists of the collection of data in fields 3 through 10 on the first entry and fields 2 through 10 on the continuations. The boundaries between these fields are not recognized and the collection is treated as if it were one field.
3. EQUATION may contain embedded blanks.
4. See **Equation Utility** (p. 271) in the Design Manual (Volume 2) for a discussion of the user defined equation feature.
5. The arithmetic operators in order of precedence are: `**, *, /, +, -`. The following relational operators may also be used: `==, /=, <, <=, >, >=`. The relational operators result in a value of 1.0 if the relation they are testing is true, or a value of 0.0 if the relation is false. The relational operators have lower precedence than the arithmetic operators.

6. The following table lists the available intrinsic functions:

Function	Description
ABS(x)	**Absolute value of x**
ACOS(x)	**Inverse cosine of x (result in radians)**
ACOSH(x)	**Inverse hyperbolic cosine of x**
ASIN(x)	**Inverse sine of x (result in radians)**
ASINH(x)	**Inverse hyperbolic sine of x**
ATAN(x)	**Inverse tangent of x (result in radians)**
ATAN2(y,x)	**Inverse tangent of y/x (result in radians, -π to π)**
ATANH(x)	**Inverse hyperbolic tangent of x**
ATANH2(y,x)	**Inverse hyperbolic tangent of y/x**
AVG(x1,x2,...,xn)	**Average: (x1+x2+...+xn)/n**
COS(x)	**Cosine of x (x in radians)**
COSH(x)	**Hyperbolic cosine of x**
COTAN(x)	**Cotangent of x (x in radians)**
DIM(x,y)	**Maximum of (0, x-y)**
EXP(x)	**e raised to power x**
INT(x)	**Convert x to integer**
LOG(x)	**Natural (base e) logarithm of x**
LOG10(x)	**Common (base 10) logarithm of x**
LOGX(x,y)	**Base x logarithm of y**
MAX(x1,x2,...,xn)	**Maximum of (x1, x2, ..., xn)**
MIN(x1,x2,...,xn)	**Minimum of (x1, x2, ..., xn)**
MOD(x,y)	**Remainder of x/y**
PI(x)	**π times x**
RSS(x1,x2,...,xn)	**Square root of sum of squares: SQRT(x1**2 + x2**2 + ... + xn**2)**
SIGN(x,y)	**Absolute value of x times sign of y**
SIN(x)	**Sine of x (x in radians)**
SINH(x)	**Hyperbolic sine of x**
SQRT(x)	**Square root of x**
SSQ(x1,x2,...,xn)	**Sum of squares: (x1**2 + x2**2 + ... + xn**2)**
SUM(x1,x2,...,xn)	**Sum: (x1 + x2 + ... + xn)**

Function	Description
TAN(x)	**Tangent of x (x in radians)**
TANH(x)	**Hyperbolic tangent of x**

7. The relational operators and the functions ABS, DIM, INT, MAX, MIN, MOD, and SIGN should be used with caution, because they can create discontinuities in the function or its derivative. Such discontinuities can cause poor convergence behavior of the optimizer.
8. The maximum number of characters that can be used to define the function names and each of the arguments is 31.
9. Layered equations can be used by separating the equations with semi colons (;). The first equation contains the argument list for all the equations. Equations may reference the value of one or more preceding equations. The value of the last equation is the result of the FINDEX and is the value used by the program as the failure index.
10. The left-hand side of the first equation must include exactly 9 arguments, enclosed in parentheses. Every argument must be referenced at least once in the right-hand side or in a subsequent layered equation. To ignore arguments, add 0.0 times argument name. The arguments passed into the equations are as follows:

Argument Position	**Recommended Name**	**Definition**
1	S1	Stress in fiber direction
2	S2	Stress in transverse direction
3	S12	Shear stress
4	XT	Fiber direction stress limit in tension (ST from MAT1, XT or E1*XT from MAT8)
5	XC	Fiber direction stress limit in compression (SC from MAT1, XC or E1*XC from MAT8)
6	YT	Transverse direction stress limit in tension (ST from MAT1, YT or E2*YT from MAT8)
7	YC	Transverse direction stress limit in compression (SC from MAT1, YC or E2*YC from MAT8)
8	S	Shear stress limit (SS from MAT1, S or G12*S from MAT8)
9	F12	MAT8 interaction term (0.0 for MAT1, F12 from MAT8)

11. MAT8 entries may include either stress or strain limits. If strain limits are given, stress limits are calculated by scaling with E1, E2 and G12 as indicated in the above table.

6.7.73 FINDEXN

Data Entry: **FINDEXN** - Failure Index Equation Data.

Description: Defines a user-supplied equation for composite failure index calculation as a function of strains and material constants.

Format:

1	2	3	4	5	6	7	8	9	10
FINDEXN	EQID	EQUATION							
+	EQUATION (Cont.)								

Example 1:Max Strain Failure

1	2	3	4	5	6	7	8	9	10
FINDEXN	2	F(S1,S2,S12,XT,XC,YT,YC,S,F12) =MAX((S1>0.0)*S1/XT+(S1<0.0)*S1/XC,							
+	(S2>0.0)*S2/YT+(S2<0.0)*S2/YC,ABS(S12/S))+0.0*F12								

Field	Information	Description
2	EQID	Unique equation identification number with respect to FINDEX and FINDEXN. (Integer > 0).
3-...	EQUATION	Equation. See Remark 10

Remarks:

1. FINDEXN equations can be referenced by field 6 of **PCOMP** or **PCOMPG** data entries.
2. The EQUATION consists of the collection of data in fields 3 through 10 on the first entry and fields 2 through 10 on the continuations. The boundaries between these fields are not recognized and the collection is treated as if it were one field.
3. EQUATION may contain embedded blanks.
4. See **Equation Utility** (p. 271) in the Design Manual (Volume 2) for a discussion of the user defined equation feature.
5. The arithmetic operators in order of precedence are: `**, *, /, +, -`. The following relational operators may also be used: `==, /=, <, <=, >, >=`. The relational operators result in a value of 1.0 if the relation they are testing is true, or a value of 0.0 if the relation is false. The relational operators have lower precedence that the arithmetic operators.
6. The following table lists the available intrinsic functions:

Function	Description
ABS(x)	**Absolute value of x**
ACOS(x)	**Inverse cosine of x (result in radians)**

6

Function	Description
ACOSH(x)	**Inverse hyperbolic cosine of x**
ASIN(x)	**Inverse sine of x (result in radians)**
ASINH(x)	**Inverse hyperbolic sine of x**
ATAN(x)	**Inverse tangent of x (result in radians)**
ATAN2(y,x)	**Inverse tangent of y/x (result in radians, -π to π)**
ATANH(x)	**Inverse hyperbolic tangent of x**
ATANH2(y,x)	**Inverse hyperbolic tangent of y/x**
AVG(x1,x2,...,xn)	**Average: (x1+x2+...+xn)/n**
COS(x)	**Cosine of x (x in radians)**
COSH(x)	**Hyperbolic cosine of x**
COTAN(x)	**Cotangent of x (x in radians)**
DIM(x,y)	**Maximum of (0, x-y)**
EXP(x)	**e raised to power x**
INT(x)	**Convert x to integer**
LOG(x)	**Natural (base e) logarithm of x**
LOG10(x)	**Common (base 10) logarithm of x**
LOGX(x,y)	**Base x logarithm of y**
MAX(x1,x2,...,xn)	**Maximum of (x1, x2, ..., xn)**
MIN(x1,x2,...,xn)	**Minimum of (x1, x2, ..., xn)**
MOD(x,y)	**Remainder of x/y**
PI(x)	**π times x**
RSS(x1,x2,...,xn)	**Square root of sum of squares:** **SQRT(x1**2 + x2**2 + ... + xn**2)**
SIGN(x,y)	**Absolute value of x times sign of y**
SIN(x)	**Sine of x (x in radians)**
SINH(x)	**Hyperbolic sine of x**
SQRT(x)	**Square root of x**
SSQ(x1,x2,...,xn)	**Sum of squares: (x1**2 + x2**2 + ... + xn**2)**
SUM(x1,x2,...,xn)	**Sum: (x1 + x2 + ... + xn)**
TAN(x)	**Tangent of x (x in radians)**
TANH(x)	**Hyperbolic tangent of x**

7. The relational operators and the functions ABS, DIM, INT, MAX, MIN, MOD, and SIGN should be used with caution, because they can create discontinuities in the function or its derivative. Such discontinuities can cause poor convergence behavior of the optimizer.
8. The maximum number of characters that can be used to define the function names and each of the arguments is 31.
9. Layered equations can be used by separating the equations with semi colons (;). The first equation contains the argument list for all the equations. Equations may reference the value of one or more preceding equations. The value of the last equation is the result of the FINDEXN and is the value used by the program as the failure index.
10. The left-hand side of the first equation must include exactly 9 arguments, enclosed in parentheses. Every argument must be referenced at least once in the right-hand side or in a subsequent layered equation. To ignore arguments, add 0.0 times argument name. The arguments passed into the equations are as follows:

Argument Position	Recommended Name	Definition
1	S1	Strain in fiber direction
2	S2	Strain in transverse direction
3	S12	Shear strain
4	XT	Fiber direction strain limit in tension (ST/E from MAT1, XT or XT/E1 from MAT8)
5	XC	Fiber direction strain limit in compression (SC/E from MAT1, XC or XC/E1 from MAT8)
6	YT	Transverse direction strain limit in tension (ST/E from MAT1, YT or YT/E2 from MAT8)
7	YC	Transverse direction strain limit in compression (SC/E from MAT1, YC or YC/E2 from MAT8)
8	S	Shear strain limit (SS/G from MAT1, S or S/G12 from MAT8)
9	F12	MAT8 interaction term (0.0 for MAT1, F12 from MAT8)

11. MAT8 entries may include either stress or strain limits. If stress limits are given, strain limits are calculated by scaling with E1, E2 and G12 as indicated in the above table.

6.7.74 FORCE

Data Entry: **FORCE** - Static Load.

Description: Defines a static load at a grid point by specifying a vector.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
FORCE	42	5		120.0	1.0	0.0	0.0		

Field	Information	Description
2	SID	Load set identification number (Integer> 0).
3	G	**GRID** identification number (Integer> 0).
4	CID	Coordinate system identification number Integer $\geq$ 0 or blank. Default = 0).
5	F	Scale factor (Real).
6-8	N1, N2, N3	Components of vector measured in coordinate system defined by CID (Real or blank. Default = 0.0; must have at least one nonzero component).

Remarks:

1. The static load applied to grid point G is given by $\bar{f} = F\,\overline{N}$ where $\overline{N}$ is the vector defined in fields 6, 7 and 8.
2. Load sets can be selected in the Solution Control Section (**LOAD**=SID) or through **RLOAD1**, **RLOAD2** or **LOAD** bulk data entries.
3. A CID of zero or blank references the basic coordinate system.

6.7.75 FORCE1

Data Entry: **FORCE1** - Static Load, Alternate Form 1.

Description: Used to define a static load by specification of a value and two grid points which determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE1	SID	G	F	G1	G2				

Example:

1	2	3	4	5	6	7	8	9	10
FORCE1	3	9	7.15	10	21				

Field	Information	Description
2	SID	Load set identification number (Integer > 0).
3	G	**GRID** identification number (Integer > 0).
4	F	Value of load (Real).
5, 6	G1,G2	Grid point identification numbers (Integer > 0. $G1 \neq G2$).

Remarks:

1. The direction of the force is determined by the vector from G1 to G2.
2. Load sets can be selected in the Solution Control Section (**LOAD**=SID) or through **RLOAD1**, **RLOAD2** or **LOAD** bulk data entries.
3. In shape optimization, the direction of the load is updated if G1 and/or G2 move.

6

6.7.76 FREQ

Data Entry: **FREQ** - Frequency List

Description: Defines a set of frequencies to be used in the solution of frequency response problems.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ	SID	F	F	F	F	F	F	F	
+	F	F	F	-etc.-					

Example:

1	2	3	4	5	6	7	8	9	10
FREQ	3	2.98	3.05	19.7	21.3	25.5	28.8	31.3	
+	29.2	22.4	19.3						

Field	Information	Description
2	SID	Frequency set identification number (Integer > 0).
3, 4, ...	F	Frequency value (Real > 0 or blank).

Remarks:

1. The units for the frequencies are cycles per unit time.
2. Frequency sets must be selected by the Solution Control data **FREQUENCY** = SID.
3. All FREQ, FREQ1 and FREQ2 data with the same frequency set identification number will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if $|f_N - f_{N-1}| < 10^{-5}|f_{MAX} - f_{MIN}|$.

6.7.77 FREQ1

Data Entry: **FREQ1 -** Frequency List, Alternate Form 1

Description: Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, frequency increment and the number of increments desired.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ1	SID	F1	DF	NDF					

Example:

1	2	3	4	5	6	7	8	9	10
FREQ1	3	2.9	.5	12					

Field	Information	Description
2	SID	Frequency set identification number (Integer > 0).
3	F1	First frequency in set (Real > 0).
4	DF	Frequency increment (Real > 0).
5	NDF	Number of frequency increments (Integer > 0).

Remarks:

1. The units for the frequency F1 and frequency increment DF are cycles per unit time.
2. The frequencies defined by this data entry are given by
   f_i = F1 + (i-1)DF   i = 1,(NDF + 1).
3. Frequency sets must be selected by the Solution Control data **FREQUENCY** = SID.
4. All FREQ, FREQ1 and FREQ2 data with the same frequency set identification number will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if $|f_N - f_{N-1}| < 10^{-5}|f_{MAX} - f_{MIN}|$.

6

6.7.78 FREQ2

Data Entry: **FREQ2 -** Frequency List, Alternate Form 2

Description: Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, ending frequency and the number of logarithmic increments desired.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ2	SID	F1	F2	NF					

Example:

1	2	3	4	5	6	7	8	9	10
FREQ2	7	20.0	100.0	39					

Field	Information	Description
2	SID	Frequency set identification number (Integer > 0).
3	F1	First frequency in set (Real > 0).
4	F2	Maximum frequency (Real > 0, F2 > F1).
5	NF	Number of logarithmic intervals (Integer > 0).

Remarks:

1. The units for the frequencies F1 and F2 are cycles per unit time.
2. The frequencies defined by this data entry are given by

$$f_i = \mathrm{F1} \cdot e^{(i-1)d} \qquad i = 1, 2, ..., (\mathrm{NF}+1), \text{ where } d = \frac{1}{\mathrm{NF}} ln \frac{\mathrm{F2}}{\mathrm{F1}}.$$

3. Frequency sets must be selected by the Solution Control data **FREQUENCY** = SID.
4. All FREQ, FREQ1 and FREQ2 data with the same frequency set identification number will be used. Duplicate frequencies will be ignored. f_N and f_{N-1} are considered duplicated if $|f_N - f_{N-1}| < 10^{-5}|f_{MAX} - f_{MIN}|$.

6

6.7.79 GENEL

Data Entry: **GENEL** - General Element

Description: Defines a element for providing stiffness or flexibility matrices.

Format:

1	2	3	4	5	6	7	8	9	10
GENEL	EID		GI1	CI1	GI2	CI2	GI3	CI3	
+	GI4	CI4	GI5	CI5	-etc.-				
+	“UD”		GD1	CD1	GD2	CD2	GD3	CD3	
+	GD4	CD4	GD5	CD5	-etc.-				
+	“K” or “Z”	KZ11	KZ21	KZ31	-etc.-		KZ22	KZ32	
+	-etc.-		KZ33	KZ43	-etc.-				
+	“S”	S11	S12	-etc.-		S21	-etc.-		

Example 1:

1	2	3	4	5	6	7	8	9	10
GENEL	99		2	1	2	2	2	3	
+	2	4	2	5	2	6			
+	UD		3	1	3	2	3	3	
+	3	4	3	4	3	4			
+	Z	3.20E-6	0.0	0.0	0.0	0.0	0.0	1.22E-6	
+	0.0	0.0	0.0	4.38E-7	1.33E-7	0.0	-2.21E-7	0.0	
+	5.18E-7	0.0	0.0	4.67E-6	0.0	4.54E-6			

Example 2: (Double Field Format)

1	2	3	4	5	6	7	8	9	10
GENEL	99		2	1	2	2	2	3	
+	2	4	2	5	2	6			
+	UD		3	1	3	2	3	3	
+	3	4	3	4	3	4			
*	Z		3.20E-6		0.0		0.0		
*	0.0		0.0		0.0		1.22E-6		
*	0.0		0.0		0.0		4.38E-7		
*	1.33E-7		0.0		-2.21E-7		0.0		
*	5.18E-7		0.0		0.0		4.67E-6		
*	0.0		4.54E-6						

6

Field	Information	Description
2	EID	Unique element identification number (Integer > 0).
4, 5 6, 7 etc.	GI1, CI1	Identification number of coordinates in the UI list, in sequence corresponding to the [K], [Z] and [S] matrices. GIi are **GRID** numbers, and CIi are the component numbers. If an **SPOINT** is given, the component number is zero (Integer).
2	"UD"	Character string which indicates the start of data belonging to UD.
4, 5 6, 7 etc.	GD1, CD1	Identification number of coordinates in the UD list, in sequence corresponding to the [K], [Z] and [S] matrices. GDi are GRID numbers, and CDi are the component numbers. If an SPOINT is given, the component number is zero (Integer).
2	"K" or "Z"	Character string which indicates the start of data belonging to [K] or [Z] (Real).
3, 4, etc.	KZij	Values of [K] or [Z] matrix ordered by columns from the diagonal, according to the UI list (Real)
2	"S"	Character string which indicates the start of data belonging to [S]
3, 4, etc.	Sij	Values of the [S] matrix ordered by rows according to the UD list (Real).

Remarks:

1. When the stiffness matrix, [K] is input, the number of significant digits should be the same for all terms.
2. If the stiffness matrix is provided, then the following equation is used:

$$\begin{Bmatrix} f_i \\ f_d \end{Bmatrix} = \begin{bmatrix} K & -KS \\ -S^TK & S^TKS \end{bmatrix} \begin{Bmatrix} u_i \\ u_d \end{Bmatrix}$$

3. If the flexibility matrix is provided, then the following equation is used:

$$\begin{Bmatrix} u_i \\ f_d \end{Bmatrix} = \begin{bmatrix} Z & S \\ -S^T & 0 \end{bmatrix} \begin{Bmatrix} f_i \\ u_d \end{Bmatrix}$$

where

$$\{u_i\} = \begin{Bmatrix} u_{i1} \\ u_{i2} \\ \dots \\ u_{im} \end{Bmatrix} \qquad \{u_d\} = \begin{Bmatrix} u_{d1} \\ u_{d2} \\ \dots \\ u_{dm} \end{Bmatrix}$$

6

$$[KZ] = \begin{bmatrix} KZ_{11} & . & . & SYM \\ KZ_{21} & KZ_{22} & . & . \\ . & . & . & . \\ KZ_{m1} & . & . & KZ_{mm} \end{bmatrix}$$ and $[KZ] = [K]$ or $[Z]$ and $[KZ]^T = [KZ]$

$$[S] = \begin{bmatrix} S_{11} & S_{12} & . & S_{1m} \\ . & S_{22} & . & . \\ . & . & . & . \\ S_{m1} & . & . & S_{mm} \end{bmatrix}$$

The required input is the $\{u_i\}$ list and the lower portion of [K] or [Z]. Additional input may include the $\{u_d\}$ list and [S]. If [S] is input, $\{u_d\}$ must also be input. If $\{u_d\}$ is input but [S] is not, [S] is internally calculated. In this case, $\{u_d\}$ must contain six and only six degrees of freedom.

The forms shown above for both stiffness and flexibility approaches assume that the element is a free body whose rigid body motions are defined by $\{u_i\} = [S]\{u_d\}$. In the example:

$$u_i = \begin{Bmatrix} u_2 \\ v_2 \\ w_2 \\ \theta_{x2} \\ \theta_{y2} \\ \theta_{z2} \end{Bmatrix} \qquad u_d = \begin{Bmatrix} u_3 \\ v_3 \\ w_3 \\ \theta_{x3} \\ \theta_{y3} \\ \theta_{z3} \end{Bmatrix}$$

$$Z = \begin{bmatrix} 3.2\times10^{-6} & & & & & \\ 0.0 & 1.22\times10^{-6} & & & SYM & \\ 0.0 & 0.0 & 1.33\times10^{-7} & & & \\ 0.0 & 0.0 & 0.0 & 5.18\times10^{-7} & & \\ 0.0 & 0.0 & -2.21\times10^{-7} & 0.0 & 4.67\times10^{-6} & \\ 0.0 & 4.38\times10^{-7} & 0.0 & 0.0 & 0.0 & 4.54\times10^{-6} \end{bmatrix}$$

Because S is not provided and u_d is provided, S is calculated automatically by *GENESIS*.

6.7.80 GRAV

Data Entry: **GRAV** - Gravity Vector.

Description: Used to define gravity vectors for use in determining gravity loading for the structural model.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
GRAV	2	4	32.2	0.0	-1.0	0.0			

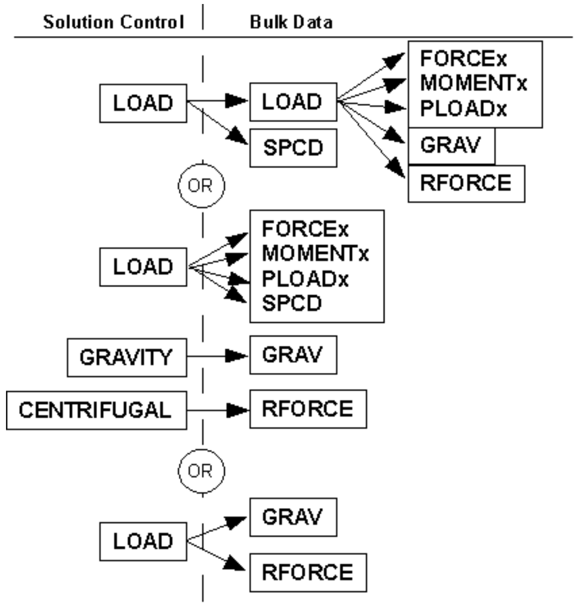
Field	Information	Description
2	SID	Gravity load set identification number (Integer > 0).
3	CID	Coordinate system identification number (Integer $\geq$ 0, or Blank. Default = 0).
4	G	Gravity vector scale factor (Real).
5-7	N1,N2,N3	Gravity vector components (Real or blank. Default = 0.0; must have at least one nonzero component).

Remarks:

1. The gravity vector is defined by $\bar{g} = G\,\bar{N}$. The direction of $\bar{g}$ is the direction of free fall.
2. There can be only one GRAV entry for each gravity load set identification number.
3. Gravity load set identification numbers must be unique with respect to load sets defined by **FORCE**, **FORCE1**, **MOMENT**, **MOMENT1**, **PLOAD1**, **PLOAD2**, **PLOAD4**, **PLOAD5**, **PLOADA** and **PLOADX1**.
4. Either the **GRAVITY** solution control command or the **LOAD** bulk data entry can be used to combine gravity loading with concentrated and pressure loads sets.

6

5. Gravity load sets can be selected in static load cases with either the Solution Control command **GRAVITY**=SID or with the Solution Control command **LOAD**=SID.



6. A gravity load can be selected in frequency response load cases by listing SID on **RLOAD1** or **RLOAD2** bulk data.
7. A CID of 0 or blank references the basic coordinate system.
8. The load generated by this entry can be printed with an **OLOAD** request in the Solution Control Data.
9. Gravity loads are internally created for all elements with non-zero mass. The only exception is for the scalar elements, CMASS1/2, that are connected to scalar points, in which case the gravity load cannot be defined. When CMASS1/2 elements are connected with grid points, the gravity load is calculated and projected in the direction of the degree of freedom's component.

6

6.7.81 GRDSET

Data Entry: **GRDSET** - Grid Point Default

Description: Defines default options for fields 3, 7 and 8 of all GRID data.

Format:

1	2	3	4	5	6	7	8	9	10
GRDSET		CP				CD	PS		

Example:

1	2	3	4	5	6	7	8	9	10
GRDSET		12				51	3456		

Field	Information	Description
3	CP	Identification number of coordinate system in which the location of grid points are defined (Integer $\geq$ 0 or blank). See Remark 4.
7	CD	Identification number of coordinate system in which the displacements, degrees of freedom, constraints, and solution vectors of the grid point are defined (Integer $\geq$ 0 or blank).
8	PS	Permanent single-point constraints associated with grid point (any combination of the digits 1-6 with no embedded blanks) (Integer $\geq$ 0 or blank). See Remark 4.

Remarks:

1. The contents of fields 3, 7 and 8 of this data are assumed for the corresponding fields of any **GRID** data whose field 3, 7 and 8 are blank. If any of these fields on the GRID data are blank, the default option defined by this data occurs for that field. If no permanent single-point constrains are desired or one of the coordinate systems is basic, the default may be overridden on the GRID entry by making of fields 3, 7, and/or 8 zero (rather than blank). Only one GRDSET data statement may appear in the user's Input Data Section.
2. The primary purpose of this data is to minimize the burden of preparing data for problems with a large amount of repetition (e.g., two-dimensional pinned joint problems).
3. At least one of the entries CP, CD, or PS must be nonzero.
4. CP=0 indicates that the grid points are defined in the basic coordinate system. Similarly, CD=0 indicates that the displacements, degrees of freedom, constraints and solution vectors are defined in the basic coordinate system.
5. Only one GRDSET statement may appear in the input data file.

6

6.7.82 GRID

Data Entry: **GRID** - Grid Point.

Description: Defines the location of a geometric grid point of the structural model, the direction of its displacement, and its permanent single-point constraints. Also defines move limits on the grid for use in design optimization.

Format:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS		
+	MV	X1L	X1U	X2L	X2U	X3L	X3U	XR	

Example:

1	2	3	4	5	6	7	8	9	10
GRID	205	2	100.0	205.5	101.8	5	246		
+	1	-0.1.	0.1	-0.2	0.2			0.3	

Field	Information	Description
2	ID	Grid point identification number (Integer > 0).
3	CP	Identification number of coordinate system in which the location of grid points are defined (Integer $\geq$ 0 or blank). See Remark 1. See **GRDSET** data for default options for field 3. See Remark 12 for replication options.
4-6	X1,X2,X3	Location of the grid point in coordinate system CP (Real or blank). See Remark 12 for replication options.
7	CD	Identification number of coordinate system in which the displacements, degrees of freedom, constraints, and solution vectors of the grid point are defined (Integer $\geq$ -1or blank). See Remark 3. See GRDSET data for default options for field 7. For CD = -1, see Remark 11. See Remark 12 for replication options.
8	PS	Permanent single-point constraint associated with grid point (any combination of the digits 1-6 with no imbedded blanks) (0 $\leq$ Integer $\leq$ 654321 or blank. See GRDSET data for default options for field 8). See Remark 12 for replication options.
2	MV	Move parameter for this coordinate (0,1 or blank. Default=0). 0 means the value of coordinate X1 through X3 will be limited to the values of X1L through X3U throughout the design process. 1 means the change in coordinate X1 through X3 will be limited to the values of X1L through X3U during each design cycle.
3	X1L	Lower bound on X1 measured in coordinate system CP (real or blank if MV=0), (real < 0.0 or blank if MV = 1), (ignored if blank).

6

4	X1U	Upper bound on X1 measured in coordinate system CP (real or blank if MV=0), (real > 0.0 or blank if MV = 1), (ignored if blank).
5	X2L	Lower bound on X2 measured in coordinate system CP (real or blank if MV=0), (real < 0.0 or blank if MV = 1), (ignored if blank).
6	X2U	Upper bound on X2 measured in coordinate system CP (real or blank if MV=0), (real > 0.0 or blank if MV = 1), (ignored if blank).
7	X3L	Lower bound on X3 measured in coordinate system CP (real or blank if MV=0), (real < 0.0 or blank if MV = 1), (ignored if blank).
8	X3U	Upper bound on X3 measured in coordinate system CP (real or blank if MV=0), (real > 0.0 or blank if MV = 1), (ignored if blank).
9	XR	Bound on the resultant move of this grid point (used only with MV=1), (real > 0.0, ignored if blank).

Remarks:

1. The meaning of X1, X2 and X3 as well as X1L through X3U depends on the type of coordinate system, CP, as follows (also, see CORDi entry descriptions):

TYPE	X1	X2	X3
Rectangular	X	Y	Z
Cylindrical	r	θ (degrees)	Z
Spherical	ρ	θ (degrees)	ϕ (degrees)

   See Coordinate input data for a definition of the coordinate system terminology.
2. The collection of all CD coordinate systems defined by all GRID entries is called the General Coordinate System. All degrees of freedom, constraints, and solution vectors are expressed in the General Coordinate System.
3. If any of the fields 3, 7 or 8 (CP, CD or PS) of this entry are blank (not zero), then the values from the GRDSET data entry are used, if it exists. A zero in field 3 or 7 indicates the basic coordinate system.
4. The continuation data is only meaningful for design optimization. This data is ignored if only an analysis is performed. This data is necessary only if the specified coordinate must lie within the limits defined on this line of data.
5. The parameter MV is used to define whether the move limits on the GRID are absolute (physical) bounds on the GRID (MV = 0), or are changes that are allowed on the GRID during each design cycle (MV = 1). MV applies only to fields 3 through 8 of the continuation data.
6. The parameter XR in field 9 of the continuation data defines a limit on the total $(\sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2})$ grid change. This parameter is used only with MV=1. If XR is nonblank, the CP coordinate system must be rectangular.

7. If one or more of fields 3-8 of the continuation data are non blank, move limits are imposed according to the value of MV. If field 9 of the continuation data is also non blank, the resultant move limit is imposed in addition to any individual move limits.
8. The GRID move limits are applied in the CP coordinate system. If this coordinate system is rectangular, spherical or cylindrical, the move limits will be applied in the units of that coordinate system (i.e. length for the X, Y, Z, r and ρ directions and degrees for the ϕ and θ directions).
9. The resultant grid move limit (XR) cannot be applied to a grid whose location is defined in a non-rectangular coordinate system.
10. All grid point identification numbers must be unique with respect to all other grid and scalar points.
11. If CD is -1, this GRID entry defines a fluid grid. Fluid grids can only connect to **CHEX20**, **CHEXA**, **CPENTA**, **CPYRA** and **CTETRA** to define fluid elements. Fluid grids may be also be referenced by **SPC**, **SPC1**, **MPC**, **DAREA**, **DELAY**, **DPHASE** and/or **SPCD** to support fluid loading and boundary conditions. All other entries that reference grids (e.g., FORCE, CROD, RBE2, etc.) may not reference fluid grids.
12. The CP, X1, X2, X3, CD and PS fields may optionally replicate the corresponding field from the preceding GRID entry. In this case, the field may be one of “=”, “==”, “**x*” or “*(*x*)”, where *x* is a real or integer value. A value of “=” means duplication of the field from preceding entry. A value of “==” means duplication the field and all trailing fields from preceding entry. In this case, all fields after the field with “==” must be blank, or a fatal error will occur. A value of “**x*” or “*(*x*)” means to take the value of the field from the preceding entry and add *x*, where *x* should be a real number for real fields or an integer for integer fields and can be positive or negative. Note that to define a fluid grid, the CD field must be a literal -1. If CD = -1 is obtained from replication, a fatal error will occur.

6

6.7.83 INCLUDE

Data Entry: **INCLUDE**

Description: Select an external file that contains bulk data statements.

Format:

```
INCLUDE 'file name'
```

Alternate Format:

```
INCLUDE = file name
```

Examples:

```
INCLUDE 'D035.DVS'
INCLUDE = D035.DVS
```

Option	Meaning
file name	External file name. The user must provide the file name according to the machine installation.

Remarks:

1. The INCLUDE data can be anywhere in the bulk data.
2. Multiple INCLUDE data are allowed in the bulk data.
3. The file name is limited to 240 characters.
4. If the quoted format is used, and the line does not end with a quote character, additional lines will be read until the closing quote is found. Leading and trailing spaces on continued and continuation lines are discarded.

6.7.84 LOAD

Data Entry: **LOAD** - Load Set Combination.

Description: Defines a new load set as a linear combination of loads from FORCE, FORCE1, MOMENT, MOMENT1, PLOAD1, PLOAD2, PLOAD4, PLOAD5, PLOADA, PLOADX1, GRAV and RFORCE entries.

Format:

1	2	3	4	5	6	7	8	9	10
LOAD	SID	S	S1	SID1	S2	SID2	S3	SID3	
+	S4	SID4	-etc.-						

Example:

1	2	3	4	5	6	7	8	9	10
LOAD	10	1.0	4.0	1	2.0	2	-1.0	3	
+	5.0	4							

Field	Information	Description
2	SID	Load set identification number (Integer > 0).
3	S	Overall scale factor (Real or Blank. Default = 1.0).
4, 6, 8 2, 4, ..	Si	Scale factor for load set SIDi (Real).
5, 7, 9 3, 5, ..	SIDi	Load set ID used by **FORCE**, **FORCE1**, **MOMENT**, **MOMENT1**, **PLOAD1**, **PLOAD2**, **PLOAD4**, **PLOAD5**, **PLOADA** and **PLOADX1** entries or a **GRAV** entry or an **RFORCE** entry (Integer > 0).

Remarks:

1. The load vector is created as follows:

$$F = S\sum_{i} S_i F_{SIDi}$$

2. Load sets can be selected in the Solution Control Section (**LOAD** = SID) or through **RLOAD1** or **RLOAD2** bulk data entries.
3. The load set IDs (SIDi) must be unique.
4. At most, one RFORCE set ID may be referenced. RFORCE may not be referenced if LOAD is selected by RLOAD1 or RLOAD2.
5. Load sets defined on other LOAD entries may not be referenced.

6. In general, the load set ID defined by a LOAD entry should not also be used by FORCEi/MOMENTi/PLOADx data. To allow this, the analysis parameter **LOADCK** must be set to 0.

6.7.85 MAT1

Data Entry: **MAT1** - Material Property Definition.

Description: Defines the material properties for linear, temperature-independent, isotropic materials.

Format:

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
+	ST	SC	SS						

Example:

1	2	3	4	5	6	7	8	9	10
MAT1	10	1.+7		0.3					
+									

Field	Information	Description
2	MID	Unique material identification number (Integer > 0). See Remark 1.
3	E	Young's modulus (Real or blank).
4	G	Shear modulus (Real or blank).
5	NU	Poisson's ratio (-1.0 < Real $\leq$ 0.5 or blank).
6	RHO	Mass density (Real $\geq$ 0.0 or blank).
7	A	Thermal expansion coefficient (Real or blank).
8	TREF	Thermal expansion reference temperature (Real or blank).
9	GE	Structural damping coefficient (Real or blank)
2-4	ST,SC,SS	Stress limits for tension, compression and von Mises or shear. Used for automatic generation of stress constraints and composite ply failure index calculations. See remark 9. (Real or blank).

Remarks:

1. The material identification number may be the same on **MAT4** and **MAT5**, but must be unique with respect to other **MAT1**, **MAT2**, **MAT3**, **MAT8**, **MAT9**, **MAT10** and **MAT11** data.
2. The mass density, RHO, will be used to automatically compute mass for structural elements.

3. Weight density may be used to field 6 if the value 1/g is entered on the PARAM entry **WTMASS**, where g is the acceleration of gravity.
4. Either E or G must be specified (i.e., non blank).
5. If any one of E, G, or NU is blank, it will be computed to satisfy the identity E = 2(1+NU)G; otherwise, values supplied by the user will be used. This calculation is only made for initial values of E, G, and NU.
6. If E and NU or G and NU are both blank, they will both be given the value 0.0.
7. Implausible data on one or more MAT1 data will result in a warning message. Implausible data is defined as any of E < 0.0 or G < 0.0 or NU > 0.5 or NU < 0.0 or $|1 - E/2(1 + NU)G| > 0.001$ (except for cases covered by Remark 6)
8. It is strongly recommended that only two of the three values E, G, and NU be input.

<table>
<tr><th>ELEMENT TYPE</th><th>E</th><th>NU</th><th>G</th></tr>
<tr><td>ROD</td><td>Extension</td><td>Not Used</td><td>Not Used</td></tr>
<tr><td>BAR</td><td>Extension and Bending</td><td>Used in Beam Library</td><td>Torsion Transverse Shear</td></tr>
<tr><td>BEAM</td><td>Extension and Bending</td><td>Not Used</td><td>Torsion Transverse Shear</td></tr>
<tr><td>SHEAR</td><td>Extension</td><td colspan="2">Shear</td></tr>
<tr><td>QUAD4
TRIA3
QUAD8
TRIA6</td><td colspan="3">Membrane and Bending</td></tr>
<tr><td>TRIAX6
HEXA
PENTA
TETRA
HEX20
PYRA</td><td colspan="3">All Terms</td></tr>
</table>

9. For BAR elements and plate/shell elements that reference PCOMP/PCOMPG data, the shear limits (SS) should be the maximum shear stress. This limit (SS) can be used only for the design element library elements supplied by VR&D, and for composite ply failure index calculations. For plate/shell elements that reference PSHELL data, for axisymmetric elements and solid elements, it (SS) should be the maximum von Mises stress. For SHEAR elements, the shear limit (SS) is the absolute value of the largest shear stress.
10. To obtain the structural damping coefficient, GE, multiply the critical damping ratio, C/C_0, by 2.0.
11. If referenced by PCOMP/PCOMPG data, GE and TREF are ignored.

Meaning of ST, SC and SS in MAT1 data

	ST	SC	SS
ROD	Tension	Compression	- - -
BAR	Tension	Compression	- - -
BEAM	- - -	- - -	- - -
QUAD4 (PSHELL), TRIA3 (PSHELL)	- - -	- - -	von Mises
QUAD8 (PSHELL), TRIA6 (PSHELL)	- - -	- - -	von Mises
PCOMP/PCOMPG: HILL	X	Y	S
PCOMP/PCOMPG: HOFFMAN, TSAI-WU, STRN	XT, YT	XC, YC	S
SHEAR	- - -	- - -	Max shear
TRIAX6	- - -	- - -	von Mises
HEXA, HEX20, PENTA, TETRA, PYRA	- - -	- - -	von Mises

6.7.86 MAT2

Data Entry: **MAT2** - Material Property Definition.

Description: Defines the material properties for linear, temperature-independent, anisotropic materials for two-dimensional elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT2	MID	G11	G12	G13	G22	G23	G33	RHO	
+	A1	A2	A12	T0	GE	ST	SC	SS	

Example:

1	2	3	4	5	6	7	8	9	10
MAT2	12	6.75+6	1.15+6	1.43+6	2.9+6	0.4+6	1.79+6		
+	4.58-6	10.32-6	-5.428-6	125.0					

Field	Information	Description
2	MID	Unique material identification number (Integer > 0). See Remark 1.
3-8	Gij	The material property matrix (Real or blank. Default = 0.0).
9	RHO	Mass density (Real $\geq$ 0 or blank. Default = 0.0).
2-4	Ai	Thermal expansion coefficient vector (Real or blank. Default = 0.0).
5	T0	Thermal expansion reference temperature (Real or blank. Default = 0.0).
6	GE	Structural damping coefficient (Real or blank).
7-9	ST,SC,SS	Stress limits for tension, compression and von Mises or shear. Used for automatic generation of stress constraints. (Real or blank).

Remarks:

1. The material identification numbers may be the same on **MAT4** and **MAT5**, but must be unique with respect to other **MAT1**, **MAT2**, **MAT3**, **MAT8**, **MAT9**, **MAT10** and **MAT11** data.
2. The mass density, RHO, will be used to automatically compute mass for all structural elements.
3. Weight density may be used to field 9 if the value 1/g is entered on the PARAM entry **WTMASS**, where g is the acceleration of gravity.

6

4. The convention for the Gij in fields 3 through 8 are represented by the matrix relationship:

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} & G_{13} \\ G_{12} & G_{22} & G_{23} \\ G_{13} & G_{23} & G_{33} \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix} - (T - T_0) \begin{Bmatrix} A_1 \\ A_2 \\ A_{12} \end{Bmatrix} \right)$$

5. Unlike the MAT1 data, data from the MAT2 data is used directly, without adjustment of equivalent E, G, or NU values.
6. If the material is referenced by the MID3 entry of the **PSHELL** data, the G13, G23 and G33 must be blank.
7. To obtain the structural damping coefficient, GE, multiply the critical damping ratio, C/C_0, by 2.0.

6.7.87 MAT3

Data Entry: **MAT3** - Material Property Definition.

Description: Defines the material properties for linear, temperature-independent, orthotropic materials by the TRIAX6 elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT3	MID	Er	Eθ	Ez	NUrθ	NUθz	NUzr	RHO	
+			Gzr	Ar	Aθ	Az	TREF	GE	

Example:

1	2	3	4	5	6	7	8	9	10
MAT3	12	6.75+6	1.15+6	1.43+6	.3	.25	.27	1.0-5	
+			2.5+6	1.0-4	1.0-4	1.1-4	68.5	.23	

Field	Information	Description
2	MID	Unique material identification number (Integer > 0). See Remark 1.
3-5	Er, Eθ Ez	Young's modulus in the r, θ and z directions, respectively, (Real > 0.0).
6-8	NUrθ, NUθz, NUzr	Poison's Ratios (coupled strain ratios in the rθ, zθ and zr directions respectively (Real)
9	RHO	Mass density (Real $\geq$ 0 or blank. Default = 0.0).
4	Gzr	Shear modulus (Real $\geq$ 0)
5-7	Ar, Aθ, Az	Thermal expansion coefficients (Real or blank).
8	TREF	Thermal expansion reference temperature (Real or blank. Default = 0.0).
9	GE	Structural damping coefficient (Real or blank).

Remarks:

1. The material identification numbers must be unique with respect to the collection of **MAT1**, **MAT2**, **MAT3**, **MAT8**, **MAT9**, **MAT10** and **MAT11** data.
2. All seven numbers of Er, Eθ, Ez, NUrθ, NUθz, NUzr and Gzr must be present.
3. A warning message will be printed if any of NUrθ or NUθz has an absolute value greater then 1.0.
4. MAT3 materials may only be referenced by **PAXIS** data.

5. The mass density, RHO, will be used to automatically compute mass for the CTRIAX6 elements.
6. The r-axis lies along the material axis (see drawing with **CTRIAX6** data). The θ-axis lies in the azimuthal direction. The z-axis is normal to both.
7. The relationship is:

$$\begin{Bmatrix} \varepsilon_r \\ \varepsilon_\theta \\ \varepsilon_z \\ \gamma_{zr} \end{Bmatrix} = \begin{bmatrix} 1/Er & -NU\theta r/E\theta & -NUzr/Ez & 0 \\ -NUr\theta/Er & 1/E\theta & -NUz\theta/Ez & 0 \\ -NUrz/Er & -NU\theta z/E\theta & 1/Ez & 0 \\ 0 & 0 & 0 & 1/Gzr \end{bmatrix} \begin{Bmatrix} \sigma_r \\ \sigma_\theta \\ \sigma_z \\ \tau_{rz} \end{Bmatrix}$$

$$+ (T - TREF) \begin{Bmatrix} Ar \\ A\theta \\ Az \\ 0 \end{Bmatrix}$$

8. To obtain the damping coefficient, GE, multiply the critical damping ratio, C/C_0 by 2.0.

6

6.7.88 MAT4

Data Entry: **MAT4** - Thermal Material Property Definition.

Description: Defines the thermal material properties for temperature-independent, isotropic materials.

Format:

1	2	3	4	5	6	7	8	9	10
MAT4	MID	K							

Example:

1	2	3	4	5	6	7	8	9	10
MAT4	110	0.65							

Alternate Format:

1	2	3	4	5	6	7	8	9	10
MAT4	MID			RHO	H				

Example:

1	2	3	4	5	6	7	8	9	10
MAT4	210			27.3	0.77				

Field	Information	Description
2	MID	Unique material identification number (Integer > 0). See Remark 1.
3	K	Thermal conductivity or convective film coefficient (Real > 0.0).
5	RHO	Mass density (Real $\geq$ 0.0 or Blank).
6	H	Convective film coefficient (Real > 0.0). See Remark2.

Remarks:

1. The material identification number must be the same as that of a **MAT1**, **MAT2**, **MAT3**, **MAT8**, **MAT9** or **MAT11** data, and **must** be unique with respect to other **MAT4** and **MAT5** data.
2. The alternate format is only used with **PCONV** to provide the convective film coefficient H.

6

6.7.89 MAT5

Data Entry: **MAT5** - Thermal Material Property Definition.

Description: Defines the thermal material properties for temperature-independent, anisotropic materials.

Format:

1	2	3	4	5	6	7	8	9	10
MAT5	MID	KXX	KXY	KXZ	KYY	KYZ	KZZ		

Example:

1	2	3	4	5	6	7	8	9	10
MAT5	110	0.65			0.093		0.025		

Field	Information	Description
2	MID	Unique material identification number (Integer > 0). See Remark 2.
3	KXX	Thermal conductivity (Real > 0.0).
4	KXY	Thermal conductivity (Real or blank).
5	KXZ	Thermal conductivity (Real or blank for PSOLID; blank for PSHELL, PCOMP or PAXIS).
6	KYY	Thermal conductivity (Real > 0.0).
7	KYZ	Thermal conductivity (Real or blank for PSOLID; blank for PSHELL, PCOMP or PAXIS).
8	KZZ	Thermal conductivity (Real > 0.0 for PSOLID; blank for PSHELL, PCOMP or PAXIS).

Remarks:

1. The thermal conductivity matrix has the form:

$$K = \begin{bmatrix} KXX & KXY & KXZ \\ KXY & KYY & KYZ \\ KXZ & KYZ & KZZ \end{bmatrix}$$

2. The material identification number must be the same as that of a **MAT1**, **MAT2**, **MAT3**, **MAT8**, **MAT9** or **MAT11** data, and must be unique with respect to other **MAT4** and **MAT5** data.
3. If MAT5 data is referenced by **PAXIS**, **PSHELL**, **PCOMP** or **PCOMPG** data, KXZ, KZY and KZZ must be blank.
4. If MAT5 data is referenced by PAXIS data, then the radial conductivity component should be input in the KXX and the axial conductivity component in the KYY. For example, the thermal conductivity matrix has the form

$$K = \begin{bmatrix} KXX & KXY & 0.0 \\ KXY & KYY & 0.0 \\ 0.0 & 0.0 & 0.0 \end{bmatrix}$$

6.7.90 MAT8

Data Entry: **MAT8** - Material Property Definition.

Description: Defines the material property for an orthotropic material for plate/shell elements (TRIA3, QUAD4, TRIA6 and QUAD8).

Format:

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	v12	G12	G1, Z	G2, Z	RHO	
+	A1	A2	TREF	XT	XC	YT	YC	S	
+	GE	F12	STRN						

Example: (Glass Epoxy)

1	2	3	4	5	6	7	8	9	10
MAT8	2	7.8+6	2.6+6	0.25	1.3+6				
+									
+									

Field	Information	Description
2	MID	Material ID (Integer > 0). See remark 1.
3	E1	Modulus of elasticity in longitudinal direction (also defined as fibre direction or 1-direction). ($\mathrm{Real} \neq 0.0$).
4	E2	Modulus of elasticity in lateral direction (also defined as matrix direction or 2-direction). ($\mathrm{Real} \neq 0.0$).
5	v12	Poisson's ratio ($-\varepsilon_2/\varepsilon_1$ for uniaxial loading in 1 - direction). Note that ν_{21} ($\nu_{21} = -\varepsilon_1/\varepsilon_2$ for uniaxial loading in the 2 - direction) is related to ν_{12}, E_1, E_2, by the relation $\nu_{12}E_2 = \nu_{21}E_1$. (Real).
6	G12	Inplane shear modulus (Real > 0.0).
7	G1, Z	Transverse shear modulus for the 1-Z plane (Real $\geq$ 0.0 or Blank. Default = 0.0).
8	G2, Z	Transverse shear modulus for the 2-Z plane (Real $\geq$ 0.0 or Blank. Default = 0.0).
9	RHO	Mass density (Real $\geq$ 0.0 or blank. Default = 0.0)
2	A1	Thermal expansion coefficient in 1-direction (Real or blank)
3	A2	Thermal expansion coefficient in 2-direction (Real or blank)

6

4	TREF	Thermal expansion reference temperature (Real or blank)
5	XT	Allowable stress in tension in the longitudinal direction. Used for composite ply failure index calculations for elements that reference PCOMP data. (Real or blank)
6	XC	Allowable stress in compression in the longitudinal direction. Used for composite ply failure index calculations for elements that reference PCOMP data. (Real or blank) (Default value for XC is -XT)
7	YT	Allowable stress in tension in the transverse direction. Used for composite ply failure index calculations for elements that reference PCOMP data. (Real or blank)
8	YC	Allowable stress in compression in the transverse direction. Used for composite ply failure index calculations for elements that reference PCOMP data. (Real or blank) (Default value for YC is -YT)
9	S	Allowable shear or von Mises stress for inplane (Real or blank). Used for composite ply index failure calculations. Also used for automatic generation of stress constraints.
2	GE	Structural damping coefficient (Real or blank).
3	F12	Interaction term for calculation of Tsai-Wu failure index for composite plies (Real or blank. Default is 0.0).
4	STRN	Real = 1.0 if XT, XC, YT, YC and S are strain allowables. Blank if XT, XC, YT, YC and S are stress allowables.

Remarks:

1. The material identification numbers may be the same for **MAT4** and **MAT5**, but must be unique with respect to other **MAT1**, **MAT2**, **MAT3**, **MAT8**, **MAT9**, **MAT10** and **MAT11** data.
2. The mass density, RHO, will be used to automatically compute mass for structural elements.
3. Weight density may be used to field 9 if the value 1/g is entered on the PARAM entry **WTMASS**, where g is the acceleration of gravity.
4. If the material is referenced by the MID3 entry of the **PSHELL** data, then G1,Z and G2,Z must be nonzero.
5. An approximate value for G1,Z and G2,Z can be taken as G12.
6. To obtain the structural damping coefficient, GE, multiply the critical damping ratio, C/C_0, by 2.0.
7. If referenced by **PCOMP** or **PCOMPG** data, TREF and GE are ignored.
8. XT, XC, YT, YC and S are required for composite failure index calculations.

6.7.91 MAT9

Data Entry: **MAT9** - Material Property Definition.

Description: Defines the material properties for linear, temperature-independent, anisotropic materials for 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:

1	2	3	4	5	6	7	8	9	10
MAT9	30	5.9+3						5.9+3	
+					5.9+3				
+	4.85+3			4.85+3		4.85+3	3.6	6.4-6	
+	6.4-6					140.0			

Field	Information	Description
2	MID	Unique material identification number (Integer > 0). See Remark 1.
3,...	Gij	Elements of the 6x6 symmetric material property matrix (Real or blank. Default = 0.0).
8	RHO	Mass density (Real $\geq$ 0.0 or blank. Default = 0.0).
9,2-6	Ai	Thermal expansion coefficient vector (Real or blank).
7	TREF	Thermal expansion reference temperature (Real or blank).
8	GE	Structural damping coefficient (Real or blank).

Remarks:

1. The material identification numbers may be the same for **MAT4** and **MAT5**, but must be unique with respect to other **MAT1**, **MAT2**, **MAT3**, **MAT8**, **MAT9** , **MAT10** and **MAT11** data.
2. The mass density RHO will be used to automatically compute mass for structural elements.
3. Weight density may be used to field 8 if the value 1/g is entered on the PARAM entry **WTMASS**, where g is the acceleration of gravity.

4. The fourth continuation entry is not required.
5. The subscripts 1 through 6 refer to x, y, z, xy, yz, zx.

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} & G_{13} & G_{14} & G_{15} & G_{16} \\ & G_{22} & G_{23} & G_{24} & G_{25} & G_{26} \\ & & G_{33} & G_{34} & G_{35} & G_{36} \\ & & & G_{44} & G_{45} & G_{46} \\ & \text{SYM} & & & G_{55} & G_{56} \\ & & & & & G_{66} \end{bmatrix} \left(\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - (T - T_{Ref}) \begin{Bmatrix} A_1 \\ A_2 \\ A_3 \\ A_4 \\ A_5 \\ A_6 \end{Bmatrix} \right)$$

6. A4, A5 and A6 must be zero for CHEXA, CPENTA, CPYRA and CTETRA elements.
7. To obtain the structural damping coefficient, GE, multiply the critical damping ratio, C/C_0, by 2.0.
8. Automatic stress constraints will not be generated for solid elements that reference MAT9 data.

6

6.7.92 MAT10

Data Entry: **MAT10** - Fluid Material Property Definition.

Description: Defines the fluid material properties for elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT10	MID	BULK	RHO	C					

Example:

1	2	3	4	5	6	7	8	9	10
MAT10	50	0.78	0.026						

Field	Information	Description
2	MID	Unique material identification number (Integer > 0). See Remark 1.
3	BULK	Bulk modulus (Real > 0.0 or blank). See Remark 3.
4	RHO	Mass density (Real > 0.0 or blank). See Remark 3.
5	C	Speed of sound (Real > 0.0 or blank). See Remark 3.

Remarks:

1. The material identification numbers must be unique with respect to other **MAT1**, **MAT2**, **MAT3**, **MAT8**, **MAT9**, **MAT10** and **MAT11** data.
2. MAT10 may only be referenced by a **PSOLID** that has “PFLUID” in the FCTN field. See this entry description for details.
3. Two of the three values BULK, RHO, and C must be specified, (i.e., non blank). If any one of BULK, RHO, or C is blank, it will be computed to satisfy the relation BULK = C^2 RHO. If all three values are specified and do not follow the relation, a fatal error will occur.
4. It is strongly recommended that only two of the three values BULK, RHO, and C be input.
5. The mass density, RHO, will be used to automatically compute mass for all fluid elements.
6. Weight density may be used in field 4 if the value 1/g is entered on the PARAM entry **WTMASS**, where g is the acceleration of gravity.

6.7.93 MAT11

Data Entry: **MAT11** - Material Property Definition.

Description: Defines the material properties for linear, temperature-independent, orthotropic materials for solid elements (CHEXA, CPENTA, CTETRA and CPYRA).

Format:

1	2	3	4	5	6	7	8	9	10
MAT11	MID	E1	E2	E3	v12	v13	v23	G12	
+	G13	G23	RHO	A1	A2	A3	TREF	GE	

Example: (Glass Epoxy)

1	2	3	4	5	6	7	8	9	10
MAT8	2	7.8+6	2.6+6	2.6+6	0.25	0.25	0.25	1.3+6	
+	1.3+6	1.3+6							

Field	Information	Description
2	MID	Material ID (Integer > 0). See remark 1.
3	E1	Modulus of elasticity in the 1-direction (also defined as the fibre or x-direction). ($\mathrm{Real} \neq 0.0$).
4	E2	Modulus of elasticity in the 2-direction (also defined as the lateral or y-direction). ($\mathrm{Real} \neq 0.0$).
5	E3	Modulus of elasticity in the 3-direction (also defined as the thickness or z-direction). ($\mathrm{Real} \neq 0.0$).
6	v12	Poisson's ratio ($-\varepsilon_2/\varepsilon_1$ for uniaxial loading in 1 - direction). Note that ν_{21} $(\nu_{21} = -\varepsilon_1/\varepsilon_2 \text{ for uniaxial loading in the 2 - direction})$ is related to ν_{12}, E_1, E_2, by the relation $\nu_{12}E_2 = \nu_{21}E_1$. (Real).
7	v13	Poisson's ratio ($-\varepsilon_3/\varepsilon_1$ for uniaxial loading in 1 - direction). Note that ν_{31} $(\nu_{31} = -\varepsilon_1/\varepsilon_3 \text{ for uniaxial loading in the 3 - direction})$ is related to ν_{13}, E_1, E_3, by the relation $\nu_{13}E_3 = \nu_{31}E_1$. (Real).

6

MAT11

8	v23	Poisson's ratio ($-\varepsilon_3/\varepsilon_2$ for uniaxial loading in 2 - direction). Note that ν_{32} $(\nu_{32} = -\varepsilon_2/\varepsilon_3$ for uniaxial loading in the 3 - direction$)$ is related to ν_{23}, E_2, E_3, by the relation $\nu_{23}E_3 = \nu_{32}E_2$. (Real).
9	G12	Shear modulus for the 1-2 plane (Real > 0.0).
2	G13	Shear modulus for the 1-3 plane (Real > 0.0).
3	G23	Shear modulus for the 2-3 plane (Real > 0.0).
4	RHO	Mass density (Real $\geq$ 0.0 or blank. Default = 0.0)
5	A1	Thermal expansion coefficient in 1-direction (Real or blank. Default = 0.0)
6	A2	Thermal expansion coefficient in 2-direction (Real or blank. Default = 0.0)
7	A3	Thermal expansion coefficient in 3-direction (Real or blank. Default = 0.0)
8	TREF	Thermal expansion reference temperature (Real or blank. Default = 0.0)
9	GE	Structural damping coefficient (Real or blank. Default = 0.0).

Remarks:

1. The material identification numbers may be the same for **MAT4** and **MAT5**, but must be unique with respect to other **MAT1**, **MAT2**, **MAT3**, **MAT8**, **MAT9**, **MAT10** and **MAT11** data.
2. MAT11 generates equivalent MAT9 data. The sorted echo will show the data for the generated MAT9.
3. The mass density, RHO, will be used to automatically compute mass for structural elements.
4. Weight density may be used in field 4 if the value 1/g is entered on the PARAM entry **WTMASS**, where g is the acceleration of gravity.
5. To obtain the structural damping coefficient, GE, multiply the critical damping ratio, C/C_0, by 2.0.
6. Automatic stress constraints will not be generated for solid elements that reference MAT11 data.
7. For material stability, the following relationship must be true:
   $1 - \nu_{12}\nu_{21} - \nu_{23}\nu_{32} - \nu_{31}\nu_{13} - 2\nu_{12}\nu_{23}\nu_{31} > 0$
   If this relationship is violated, the material is implausible and fatal errors may occur when calculating element matrices.

6

6.7.94 MOMENT

Data Entry: **MOMENT** - Static Moment.

Description: Defines a static moment at a grid point by specifying a vector.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
MOMENT	10	7	2	50.0	0.0	0.0	1.0		

Field	Information	Description
2	SID	Load set identification number (Integer > 0).
3	G	Grid point identification number (Integer > 0).
4	CID	Coordinate system identification number (Integer $\geq$ 0, or Blank. Default = 0)
5	M	Scale factor (Real).
6-8	N1, N2, N3	Components of vector measured in coordinate system defined by CID (Real or blank; must have at least one nonzero component).

Remarks:

1. The static moment applied to grid point G is given by $\overline{m} = M\,\overline{N}$.
2. Load sets can be selected in the Solution Control Section (**LOAD**=SID) or through **RLOAD1**, **RLOAD2** or **LOAD** bulk data entries.
3. A CID of zero blank references the basic coordinate system.

6

6.7.95 MOMENT1

Data Entry: **MOMENT1 -** Static Moment, Alternate Form 1.

Description: Defines a static moment by specification of a value and two grid points which determine the direction

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT1	SID	G	M	G1	G2				

Example:

1	2	3	4	5	6	7	8	9	10
MOMENT1	10	9	1.35	12	8				

Field	Information	Description
2	SID	Load set identification number (Integer > 0).
3	G	Grid point identification number (Integer > 0).
4	M	Value of moment (Real).
5,6	G1,G2	Grid point identification numbers (Integer > 0. $G1 \neq G2$).

Remarks:

1. The direction of the moment vector is determined by the vector from G1 to G2.
2. Load sets can be selected in the Solution Control Section (**LOAD**=SID) or through **RLOAD1**, **RLOAD2** or **LOAD** bulk data entries.
3. The direction of the moment is updated if G1 and/or G2 move during shape optimization.

6

6.7.96 MPC

Data Entry: **MPC** - Multipoint Constraint.

Description: Defines a multipoint constraint equation of the form $\sum_{i} A_j u_j = 0$, where u_j represents degree of freedom C_j at grid point G_j.

Format:

1	2	3	4	5	6	7	8	9	10
MPC	SID	G1	C1	A1	G2	C2	A2		
+		G3	C3	A3	-etc.-				

Example 1: Used for a structural analysis load case (U2=W2)

1	2	3	4	5	6	7	8	9	10
MPC	1	2	1	1.0	2	3	-1.0		

Example 2: Used for a heat transfer analysis load case (T1=T2)

1	2	3	4	5	6	7	8	9	10
MPC	2	1		1.0	2		-1.0		

Field	Information	Description
2	SID	Set identification number (Integer > 0).
3,6	Gj	Identification number of **GRID** or **SPOINT** (Integer > 0).
4,7	Cj	Component number - any one of the digits 1-6, or blank for heat transfer analysis and scalar points ($1 \leq$ Integer $\leq$ 6 or blank).
5,8	Aj	Coefficient (Real; Aj must be nonzero or blank).

Remarks:

1. The first coordinate in the sequence is assumed to be the dependent coordinate.
2. Forces of multipoint constraint are not recovered.
3. Multipoint constraint sets must be selected in the Solution Control Section (**MPC**=SID) to be used.
4. Degrees of freedom specified as dependent may not be listed as dependent on rigid or interpolation elements or on other MPCs. Also, dependent degrees of freedom may not be listed on **SPC**, **SPC1**, **ASET2**, **ASET3** or **SUPORT1** entries.
5. Continuation data entries are optional.
6. The component numbers must be blank for MPC sets referenced by heat transfer loadcases and SPOINTs.

6

7. The A_j coefficients for independent degrees of freedom can be blank. A blank will produce the result that the corresponding independent degree of freedom will not affect the dependent degree of freedom.
8. In one MPC entry, the referenced grids must either all be structure grids or all be fluid grids. An MPC entry may not reference both fluid grids and structure grids.

6.7.97 MPCADD

Data Entry: **MPCADD** - Multi point constraint set combination.

Description: Defines a new MPC set as a union of MPC sets defined on MPC entries.

Format:

1	2	3	4	5	6	7	8	9	10
MPCADD	SID	SID1	SID2	SID3	SID4	SID5	SID6	SID7	
+	SID8	SID9	-etc.-						

Example:

1	2	3	4	5	6	7	8	9	10
MPCADD	20	1	2	9					

Field	Information	Description
2	SID	Unique multi-point constraint set Identification number (Integer > 0).
3, 4, 5, ...	SIDi	Multi-point constraint set ID used by MPC entries (Integer > 0).

Remarks:

1. Multi-point constraint sets can be selected in the solution control section (**MPC** = SID).
2. The multi-point constraint IDs (SIDi) must be unique.
3. MPC sets defined by other MPCADD entries may not be referenced.
4. The MPC set ID defined by MPCADD must not also be used by **MPC** bulk data entries.

6

6.7.98 NLPARM

Data Entry: **NLPARM** - Nonlinear static analysis parameters.

Description: Defines parameters to control nonlinear static analysis.

Format:

1	2	3	4	5	6	7	8	9	10
NLPARM	SID					MAXITER			

Example:.

1	2	3	4	5	6	7	8	9	10
NLPARM	1					15			

Field	Information	Description
2	SID	Unique nonlinear parameter set identification number (Integer > 0).
7	MAXITER	Maximum number of solver iterations allowed. (Integer > 0 or blank). Default = 25

Remarks:

1. Nonlinear parameter sets must be selected in the Solution Control Section (**NLPARM**=SID) to be used.

6

6.7.99 NSM

Data Entry: **NSM** - Nonstructural Mass.

Description: Defines a set of nonstructural mass to be added to listed properties or elements.

Format:

1	2	3	4	5	6	7	8	9	10
NSM	SID	TYPE	ID1	VALUE1	ID2	VALUE2	ID3	VALUE3	

Example 1: Assign a non structural mass per unit area of 0.1 to all elements associated to PSHELL 15.

1	2	3	4	5	6	7	8	9	10
NSM	1	PSHELL	15	0.10					

Field	Information	Description
2	SID	Nonstructural mass set identification number (Integer > 0).
3	TYPE	One of the words **PSHELL**, **PCOMP**, **PSHEAR**, **PBAR**, **PBARL**, **PBEAM**, **PBEAML**, **PROD** or ELEMENT.
4,6,8	IDi	Property or element identification number (Integer >0). See Remark 2.
5,7,9	VALUEi	Nonstructural mass per unit length or area (Real>0.0).

Remarks:

1. Nonstructural mass sets must be selected in the Solution Control section (**NSM**=SID) to be used.
2. If TYPE is ELEMENT, then the IDi fields are element identification numbers. Otherwise, they are property identification numbers. For element IDi, only **CBAR**, **CBEAM**, **CROD**, **CQUAD4**, **CQUAD8**, **CTRIA3**, **CTRIA6** or **CSHEAR** elements may be listed.

6.7.100 NSM1

Data Entry: **NSM1** - Nonstructural Mass, Alternate Form 1.

Description: Defines a set of nonstructural mass to be added to listed properties or elements.

Format:

1	2	3	4	5	6	7	8	9	10
NSM1	SID	TYPE	VALUE	ID1	ID2	ID3	ID4	ID5	
+	ID6	ID7	ID8	-etc.-					

Example 1: Assign a non strucural mass per unit area of 0.1 to all elements associated to PSHELLs 1, 3, 7, 8, 9, 11, 12 and 15.

1	2	3	4	5	6	7	8	9	10
NSM1	12	PSHELL	0.1	1	3	7	8	9	
+	11	12	15						

Alternate Format 1:

1	2	3	4	5	6	7	8	9	10
NSM1	SID	TYPE	VALUE	ID1	“THRU”	ID2	“BY”	N	

Example: Assign a non strucural mass per unit length of 0.3 to rod elements associated to PROD 101, 102, 103, 104 and 105.

1	2	3	4	5	6	7	8	9	10
NSM1	2	PROD	0.3	101	THRU	105			

Alternate Format 2:

1	2	3	4	5	6	7	8	9	10
NSM1	SID	TYPE	VALUE	“ALL”					

Example: Assign a non strucural mass per unit area of 0.4 to all shear elements.

1	2	3	4	5	6	7	8	9	10
NSM1	2	PSHEAR	0.4	ALL					

Field	Information	Description
2	SID	Set identification number (Integer > 0).
3	TYPE	One of the words **PSHELL**, **PCOMP**, **PSHEAR**, **PBAR**, **PBARL**, **PBEAM**, **PBEAML**, **PROD** or ELEMENT.
4	VALUE	Nonstructural mass per unit length or area (Real>0.0).

5, 6, ...	IDi	Property or element identification number (Integer >0). See Remark 2.

Remarks:

1. Nonstructural mass sets must be selected in the Solution Control section (**NSM**=SID) to be used.
2. If TYPE is ELEMENT, then the IDi fields are element identification numbers. Otherwise, they are property identification numbers. For element IDi, only **CBAR**, **CBEAM**, **CROD**, **CQUAD4**, **CQUAD8**, **CTRIA3**, **CTRIA6** or **CSHEAR** elements may be listed.
3. PBAR and PBARL properties are considered as the same type. The entry NSM1,10,PBAR,10.0,ALL will include all elements that reference PBAR and PBARL.
4. PBEAM and PBEAML properties are considered as the same type. The entry NSM1,10,PBEAM,10.0,ALL will include all elements that reference PBEAM and PBEAML.
5. As many continuation data as desired may appear when “THRU” or “ALL” are not used. When the words “ALL” or “THRU” exists, no continuation line is allowed.
6. When word “THRU” exists, ID1<ID2.

6.7.101 NSMADD

Data Entry: **NSMADD -** Nonstructural mass set combination

Description: Defines a nonstructural mass set as the union of listed nonstructural mass sets.

Format:

1	2	3	4	5	6	7	8	9	10
NSMADD	SID	SID1	SID2	SID3	SID4	SID5	SID6	SID7	
+	SID8	SID9	-etc.-						

Example:

1	2	3	4	5	6	7	8	9	10
NSMADD	10	1	3	7					

Field	Information	Description
2	SID	Unique nonstructural set Identification number (Integer > 0).
3, 4, 5, ...	SIDi	Nonstructural set ID used by **NSM**, **NSM1**, **NSML** or **NSML1** entries (Integer > 0).

Remarks:

1. Nonstructural mass sets must be selected in the Solution Control section (**NSM** = SID) to be used.
2. The nonstructural set IDs (SIDi) must be unique.
3. NSMADD cannot reference a set ID defined by another NSMADD.

6.7.102 NSML

Data Entry: **NSML** - Lumped Nonstructural Mass

Description: Defines a set of lumped nonstructural mass to be divided among elements.

Format:

1	2	3	4	5	6	7	8	9	10
NSML	SID	TYPE	ID1	VALUE1	ID2	VALUE2	ID3	VALUE3	

Example 1: Assign a non strucural mass per unit area of 0.1 to all elements associated to PSHELL 15

1	2	3	4	5	6	7	8	9	10
NSML	1	PSHELL	1	0.10					

Field	Information	Description
2	SID	Set identification number (Integer > 0).
3	TYPE	One of the words **PSHELL**, **PCOMP**, **PSHEAR**, **PBAR**, **PBARL**, **PBEAM**, **PBEAML**, **PROD** or ELEMENT.
4,6,8	IDi	Property or element identification number (Integer >0). See Remark 2.
5,7,9	VALUEi	Value of a lumped mass to be distributed among all the elements that refereces IDi (Real>0.0).

Remarks:

1. Nonstructural mass sets must be selected in the Solution Control section (**NSM**=SID) to be used.
2. If TYPE is ELEMENT, then the IDi fields are element identification numbers. Otherwise, they are property identification numbers. For element IDi, only **CBAR**, **CBEAM**, **CROD**, **CQUAD4**, **CQUAD8**, **CTRIA3**, **CTRIA6** or **CSHEAR** elements may be listed.
3. For **PSHELL**, **PCOMP** or **PSHEAR**, the NSM is calculated by dividing the lumped mass by the sum of the areas of all the element referenced by the property ID.
4. For **PBAR**, **PBARL**, **PBEAM**, **PBEAML** or **PROD**, the NSM is calculated by dividing the lumped mass by the sum of the lengths of all the element referenced by the property ID.
5. For ELEMENT, the NSM is calculated by dividing the lumped mass by the length or area, as appropriate, of the listed element.

6

6.7.103 NSML1

Data Entry: **NSML1** - Lumped Nonstructural Mass, Alternate Form 1.

Description: Defines a set of lumped nonstructural mass to be divided among elements.

Format:

1	2	3	4	5	6	7	8	9	10
NSML1	SID	TYPE	VALUE	ID1	ID2	ID3	ID4	ID5	
+	ID6	ID7	ID8	-etc.-					

Example 1: Distribute a lumped non structural mass of 25.0 among all elements that reference PSHELL 16.

1	2	3	4	5	6	7	8	9	10
NSML1	1	PSHELL	16	25.0					

Example 1: Distribute a lumped non strucural mass of 40.0 to all elements associated to PSHELLs 1, 3, 7, 8, 9, 11, 12 and 15.

1	2	3	4	5	6	7	8	9	10
NSML1	12	PSHELL	40.0	1	3	7	8	9	
+	11	12	15						

Alternate Format 1:

1	2	3	4	5	6	7	8	9	10
NSML1	SID	TYPE	VALUE	ID1	“THRU”	ID2	“BY”	N	

Example: Distribute a lumped non strucural mass of 20.0 to the rod elements associated to PROD 101, 102, 103, 104 and 105.

1	2	3	4	5	6	7	8	9	10
NSML1	2	PROD	20.0	101	THRU	105			

Alternate Format 2:

1	2	3	4	5	6	7	8	9	10
NSML1	SID	TYPE	VALUE	“ALL”					

Example: Distribute a non strucural mass of 30.0 to all shear elements.

1	2	3	4	5	6	7	8	9	10
NSML1	2	PSHEAR	30.0	ALL					

Field	Information	Description
2	SID	Set identification number (Integer > 0).
3	TYPE	One of the words **PSHELL**, **PCOMP**, **PSHEAR**, **PBAR**, **PBARL**, **PBEAM**, **PBEAML**, **PROD** or ELEMENT.
4	VALUE	A lumped mass value to be distributed among all the elements that references the listed IDi (Real>0.0).
5,6,...	IDi	Property or element identification number (Integer >0). See Remark 2.

Remarks:

1. Nonstructural mass sets must be selected in the Solution Control section (**NSM**=SID) to be used.
2. If TYPE is ELEMENT, then the IDi fields are element identification numbers. Otherwise, they are property identification numbers. For element IDi, only **CBAR**, **CBEAM**, **CROD**, **CQUAD4**, **CQUAD8**, **CTRIA3**, **CTRIA6** or **CSHEAR** elements may be listed. Length elements (CBAR, CBEAM, CROD) may not be mixed with area elements (CQUAD4, CQUAD8, CTRIA3, CTRIA6, CSHEAR).
3. For **PSHELL**, **PCOMP** or **PSHEAR**, the NSM is calculated by dividing the lumped mass by the sum of the areas of all the element referenced by all listed properties IDs.
4. For **PBAR**, **PBARL**, **PBEAM**, **PBEAML** or **PROD**, the NSM is calculated by dividing the lumped mass by the sum of the lengths of all the element referenced by all listed properties IDs.
5. For ELEMENT, the NSM is calculated by dividing the lumped mass by the sum of the lengths or areas, as appropriate, of the listed elements.
6. As many continuation data as desired may appear when “THRU” or “ALL” are not used. When the words “ALL” or “THRU” exists, no continuation line is allowed.
7. When word “THRU” exists, ID1<ID2.

6.7.104 PARAM

Data Entry: **PARAM** - Parameters.

Description: Specifies values for analysis parameters used in solution sequence.

Format:

1	2	3	4	5	6	7	8	9	10
PARAM	N	V1							

Example:

1	2	3	4	5	6	7	8	9	10
PARAM	AUTOSPC	YES							

Field	Information	Description
1	N	Parameter name (one to eight alphanumeric characters, the first of which is alphabetic).
2	V1	Parameter value; Real or Integer or Word.

Remarks:

1. The list of parameters that can be changed through the PARAM entry is given below.

6

Parameter for Linear Equation Solver Selection.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
SOLVER	1 2	Installation Dependent Usually 1	Determines the type of linear equation solver to use. If SOLVER = 1, use the sparse matrix solver. If SOLVER = 2, use the skyline solver. If your installation has the sparse matrix solver, the default is 1. Otherwise the default is 2 and SOLVER = 1 is not allowed. SOLVER must be 1 if the Lanczos method or the SMS method is used for eigenvalue calculations.

Parameters for Numerical Conditioning of the Linear Equation Solver.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
AUTOSPC	YES (or 1) NO (or -1)	NO	The AUTOSPC switch. If AUTOSPC = YES (or 1), then perform automatic constraining of degrees of freedom with little or no stiffness.
BAILOUT	YES (or 0) NO (or -1)	NO	The bailout switch. If BAILOUT = YES (or 0), then stop if the factor ratio of the triangularized matrix is greater than MAXRATIO. If BAILOUT = NO (or -1),the program will solve the problem with singularities.
EPZERO	Real > 0.0	1.0E-8	The AUTOSPC threshold value. Degrees of freedom with less than EPZERO stiffness will be constrained.
MAXRATIO	Real > 0.0	1.0E8	The maximum factor ratio; used to determine singularities in the stiffness matrix during decomposition.
PRGPST	YES NO	YES	Controls the printing of the table listing all degrees of freedom that are constrained by AUTOSPC.
RBE3SPC	YES NO FIX	NO	If RBE3SPC = NO, then any RBE3 element that has independent grids that are not also connected to some other element types (i.e., free independent dofs) will generate a fatal error. If RBE3SPC = YES, then RBE3 elements with free independent dofs will generate warning messages. If RBE3SPC = FIX, then RBE3 elements with free independent dofs will be ignored.
AUTOMSET	YES NO	NO	If AUTOMSET = YES, then the degrees of freedom made dependent by MPC and/or RBAR, RBE1, RBE2, RBE3, RROD, RSPLINE and BOLT elements will be determined automatically. In this case, the dependent d.o.f. specifications on the corresponding bulk data entries will be discarded, and consequently it is allowed to specify the same d.o.f. as dependent on multiple entries. AUTOMPC is an alias for this parameter.

BUSHSTIF	Real > 0.0	1.0E31	The maximum allowable stiffness for CBUSH elements. Any PBUSH K value greater than BUSHSTIF will be replaced by BUSHSTIF.
ELASSTIF	Real > 0.0	1.0E31	The maximum allowable stiffness for CELASi elements. Any PELAS K value greater than ELASSTIF will be replaced by ELASSTIF.

Parameters for Dynamic Analysis.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
G	Real ≥ 0.0	0.0	Structural damping coefficient applied to the global stiffness matrix.
GFL	Real ≥ 0.0	0.0	Damping coefficient applied to the fluid global “stiffness” matrix.
HFREQ	Real ≥ 0.0	1.0E30	Upper frequency cutoff for modes to include in the modal basis for modal dynamic frequency response analysis. If a loadcase has a **MODESELECT** solution control command, then this parameter will be ignored. This parameter can remove modes from the modal basis, and reduce the accuracy of the modal frequency response solution. If this parameter is used, the results should be carefully reviewed for adequacy.
KDAMP	-1 1	1	When KDAMP = 1, the modal damping is added to the viscous damping matrix. When KDAMP = -1, the modal damping is added to the complex stiffness matrix.
LFREQ	Real ≥ 0.0	0.0	Lower frequency cutoff for modes to include in the modal basis for modal dynamic frequency response analysis. If a loadcase has a **MODESELECT** solution control command, then this parameter will be ignored. This parameter can remove modes from the modal basis, and reduce the accuracy of the modal frequency response solution. If this parameter is used, the results should be carefully reviewed for adequacy.
RESVEC	YES NO	YES	If RESVEC = YES, then residual vectors are used with the normal mode shapes to increase the accuracy of modal dynamic analysis. To not use residual vectors set RESVEC = NO.

Parameters for Eigenvalue and Dynamic Analysis.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
COUPMASS	YES NO FULL	YES	A full consistent mass matrix is used if COUPMASS = FULL. A coupled mass matrix (linear combination of consistent and lumped) is used if COUPMASS = YES (or integer > -1). A lumped (diagonal) mass matrix is used if COUPMASS = NO (or integer < -1).
EPSEIG	Real > 0.0	1.0E-6	The convergence factor for the eigenvalues when subspace iteration is used.
ITMXSS	Integer > 0	50	Maximum number of subspace iteration cycles allowed.
LIMITLSF	YES NO	Installation Dependent	A value of NO indicates that there is no limit in file sizes for the Lanczos eigenvalue solver. A value of YES means that there is a limit. Normally, LIMITLSF is NO except on installations that have requested the limit to be YES.
SMSMAX	Integer ≥ 0	Installation Dependent	Maximum number of degrees of freedom in a supernode in SMS. Influences the incore memory requirements of SMS.
WTMASS	Real > 0.0	1.0	The conversion factor from weight units to mass units. If WTMASS is not 1.0, then all mass-related entries in the model (e.g., density) are assumed to be entered in weight units, and are scaled by WTMASS to get mass values.

Parameters Associated with the Finite Element Mesh.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
MSMOOTH	ON OFF	OFF	If MSMOOTH = ON, then mesh smoothing is performed on 2D planar surfaces and 3D elements.
MIDSIDE	0 1 2	0	If MIDSIDE=1, then each midside node of all second order elements is moved to the physical midpoint of its edge. If MIDSIDE=2, then each midside node of all second order elements is moved to the physical midpoint of its edge and perturbations applied to corner nodes of elements are averaged and applied to the corresponding midside nodes. If MIDSIDE=0, then no changes are made to midside nodes.

Parameter for Inertia Relief.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
IRTOL	Real > 0.0	1.0E-6	The maximum energy error ratio. Used to determine sufficient support on inertia relief loadcases.
INREL	0 -2	0	If INREL = -2, then default for the SUPORT solution control command is changed to AUTO.

Parameter for Element on the Fly.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
EOF	YES NO	Installation Dependent	Finite element matrices are calculated as they are needed, rather than stored. EOF = YES may reduce elapsed run time on computers with fast CPU's. This parameter can only be used if the analysis parameter SOLVER=1 (see above). If SOLVER=2, then EOF=NO.

Parameters for Model Resultants.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
GRDPNT	Integer ≥ -1	-1	The reference grid point for system moment of inertia calculations. A value of 0 will use the origin of the basic coordinate system as reference point. When GRDPNT = -1, the default, then system moments of inertia are not calculated.
PRTMAXIM	YES NO	YES	Controls the printing of the applied load, spc force and displacement translational maximum for each loadcase.
PRTRESLT	YES NO	YES	Controls the printing of the applied load and spc force resultants. The resultants are calculated in the basic coordinate system at the point defined with the analysis parameter GRDPNT. If GRDPNT is -1, then the resultants are calculated at the origin of the basic coordinate system.
SPCFTOL	Real ≥ 0.0	1.0E-8	SPCFORCE filter parameter. Reaction forces with a norm less than SPCFTOL will not be printed in the output file.

Parameters for Direct Matrix Input.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
CB2	Real	1.0	Scale factor to multipy DMIG matrices selected by B2GG.
CK2	Real	1.0	Scale factor to multipy DMIG matrices selected by K2GG.
CK42	Real	1.0	Scale factor to multipy DMIG matrices selected by K42GG.
CM2	Real	1.0	Scale factor to multipy DMIG matrices selected by M2GG.
CP2	Real	1.0	Scale factor to multipy DMIG matrices selected by P2G.

Parameter for CTRIA3/CQUAD4/CTRIA6/CQUAD8.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
THETA	Real Integer $\geq$ 0	0.0	Specifies a default value for THETA field of CTRIA3, CQUAD4, CTRIA6 and CQUAD8 entries. THETA is used for material property orientation specification. If Real, specifies the material property orientation angle in degrees. If Integer, the orientation of the material x-axis is along the projection on to the plane of the element of the x-axis of the coordinate system specified by the integer value.
T3SRM	0 1	1	Method for stress, strain and force recovery used in CTRIA3 elements. 0: User center of element coordinate system method 1: Use three points average method, default.
T6TOT3	0 1	0	1: CTRIA6 data entries will be read as CTRIA3 ignoring the midside grids. 0: Will not read CTRIA6 as CTRIA3.
BCSRCH	0 1	1	Flag to determine whether or not to account for the thickness of the element when calculating the normal distance for **BCPAIR** contact surfaces. 1: Thickness will be accounted for. Distance is from the top surface of the element. 0: Thickness is ignored. Distance is from the element reference plane.
GLSRCH	0 1	1	Flag to determine whether or not to account for the thickness of the element when calculating the normal distance for **CGLUE** connected surfaces. 1: Thickness will be accounted for. Distance is from the top or bottom surface of the element, depending on which outward normal is oriented appropriately. 0: Thickness is ignored. Distance is from the element reference plane.

6

Parameter for PLOAD4/5.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
PLOADM	0 1	1	Controls the way vector pressures are treated. A value of 1, the default, will use the full vector. A value of 0 will use the normal component only. This parameter affects PLOAD4 and PLOAD5.

Parameters for Shape Distortion Checking.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
SHAPECK	0 1 2 3 4 5	1	Controls the checking of the shape of regular 2D and 3D finite elements and of the DOMAIN elements. = 0 Skip all element shape checking. =1 Count shape distortion errors as non-fatal and suppress printing of warning-level problems. Error-level problems are printed as warnings. =2 Count shape distortion errors as non-fatal. Error-level problems are printed as warnings. =3 Suppress printing of warning-level problems. =4 Perform normal checking. Error-level problems are printed as errors and warning-level problems are printed as warnings. =5 Perform normal checking and print shape characteristics of every element.

Parameter for REDUCE Mode.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
SEMPC	Integer ≥ 0	0	If running in **REDUCE** mode and there is an eigenvector (**DISPLACEMENT** or **SVECTOR**) output request and SEMPC > 0, then **MPC** entries will be generated for the three translational displacements of each grid in the output set. These MPC entries can be used in a subsequent run of the residual model to recover degrees of freedom omitted by the superelement reduction. The MPC entries will be written to the DMIG post file. The entries will use the value of SEMPC as the MPC set id. If SEMPC is 0, then no MPC entries will be output.

Parameter for Eigenvalue/vector reuse.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
MODERST	NONE WRITE READ	NONE	If MODERST is WRITE, then eigenvalues and eigenvectors will be written into a special eigen database file after the eigensolve finishes. If MODERST is READ, then eigenvalues and eigenvectors will be read from a previously written eigen database file instead of performing eigenanalysis.

Parameter for Contact surface adjustment.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
BCADJUST	Real	0.0	If any **BCPAIR** entry specifies 'ADJUST' for GPAD (field 6), then the value of this parameter controls which potential contact points have their initial offsets adjusted. If the calculated surface distance at a contact point is less than BCADJUST, then the initial offset for that point is replaced with 0.0.

Parameters for Eqivalent Radiated Power

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
ERPC	Real	1.0	Speed of sound in the fluid. See **Equivalent Radiated Power** (Sec. 2.16.2).
ERPREFDB	Real > 0.0	1.0	Reference value for decibel calculation. See **Equivalent Radiated Power** (Sec. 2.16.2).
ERPRHO	Real	1.0	Density in the fluid. See **Equivalent Radiated Power** (Sec. 2.16.2).
ERPRLF	Real	1.0	Radiation loss factor. See **Equivalent Radiated Power** (Sec. 2.16.2).
RHOCP	Real > 0.0	1.0	Scale factor for decibel calculation. See **Equivalent Radiated Power** (Sec. 2.16.2).

Parameters for Postprocessing.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
IPRM16	0 1 2	0	Experimental parameter to control whether or not the model data is written to OUTPUT2 formatted post-processing files. = 0 The default, no model data will be written. = 1 The following tables will be included in OUTPUT2 files: Grid point definitions (GEOM1), Element definitions (GEOM2), Element properties (EPT) and Material properties (MPT). = 2 The following tables will be included in OUTPUT2 files: Coordinate system transformation (CSTM), Grid point list (GPL), Grid point definitions (BGPDT) and Element connections (ECT).
OP2ZIP	0 1 2 3 4 5 6	1	Compression control for OUTPUT2 postprocessing files. = 0 No file compression will be used. ≥ 1 OUTPUT2 files will be compressed in the gzip format, and the files will have the extension ".op2.gz". For OP2ZIP = 1, the compression is lossless. For values > 1, result data will be modified in manners intended to improve the compression ratio. Higher values will result in smaller file sizes, at the expense of a greater loss of information. The modifications are as follows: ≥ 2 Rotation components on displacement, velocity, acceleration and eigenvector results will be set to zero. ≥ 3 All element and grid result values will be reduced in precision by truncating the floating point representation, with higher values resulting in a greater loss of digits. At OP2ZIP = 4, there will be error in the fifth significant digit. At OP2ZIP = 6 there will be error in the third significant digit. It is not recommended to use values above 1 if the result data may be used for any purposes other than visualization.
OPPTH0	0 1	1	Postprocessing control for thickness 0: Print only current thickness 1: Print current thickenss and difference from original thickness
OPPTHK	0 1 2	0	Postprocessing control for thickness 0: Use standard element type indicators in the postprocessing file 1: PCOMP uses then PSHELL element type 2: PSHELL and PCOMP use the PSOLID element type
PCH2PST	YES NO	NO	PCH2PST=YES will cause *GENESIS* to use the "PST" extension instead of the "pch" extension for PUNCH format post-processing files.

6

TAPELBL	0 1	1	When TAPELBL=0, the NASTRAN label is not printed in the OUTPUT2 postprocessing file.
ZIPLVL	1-9	1	Controls the aggressiveness of the compressor engine when writing compressed OUTPUT2 files. Using a higher level will typically result in smaller file sizes at the expense of increased CPU time. This parameter is only effective when parameter OP2ZIP > 0.

Other Parameters.

PARAMETER	POSSIBLE VALUES	DEFAULT	DESCRIPTION
LOADCK	0 1	1	In static analysis, when LOADCK=0, *GENESIS* will allow a load set ID defined by a LOAD bulk data statement to also be used by FORCEi/MOMENTi/PLOADx entries in the bulk data.
SHELLCK	YES NO	YES	When SHELLCK is YES, the program checks if the bending stiffness from PSHELL data corresponds to a uniform section. For uniform sections, field 6 corresponds to either DF=1.0 or D = T**3/12. If the executive control command SOL COMPAT1is present in the input data then the 6th field represents DF otherwise the 6th field represents D. If SHELLCK is NO, no check is performed. Use this parameter carefully and only when nonuniform sections are used.
FINDEXCK	YES NO	YES	When FINDEXCK is YES, the program checks if the material properties needed for failure index calculations are available or not. If they are not available and FINDEX is requested then *GENESIS* will stop with an error message. If SHELLCK is NO, the program will not stop for the above reason but will turn off the FINDEX calculation request.
RANDOM	Integer	0	Set the seed for the pseudo-random number generator. Subspace iteration and certain design options use pseudo-random numbers to initalize certain arrays. Changing the seed will lead to a different initializing sequence and may alter the results.
SHL2SKN	Real ≥ 0.0	0.0	PSHELL properties with a thickness (field 4) less than or equal to this value will be automatically converted to PSKIN properties. This is intended to avoid numerical ill-conditioning that can result from such small thicknesses.
DUPTOL	0-5	0	Set the allowable tolerance for duplicate grid elimination. Two GRID entries with the same ID are only allowed if the X1,X2,X3 coordinates are within a close tolerance. If DUPTOL=0, there is no tolerance, that is, the coordinates must match exactly or a fatal error will occur. Higher values of DUPTOL allow for greater tolerances. At DUPTOL=5, coordinate differences up to about 0.1% are allowed.

6.7.105 PAXIS

Data Entry: **PAXIS** - Axisymmetric Element Property.

Description: Defines the material of **CTRIAX6** elements.

Format:

1	2	3	4	5	6	7	8	9	10
PAXIS	PID	MID	PMULT						

Example:

1	2	3	4	5	6	7	8	9	10
PAXIS	3	4	0.5						

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	MID	Material identification number of **MAT1**, **MAT3**, **MAT4** or **MAT5** data (Integer > 0).
4	PMULT	Multiplier on element stiffness, load, mass and damping properties (Real > 0.0 or Blank) (Default=1.0).

Remarks:

1. Property identification numbers must be unique with respect to all other property identification numbers.
2. For structural problems only, MAT1 or MAT3 data may be referenced. For heat transfer, MAT4 or MAT5 data may be referenced.
3. Material properties and stresses are defined in the r_m, z_m coordinate system.
4. When only structural loadcases are requested in the solution control data, heat transfer materials are optional. However, if heat transfer is requested, both structural and heat transfer materials must be supplied.

6.7.106 PBAR

Data Entry: **PBAR** - Beam Property.

Description: Defines the properties of a simple beam which is used to create bar elements via the **CBAR** data.

Format:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM		
+	C1	C2	D1	D2	E1	E2	F1	F2	
+	AS1 or K1	AS2 or K2	I12						

Example:

1	2	3	4	5	6	7	8	9	10
PBAR	1	2	1.5+4	2.81+7	1.25+7	2.03+6			
+	50.0	75.0	50.0	-75.0	-50.0	-75.0	-50.0	75.0	
+									

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	MID	Material identification number (Integer > 0).
4	A	Area of bar cross-section (Real > 0.0).
5,6	I1,I2	Area moments of inertia (Real) (I1 > 0.0, I2 > 0.0).
7	J	Torsional constant (Real $\geq$ 0 or blank. Default=0.0).
8	NSM	Nonstructural mass per unit length (Real $\geq$ 0 or blank. Default=0.0).
2-9	Ci, Di, Ei, Fi	Stress recovery locations (Real or blank. Default=0.0).
2,3	AS1,AS2 or K1,K2	Shear areas, AS1 and AS2, for shear stiffness calculations (Real $\geq$ 0.0 or blank. Default=0.0). or Shear area factors, K1=AS1/A and K2=AS2/A (Real $\geq$ 0.0 or blank. Default=0.0). See remark 4. Note a blank or a 0.0 value will cause the program to neglect shear deformation. See remark 5.
4	I12	Area moment of inertia (Real or blank; $I_1 I_2 > I_{12}^2$. Default=0.0).

6

Remarks:

1. Property identification numbers must be unique with respect to all property identification numbers.
2. PBAR data may only reference **MAT1** material data. For heat transfer analysis, only **MAT4** material data can be referenced.
3. The beam element geometry is shown below.
4. The meaning of the data in fields 2 and 3 of the second continuation line depends on the compatibility format mode of the input file, as specified by the executive control command **SOL**. If the mode is COMPAT0, then these fields are interpreted as shear areas AS1 and AS2. If the mode is COMPAT1, those fields are interpreted as shear area factors K1=AS1/A and K2=AS2/A.
5. The transverse shear area in planes 1 and 2 are AS1 and AS2, respectively. The default values for AS1 and AS2 produce no shear deformation; in other words, the transverse shear flexibilities are set equal to zero. AS1 and AS2 must be 0.0 (or blank) if $I12 \neq 0$.
6. In a heat transfer loadcase, only the area is used to calculate the conductive properties of the bar. All other properties are ignored. NSM is still used to compute the total mass of the system.
7. When only structural loadcases are specified in the solution control, only a structural material is necessary to be specified. However, when only heat transfer loadcases are specified, both structural and heat transfer materials must be specified.
8. If all values on the second continuation line are blank, then this line may be omitted. If all values on both the first and second continuation lines are blank, then both may be omitted.

The stress recovery locations C1 and C2, etc., are the y and z coordinates in the BAR element coordinate system of a point at which stresses are computed. Stresses are computed at both ends of the BAR. Tension stresses are positive

.

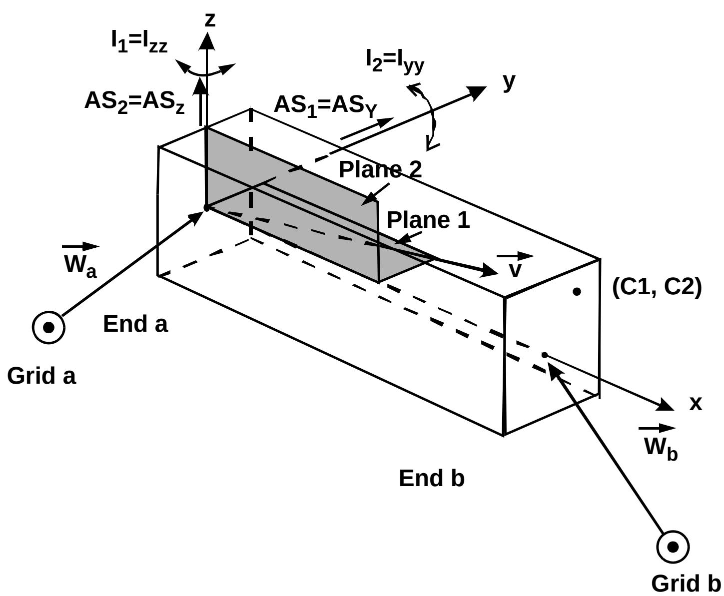


Figure 6-43

6.7.107 PBARL

Data Entry: **PBARL** - Bar Property.

Description: Defines the properties of a simple beam which is used to create bar elements via the **CBAR** data using libraries of available cross sections.

Format:

1	2	3	4	5	6	7	8	9	10
PBARL	PID	MID	LIBRARY	TYPE					
+	d1	d2	d3	d4	-etc.-	NSM			

Example:

1	2	3	4	5	6	7	8	9	10
PBARL	1	2	CSLIB1	BOX3					
+	20.0	2.0	34.0						

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	MID	Material identification number (Integer > 0).
4	LIBRARY	Cross-section Library (Character or blank. Default=CSLIB2) The following libraries are defined: CSLIB1 for cross sections from the GENESIS DVPROP3 Bar Library (see Remark 8). CSLIB2 for auxiliary cross sections (see Remark 9).
5	TYPE	Cross-section Type (Character). For LIBRARY="CSLIB1", one of: "SQUARE", "RECT", "CIRCLE", "TUBE", "SPAR", "BOX3", "BOX4", "IBEAM", "RAIL", "TEE", or "ANGLE". For LIBRARY="CSLIB2", one of: "I", "I1", "H", "Z", "T", "T1", "T2", "CROSS", "CHAN", "CHAN1", "CHAN2", "HAT", "BAR", "BOX", "BOX1", "HEXA", "ROD", or "TUBE".
2-	di	Cross section dimension (Real > 0.0) (See Remark 4).
	NSM	Nonstructural mass per unit length (Real $\geq$ 0 or blank. Default=0.0) (See Remark 4).

6

Remarks:

1. Property identification numbers must be unique with respect to <u>all</u> property identification numbers.
2. PBARL data may only reference **MAT1** material data. For heat transfer analysis, only **MAT4** material data can be referenced.
3. PBARL generates equivalent **PBAR** data. The sorted echo will show the data for the generated PBAR.
4. The number of cross section dimensions depends on the LIBRARY and TYPE. NSM is specified after the last cross-section dimension for the given TYPE.
5. The bar element geometry is shown below.
6. In a heat transfer loadcase, only the area is used to calculate the conductive properties of the bar. All other properties are ignored. NSM is still used to compute the total mass of the system.
7. When only structural loadcases are specified in the solution control, only a structural material is necessary to be specified. However, when only heat transfer loadcases are specified, both structural and heat transfer materials must be specified.
8. Cross sections for LIBRARY=”CSLIB1” are shown in **Figure 6-45**. These cross sections use the DVPROP3 dimensions to facilitate design. For TYPE=”SPAR”, d1 is an area; all other dimensions are lengths.
9. Cross sections for LIBRARY=”CSLIB2” are shown in **Figure 6-46**. Note that the y-z orientation is different from that used in ”CSLIB1”.

.

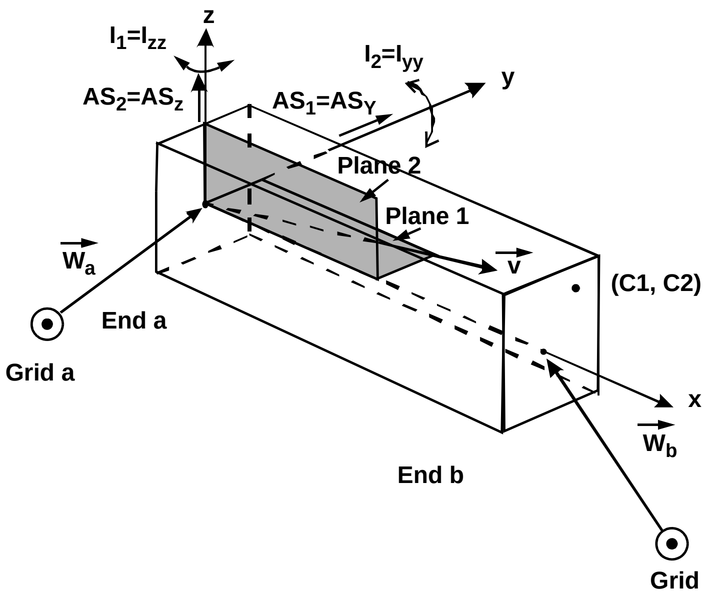


Figure 6-44

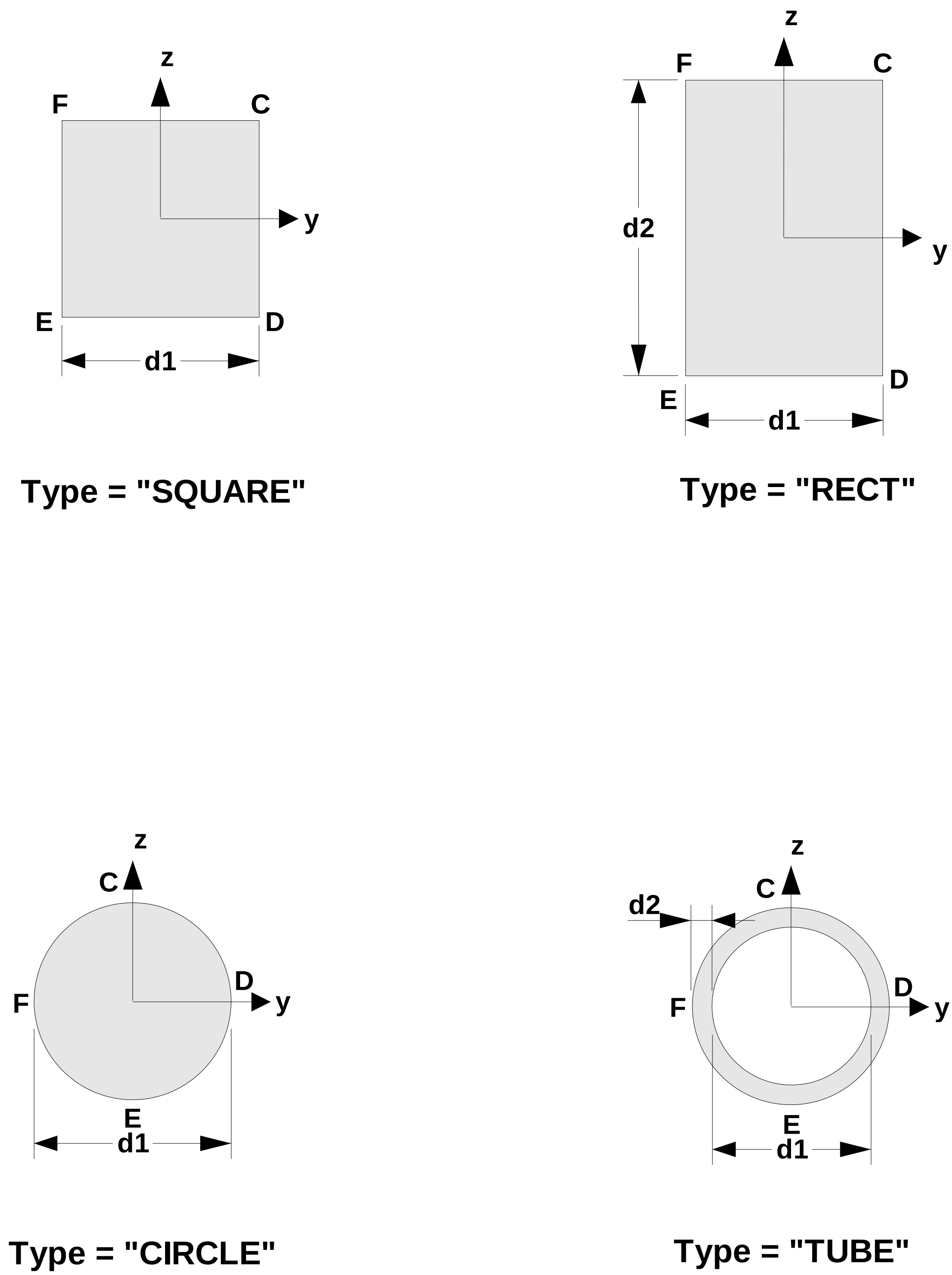
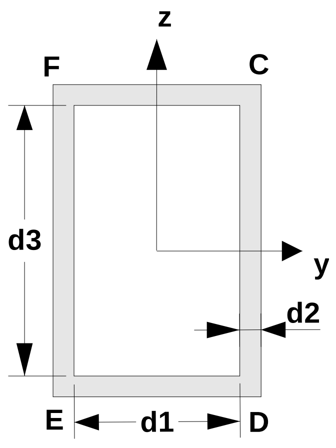
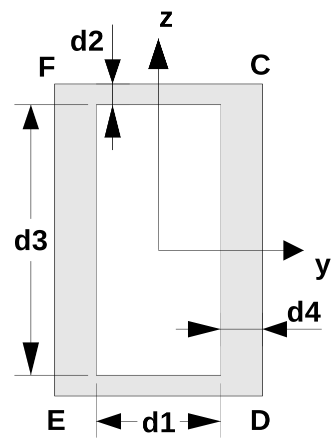


Figure 6-45

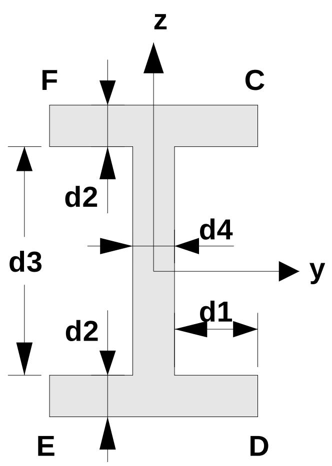
(CSLIB1)



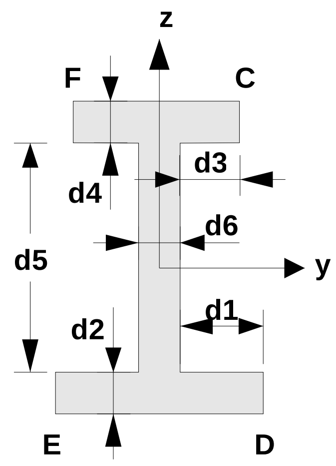
Type = "BOX3"



Type = "BOX4"



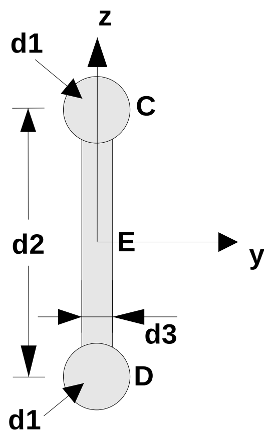
Type = "IBEAM"



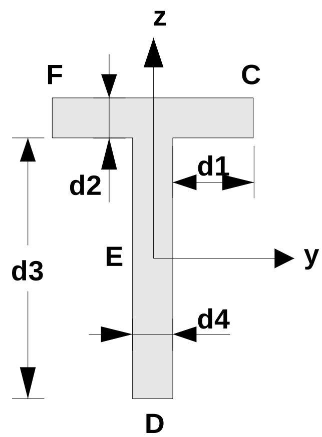
Type = "RAIL"

Figure 6-45 (cont.)

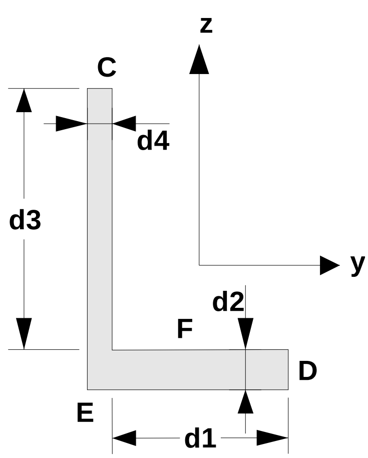
(CSLIB1)



Type = "SPAR"



Type = "TEE"



Type = "ANGLE"

Figure 6-45 (cont.)

(CSLIB1)

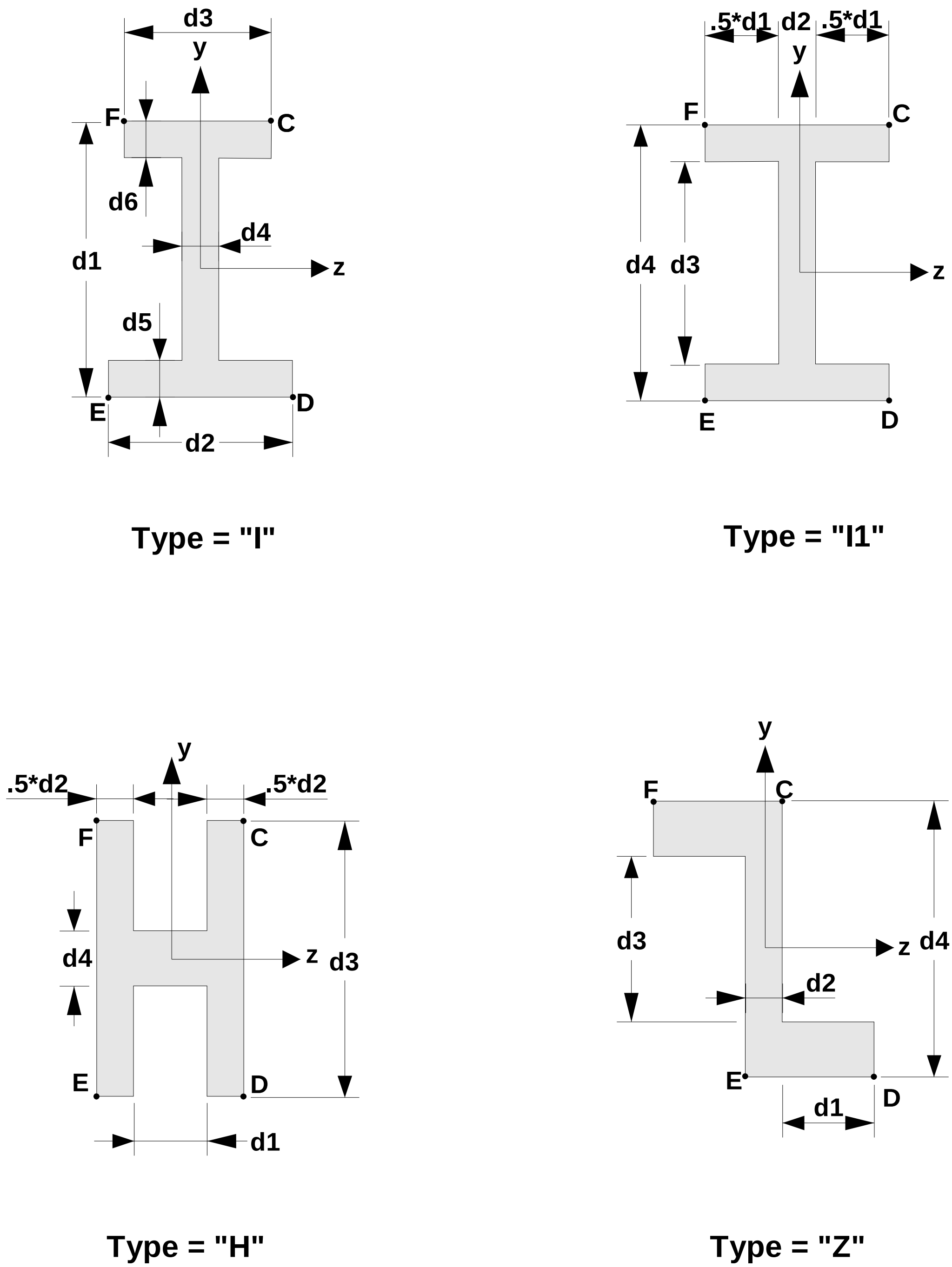
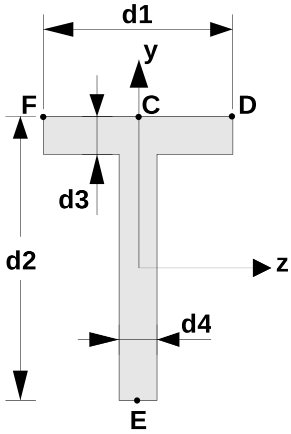
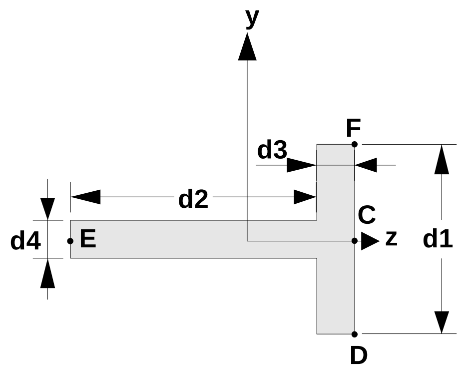


Figure 6-46

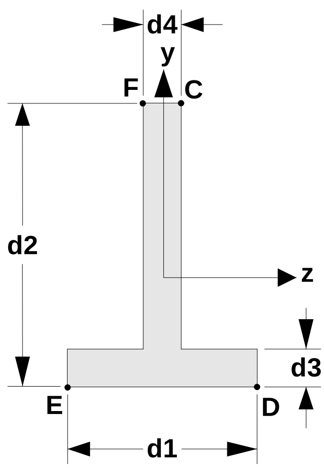
(CSLIB2)



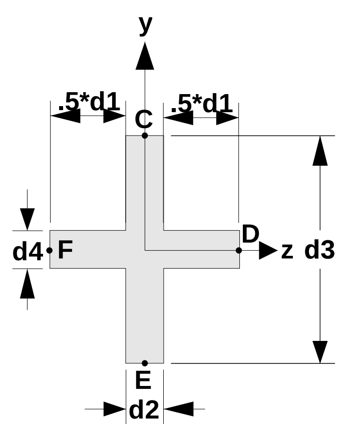
Type = "T"



Type = "T1"



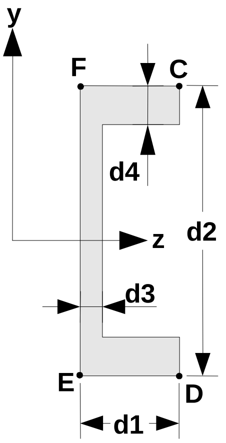
Type = "T2"



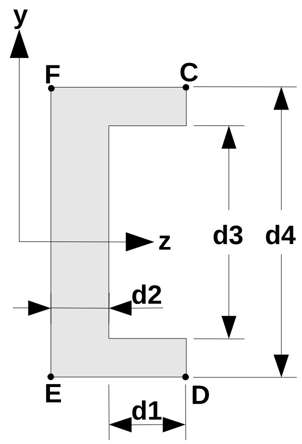
Type = "CROSS"

Figure 6-46 (cont.)

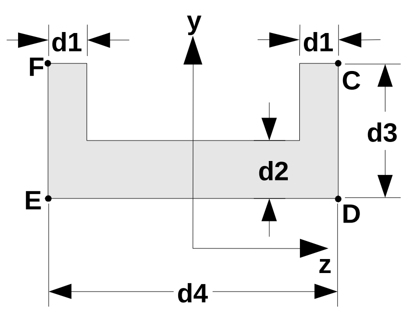
(CSLIB2)



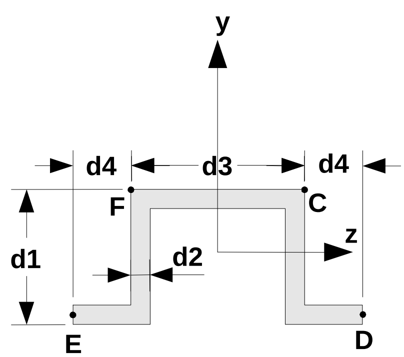
Type = "CHAN"



Type = "CHAN1"



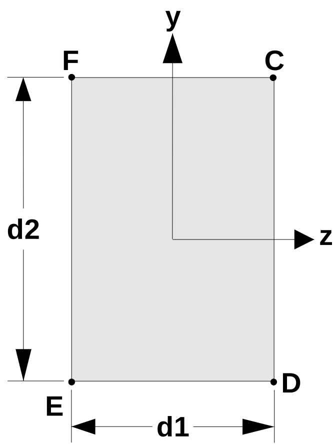
Type = "CHAN2"



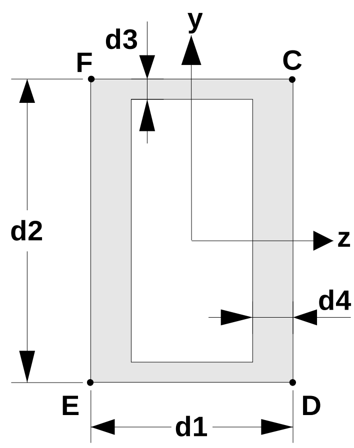
Type = "HAT"

Figure 6-46 (cont.)

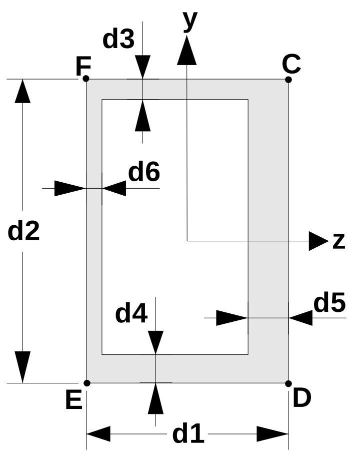
(CSLIB2)



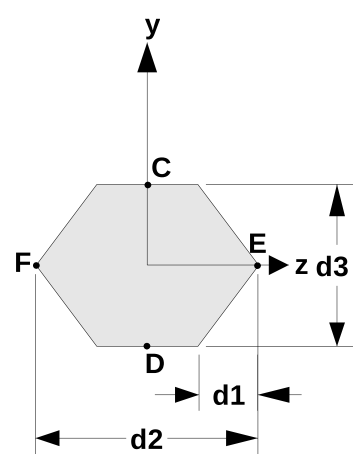
Type = "BAR"



Type = "BOX"



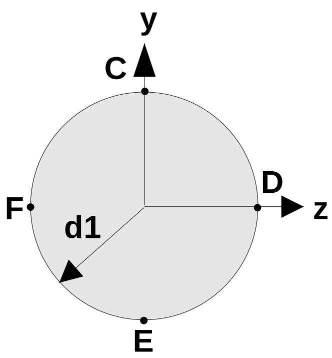
Type = "BOX1"



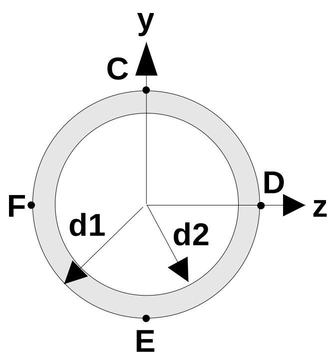
Type = "HEXA"

Figure 6-46 (cont.)

(CSLIB2)



Type = "ROD"



Type = "TUBE"

Figure 6-46 (cont.)

(CSLIB2)

6.7.108 PBEAM

Data Entry: **PBEAM** - Beam Property for CBEAM entry.

Description: Defines the properties of a simple or tapered beam which is used to create beam elements via the **CBEAM** data.

Format:

1	2	3	4	5	6	7	8	9	10
PBEAM	PID	MID	A(A)	I1(A)	I2(A)	I12(A)	J(A)	NSM(A)	
+	C1(A)	C2(A)	D1(A)	D2(A)	E1(A)	E2(A)	F1(A)	F2(A)	

The next two continuations can be repeated to describe up to nine intermediate sections and the end section, B, as described in Remark 3. SO and X/XB are required for each section.

1	2	3	4	5	6	7	8	9	10
+	SO	X/XB	A	I1	I2	I12	J	NSM	
+	C1	C2	D1	D2	E1	E2	F1	F2	

The last two continuations are:

1	2	3	4	5	6	7	8	9	10
+	K1	K2			NSI(A)	NSI(B)	CW(A)	CW(B)	
+	M1(A)	M2(A)	M1(B)	M2(B)	N1(A)	N2(A)	N1(B)	N2(B)	

Example: Uniform Beam without Shear Deformation

1	2	3	4	5	6	7	8	9	10
PBEAM	10	51	110.0	4504.0	49437.1				
+	15.0	26.0	0.0	26.0	0.0	0.0	0.0	-26.0	
+	0.0	0.0							

Example: Uniform Beam with Shear Deformation

1	2	3	4	5	6	7	8	9	10
PBEAM	12	51	110.0	4504.0	49437.1				
+	15.0	26.0	0.0	26.0	0.0	0.0	0.0	-26.0	
+	1.0	1.0							

Example: Tapered Beam with Two Sections

1	2	3	4	5	6	7	8	9	10
PBEAM	14	51	35.0	1119.09	1119.09	-668.829	11.6667	0.0	
+	13.1286	13.1286	22.8714	22.8714	13.1286	22.8714	22.8714	13.1286	
+	YES	1.0	21.0	242.702	242.702	-144.048	7.0	0.0	
+	7.88095	7.88095	14.119	14.119	7.88095	14.119	14.119	7.88095	
+	1.0	1.0							

Example: Uniform Beam with Mass Offset

1	2	3	4	5	6	7	8	9	10
PBEAM	18	51	35.0	1119.09	1119.09	-668.829	11.6667	0.0	
+	13.1286	13.1286	22.8714	22.8714	13.1286	22.8714	22.8714	13.1286	
+	1.0	1.0			1.0	1.0			
+	100.0		100.0						

Example: Uniform Beam with Warping. Neutral axis is offset from shear axis

1	2	3	4	5	6	7	8	9	10
PBEAM	18	51	35.0	1119.09	1119.09	-668.829	11.6667	0.0	
+	13.1286	13.1286	22.8714	22.8714	13.1286	22.8714	22.8714	13.1286	
+	1.0	1.0					1.0	1.0	
+					100.0		100.0		

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	MID	Material identification number (Integer > 0). See Remark 2.
4	A(A)	Area of beam cross-section at end A (Real > 0.0).
5	I1(A)	Area moments of inertia at end A for bending in plane 1 about the neutral axis. $I1 = I_{(ZZ)na}$ (Real $\geq$ 0)
6	I2(A)	Area moments of inertia at end A for bending in plane 2 about the neutral axis. $I2 = I_{(yy)na}$ (Real $\geq$ 0)
7	I12(A)	Area product of inertia at end A. $I12 = I_{(Zy)na}$ (Real $\geq$ 0.0 but $I1*I2 > I_{12}^2$ Default = 0.0)
8	J(A)	Torsional constant at end A. $J = I_{(xx)na}$ (Real $\geq$ 0 but > 0.0 if warping is present. Default = 0.0).
9	NSM(A)	Nonstructural mass per unit length (Real $\geq$ 0. Default = 0.0).

2-9	Ci(A), Di(A), Ei(A), Fi(A)	Stress recovery locations at end A (Real. Default = 0.0).
2	SO	Stress/force output request option (Character). "YES" means stresses at points C_i, D_i, E_i, F_i on the next continuation and forces are recovered. "YESA" means stresses at the same points as end A and forces are recovered. "NO" means no stresses or forces are recovered.
3	X/XB	Distance from end A in the element coordinate system divided by the length of the element. (Real $>$ 0).
4-9	A, I1, I2, I12, J, NSM	Area, moment of inertia, torsional stiffness parameter and non-structural mass for the cross section located at X/XB (All real, J > 0.0 if warping is present). See Remark 5.
2-9	Ci, Di, Ei, Fi	Stress recovery location for the cross section located at X/XB. (Real. Default = 0.0).
2, 3	K1, K2	Shear stiffness factor K in K*A*G for plane1 and plane 2. (Real $\geq$ 0.0. Defaults = 1.0, 1.0).
6, 7	NSI(A), NSI(B)	Non structural mass moment of inertia per unit length about the non structural mass center of gravity at ends A and B. (Real. Default for NSI(A) is 0.0. Default for NSI(B) is same as NSI(A)).
8, 9	CW(A), CW(B)	Warping coefficients for ends A and B. (Default = 0.0). Default of CW(B) is CW(A).
2-5	M1(A), M2(A), M1(B), M2(B)	Locations (y, z coordinates) of the center of gravity of nonstructural mass for ends A and B. (Real). (Default = 0.0). M1(A) = y_{ma}, M2(A) = z_{ma}, M1(B) = y_{mb}, M2(B) = z_{mb}. See figure below. Default = 0.0 for M1(A) and M2(A). Default of M1(B) and M2(B) is the same as end A.
6-9	N1(A), N2(A), N1(B), N2(B)	Locations (y, z coordinates) of the neutral axis for ends A and B. (Real). (Default = 0.0). Default = 0.0 for N1(A) and N2(A). Default of N1(B) and N2(B) is the same as end A.

Remarks:

1. Property identification numbers must be unique with respect to all property identification numbers.
2. PBEAM data may only reference **MAT1** material data for structural analysis. For heat transfer analysis, only **MAT4** material data can be referenced.
3. Continuation lines requirement:
    a. The first continuation entry, which contains the fields C1(A) through F2(A) can be omitted only if no stress data at end A is to be recovered and a continuation with the SO field is specified.
    b. If SO is "YESA" or "NO", the continuation for Ci, Di, Ei and Fi must be omitted. If SO is "YES", Ci, Di, Ei and Fi are required on the next continuation line.

c. The second and third continuation entries can be repeated up to ten more times for unique intermediate X/XB values. The order of these continuation pairs is independent of the X/XB value, but one of them must be X/XB = 1.0, corresponding to end B.

d. The last two continuation entries, which contain fields K1 through N2(B), are optional and may be omitted if the default values are appropriate.

5. If any of fields 4 through 9 are left blank on the continuation with the value of X/XB = 1.0, then the values for A, I1, I2, I12, J and/or NSM are set to the values for end A (Default). For the continuations that have intermediate values (between 0.0 and 1.0) of X/XB, if any of the fields 4 through 9 are blank (Default), a linear interpolation between the values at ends A and B is constructed to fill the blank section properties.
6. If requested in the solution control section, stresses and forces are printed at X/XB = 0.0 and other sections with SO = “YES” or “YESA”.
7. According to Timoshenko beam theory, the shear stiffness factors, K1 and K2 adjust the effective transverse shear cross-section area. Zero values of K1 and K2 results in the Bernoulli-Euler beam theory, which neglects shear deformation.
8. The warping coefficients, CWA and CWB, are averaged to create a unique warping coefficient, CW. The warping contributions to the stiffness matrix are calculated by solving the following differential equation:

$$\frac{d^4\theta}{dx^4} - \frac{k^2}{l^2}\frac{d^2\theta}{dx^2} = \frac{m}{EC_W}$$

where

θ = rotation of the beam

k = nondimensional parameter given by $k = l\sqrt{\frac{GJ}{EC_W}}$

l = length of beam
E = Modulus of elasticity
G = Shear modulus

C_W = Warping coefficient (units are length^6)
J = torsional moment of inertia
m = torsional moment per unit length

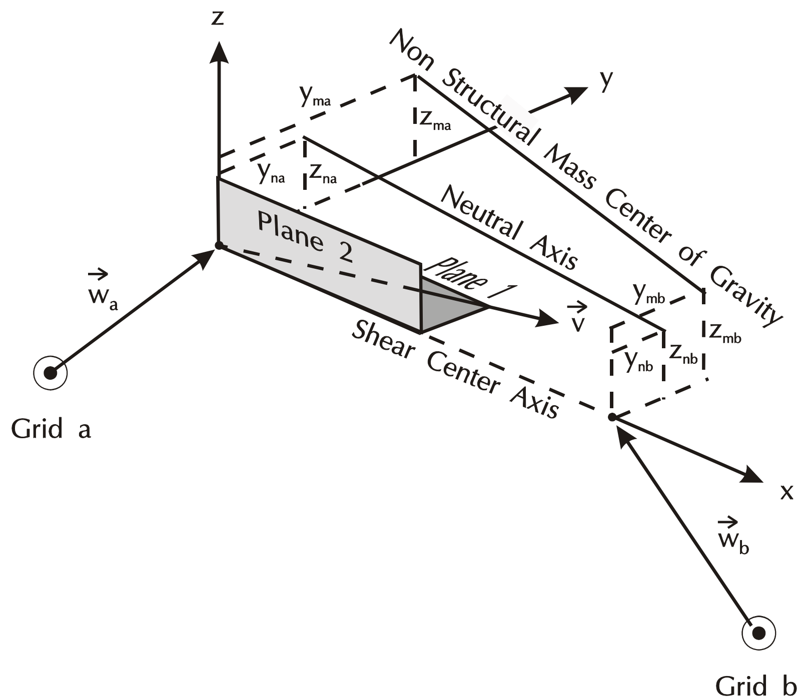

z
y
x
Non Structural Mass Center of Gravity
Neutral Axis
Shear Center Axis
Plane 2
Plane 1
y_{ma}
z_{ma}
y_{na}
z_{na}
y_{mb}
z_{mb}
y_{nb}
z_{nb}
$\vec{w}_a$
$\vec{v}$
$\vec{w}_b$
Grid a
Grid b

Figure 6-47

6.7.109 PBEAML

Data Entry: **PBEAML** - Property for CBEAM entry.

Description: Defines the properties of a simple beam which is used to create beam elements via the **CBEAM** data using libraries of available cross sections.

Format:

1	2	3	4	5	6	7	8	9	10
PBEAML	PID	MID	LIBRARY	TYPE					
+	d1(A)	d2(A)	...	dn(A)	NSM(A)	SO(1)	X(1)/XB	d1(1)	
+	d2(1)	etc.	dn(1)	NSM(1)	SO(2)	X(2)/XB	d1(2)	d2(2)	
+	etc.	dn(2)	etc	SO(m)	X(m)/XB	d1(m)	d2(m)	etc.	
+	dn(m).	NSM(m)	SO(B)	1.0	d1(B)	d2(B)	etc.	d2(m)	

Example: Constants section I beam

1	2	3	4	5	6	7	8	9	10
PBEAML	1	2	CSLIB2	I					
+	20.0	5.0	4.0	1.0	1.0	1.0			

Example: Taper section I beam, using 5 cross sections (3 internal stations)

1	2	3	4	5	6	7	8	9	10
PBEAML	1	2	CSLIB2	I					
+	20.0	5.0	4.0	1.0	1.0	1.0		yes	
+	0.25	18.0	5.0	4.0	1.0	1.0	1.0		
+	yes	0.50	16.0	5.0	4.0	1.0	1.0	1.0	
+		yes	0.75	14.0	5.0	4.0	1.0	1.0	
+	1.0		yes	1.0	12.0	5.0	4.0	1.0	
+	1.0								

In the above format n correspondn to the number of cross-section dimensions and m corresponds to the number of intermediate stations

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	MID	Material identification number (Integer > 0).
4	LIBRARY	Cross-section Library (Character or blank. Default=CSLIB2) The following libraries are defined: CSLIB1 GENESIS Library (see Remark 8). CSLIB2 for auxiliary cross sections (see Remark 9).

5	TYPE	Cross-section Type (Character). For LIBRARY="CSLIB1", one of: "SQUARE", "RECT", "CIRCLE", "TUBE", "SPAR", "BOX3", "BOX4", "IBEAM", "RAIL", "TEE", or "ANGLE". For LIBRARY="CSLIB2", one of: "I", "I1", "H", "Z", "T", "T1", "T2", "CROSS", "CHAN", "CHAN1", "CHAN2", "HAT", "BAR", "BOX", "BOX1", "HEXA", "ROD", "TUBE" or "L".
	di(A), di(B), di(j)	Cross section dimensions at ends A and B and itermediate station j (Real > 0.0) (See Remark 4).
	NSM(A), NSM(B), NSM(j)	Nonstructural mass per unit length at ends A and B and intermediate station j(Real $\geq$ 0 or blank. Default=0.0) (See Remark 4).
2	SO(j),SO(B)	Stress/force output request option (Character). "YES" means stresses at points C_i, D_i, E_i, F_i on the next continuation and forces are recovered. " "NO" means no stresses or forces are recovered.
3	Xj/XB	Distance from end A to intermediate station j in the element coordinate system divided by the length of the element. (Real $>$ 0).Default 1.0.

6

Remarks:

1. Property identification numbers must be unique with respect to <u>all</u> property identification numbers.
2. PBEAML data may only reference **MAT1** material data. For heat transfer analysis, only **MAT4** material data can be referenced.
3. PBEAML generates equivalent **PBEAM** data. The sorted echo will show the data for the generated PBEAM.
4. The number of cross section dimensions depends on the LIBRARY and TYPE. .
5. The beam element geometry is shown below.
6. In a heat transfer loadcase, only the area is used to calculate the conductive properties of the bar. All other properties are ignored. NSM is still used to compute the total mass of the system.
7. When only structural loadcases are specified in the solution control, only a structural material is necessary to be specified. However, when only heat transfer loadcases are specified, both structural and heat transfer materials must be specified.
8. Cross sections for LIBRARY=”CSLIB1” are shown in **Figure 6-49**. Although these sections match the cross sections used by DVPROP3, none of the PBEAML dimensions can be designed. For TYPE=”SPAR”, d1 is an area; all other dimensions are lengths.
9. Cross sections for LIBRARY=”CSLIB2” are shown in **Figure 6-50**. Note that the y-z orientation is different from that used in ”CSLIB1”.
10. The element coordinate system is located in the shear center.

.

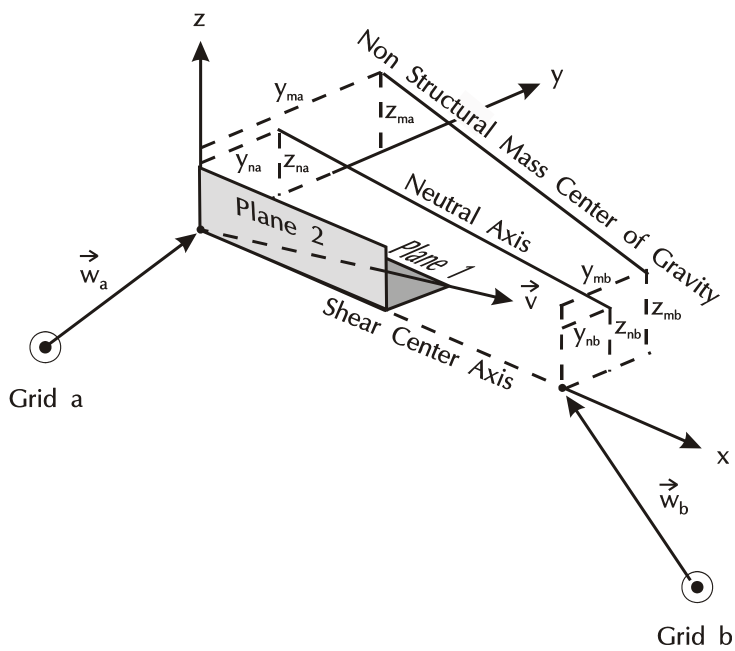
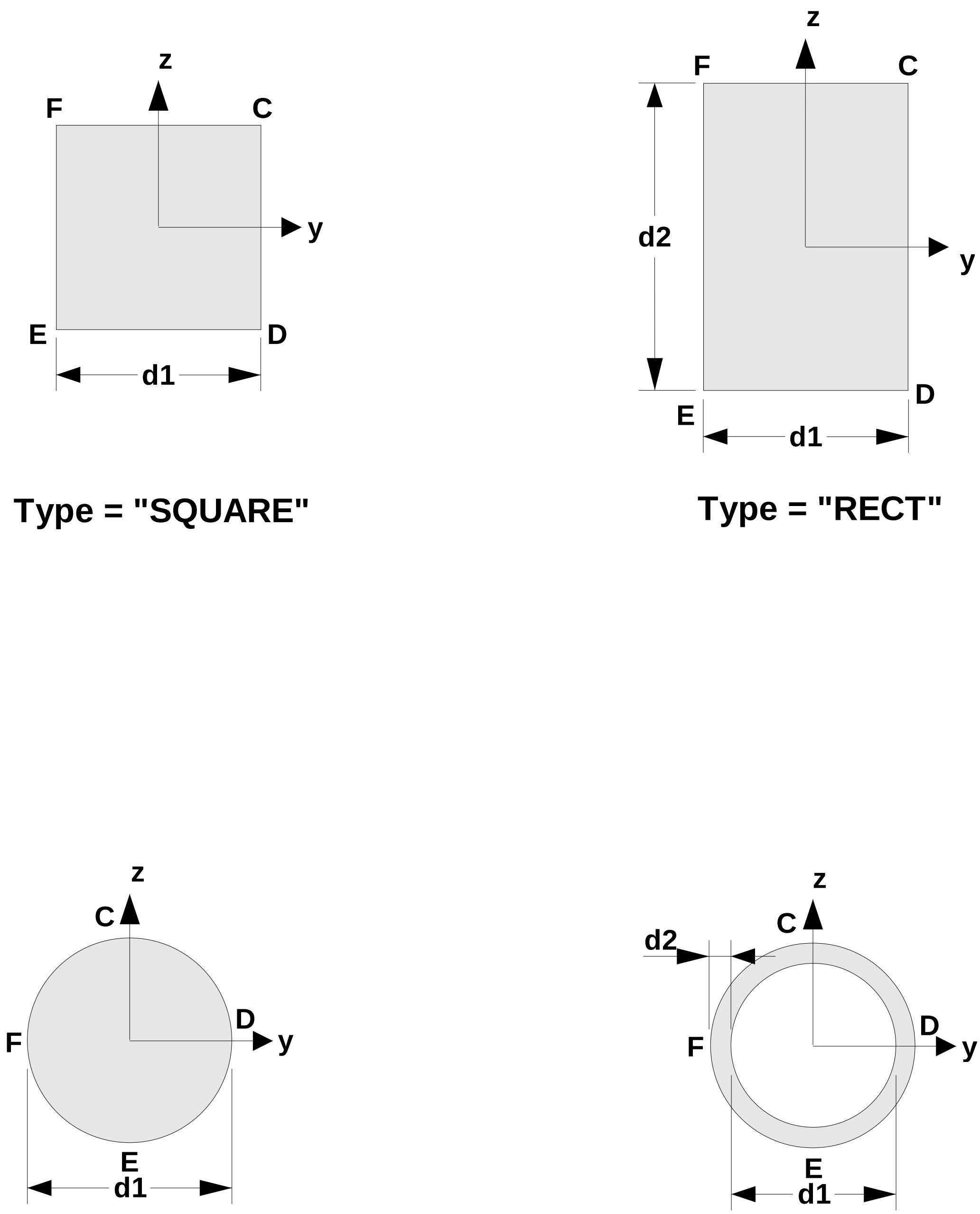


Figure 6-48



Type = "SQUARE"

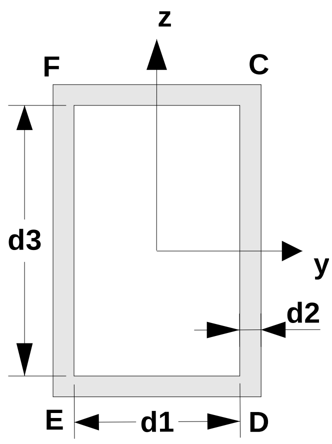
Type = "RECT"

Type = "CIRCLE"

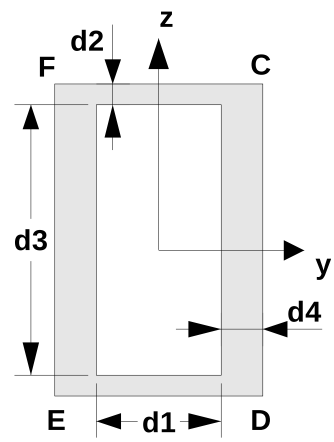
Type = "TUBE"

Figure 6-49

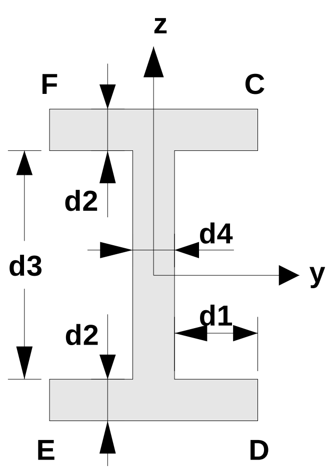
(CSLIB1)



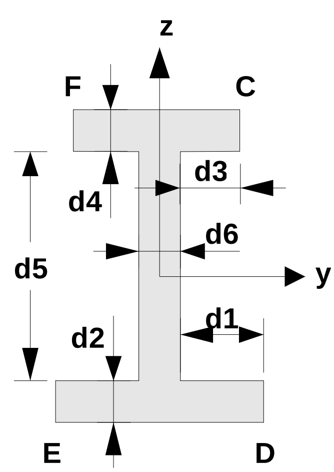
Type = "BOX3"



Type = "BOX4"



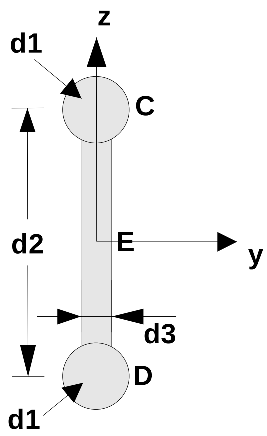
Type = "IBEAM"



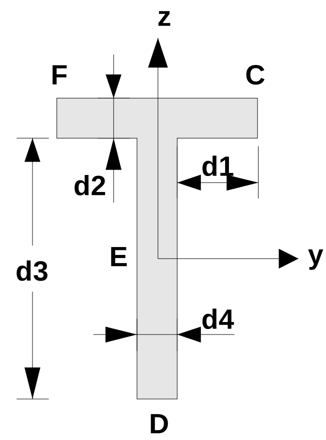
Type = "RAIL"

Figure 6-49 (cont.)

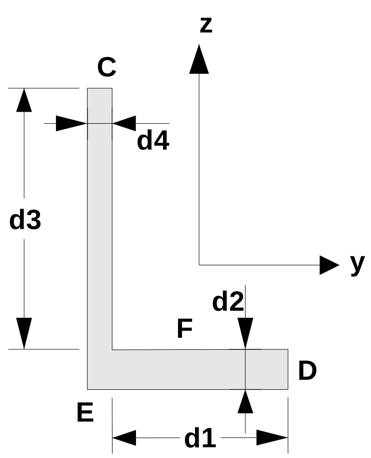
(CSLIB1)



Type = "SPAR"



Type = "TEE"



Type = "ANGLE"

Figure 6-49 (cont.)

(CSLIB1)

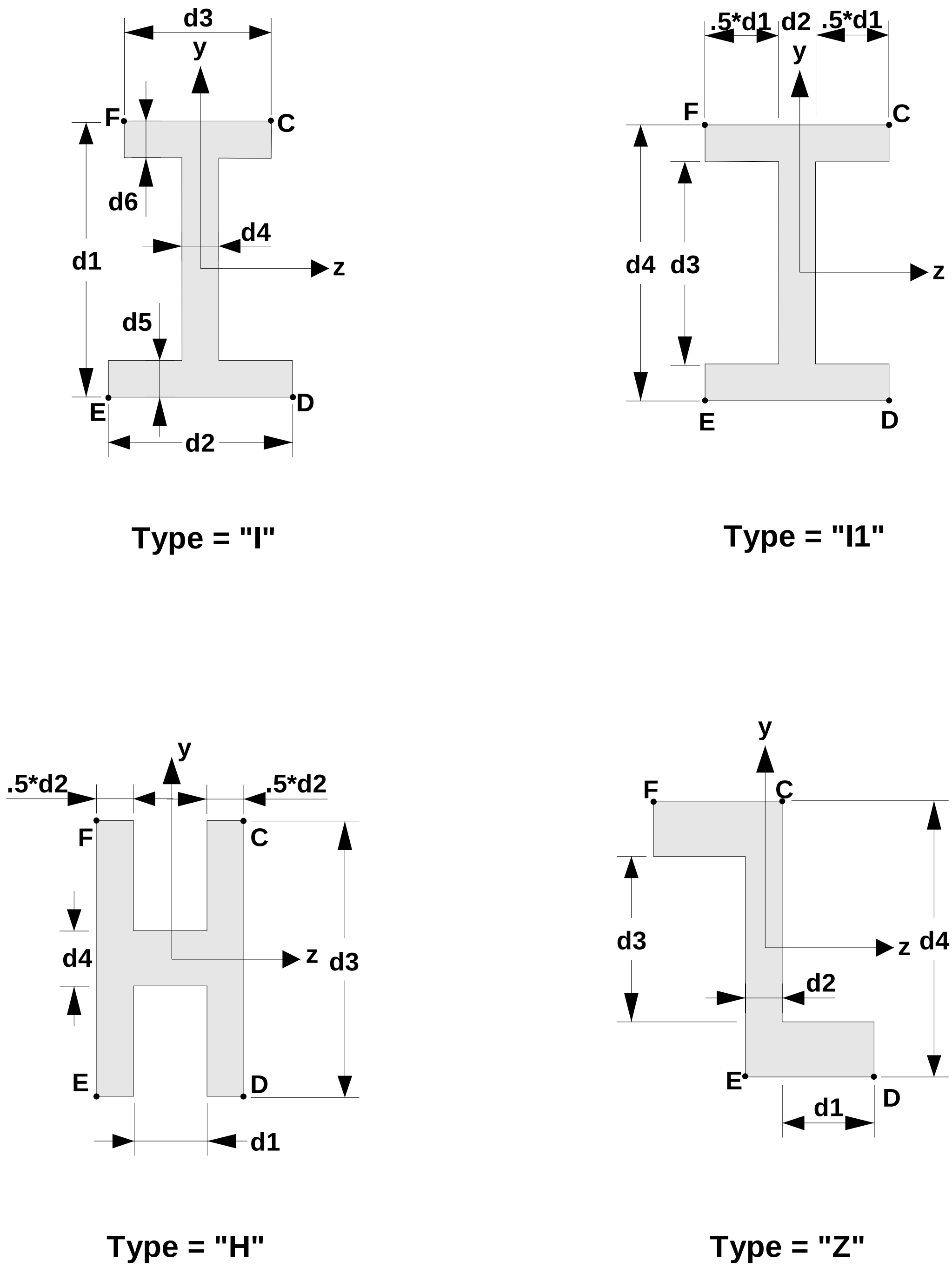
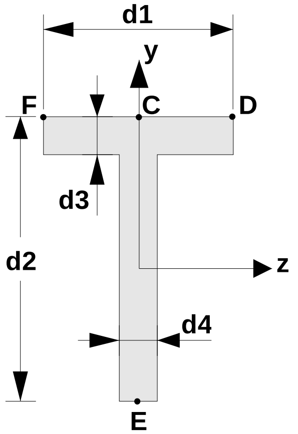
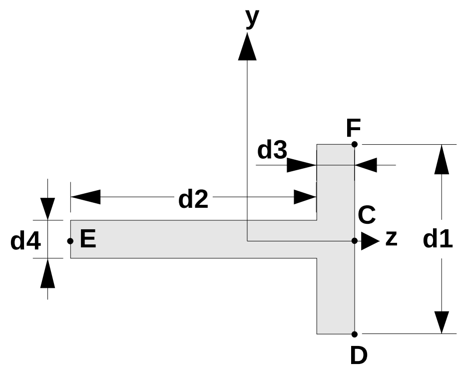


Figure 6-50

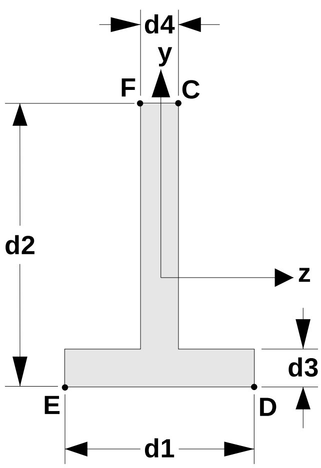
(CSLIB2)



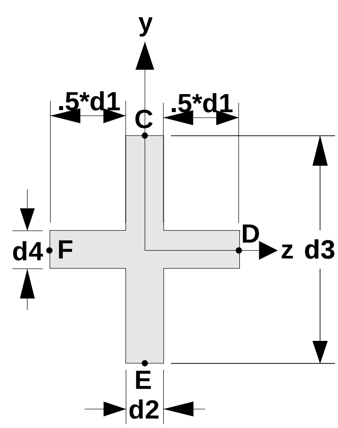
Type = "T"



Type = "T1"



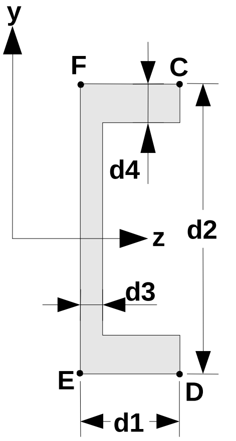
Type = "T2"



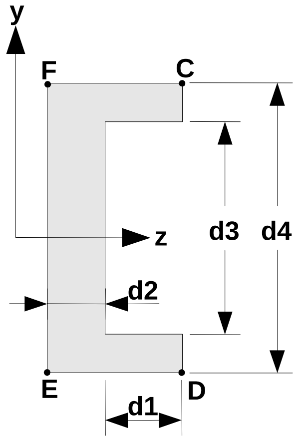
Type = "CROSS"

Figure 6-50 (cont.)

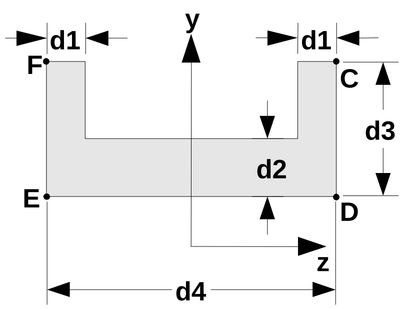
(CSLIB2)



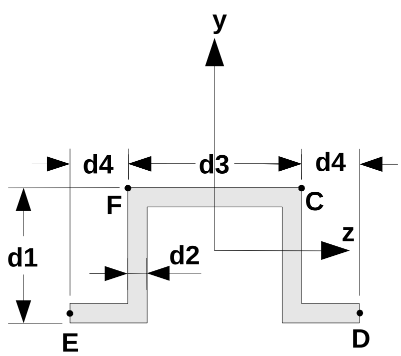
Type = "CHAN"



Type = "CHAN1"



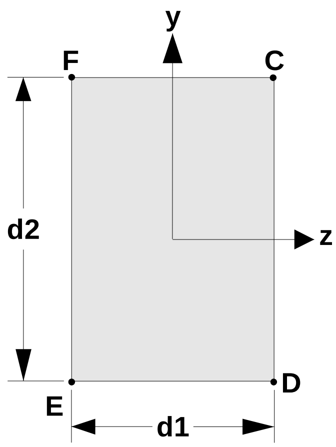
Type = "CHAN2"



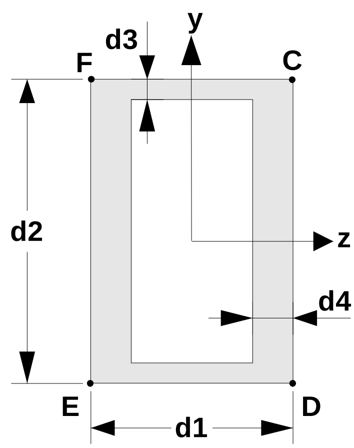
Type = "HAT"

Figure 6-50 (cont.)

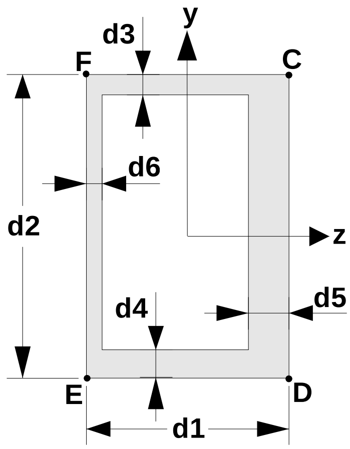
(CSLIB2)



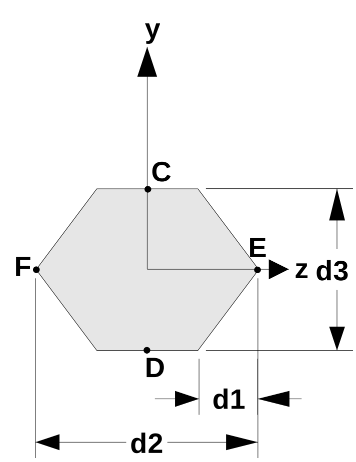
Type = "BAR"



Type = "BOX"



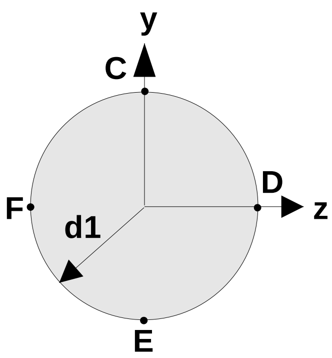
Type = "BOX1"



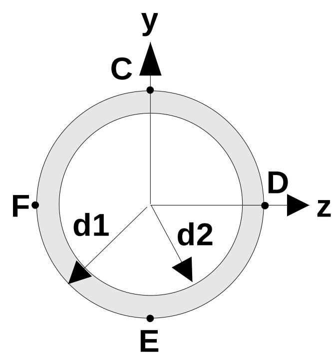
Type = "HEXA"

Figure 6-50 (cont.)

(CSLIB2)



Type = "ROD"



Type = "TUBE"

Figure 6-50 (cont.)

(CSLIB2)

6.7.110 PBUSH

Data Entry: **PBUSH** - Generalized Elastic Element Property.

Description: Used to define properties of a generalized spring-damper.

Format:

1	2	3	4	5	6	7	8	9	10
PBUSH	PID	“K”	K1	K2	K3	K4	K5	K6	
+		“B”	B1	B2	B3	B4	B5	B6	
+		“GE”	GE						
+		“RCV”	ST	SR	ET	ER			

Example:

1	2	3	4	5	6	7	8	9	10
PBUSH	1001	K	2.3E6	2.6E6	1.4E6	4.3E9	5.7E9	2.7E9	
+		B	1.3E3	1.3E3	1.3E3	4.2E4	4.2E4	4.2E4	
+		GE	0.02						
+		RCV	0.1	4.2	1.0	1.0			

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	“K”	Label indicating the line contains stiffness values.
4-6	K1, K2, K3	Translational stiffness values in the element coordinate system. (Real. Default = 0.0)
7-9	K4, K5, K6	Rotational stiffness values in the element coordinate system. (Real. Default = 0.0)
3	“B”	Label indicating the line contains viscous damping values.
4-6	B1, B2, B3	Translational viscous damping values in the element coordinate system. (Real. Default = 0.0)
7-9	B4, B5, B6	Rotational viscous damping values in the element coordinate system. (Real. Default = 0.0)
3	“GE”	Label indicating the line contains a structural damping coefficient.
4	GE	Structural damping coefficient. (Real. Default = 0.0)
3	“RCV”	Label indicating the line contains stress and strain recovery coefficients.
4	ST	Stress recovery coefficient for translational components. (Real. Default = 1.0)

5	SR	Stress recovery coefficient for rotational components. (Real. Default = 1.0)
6	ET	Strain recovery coefficient for translational components. (Real. Default = 1.0)
7	ER	Strain recovery coefficient for rotational components. (Real. Default = 1.0)

Remarks:

1. The element stresses are calculated by multiplying the stress coefficients by the recovered forces.
2. The element strains are calculated by multiplying the strain coefficients by the recovered displacements.
3. The order of "K", "B", "GE" and "RCV" labeled lines are not important.
4. The continuation lines may be omitted.
5. To avoid numerical errors caused by ill-conditioning, the stiffness Ki values are limited by the value of **PARAM BUSHSTIF**.

6.7.111 PBUSHT

Data Entry: **PBUSHT** - Frequency Dependent Generalized Elastic Element Property.

Description: Used to define frequency dependent properties of a generalized spring-damper.

Format:

1	2	3	4	5	6	7	8	9	10
PBUSHT	PID	“K”	TKID1	TKID2	TKID3	TKID4	TKID5	TKID6	
+		“B”	TBID1	TBID2	TBID3	TBID4	TBID5	TBID6	
+		“GE”	TGEID1	TGEID2	TGEID3	TGEID4	TGEID5	TGEID6	

Example:

1	2	3	4	5	6	7	8	9	10
PBUSHT	111	K	10	10	10	20	20	20	
+		B	30						
+		GE	40						

Field	Information	Description
2	PID	Property identification number that matches the identification number on a **PBUSH** entry (Integer > 0).
3	“K”	Label indicating the line contains stiffness frequency table identification numbers.
4-9	TKIDi	Identification number of a **TABLED1**, **TABLED2**, **TABLED3** or **TABLED4** entry that defines the stiffness vs. frequency relationships in directions 1 though 6. (Integer > 0 or blank)
3	“B”	Label indicating the line contains viscous damping frequency table identification numbers.
4-9	TBIDi	Identification number of a TABLEDi entry that defines the force per unit velocity vs. frequency relationships in directions 1 though 6. (Integer > 0 or blank).
3	“GE”	Label indicating the line contains structural damping frequency table identification numbers
4-9	TGEIDi	Identification number of a TABLEDi entry that defines the structural damping vs. frequency relationships in directions 1 though 6. (Integer > 0 or blank)

6

Remarks:

1. The “K”, “B”, and “GE” fields are associated with the same entries on the PBUSH entry which provide the nominal values. If the table ID for any field is blank, the nominal value for that property will be used for all loading frequencies.
2. The nominal values are used for all analysis types except frequency response analyses. For modal frequency response the system modes are computed using the nominal Ki values. The frequency dependent values are used at each loading frequency.
3. The order of "K", "B" and "GE" labeled lines are not important.
4. The continuation lines may be omitted.

6.7.112 PCOMP

Data Entry: **PCOMP** - Layered Composite Element Property.

Description: Used to define the properties of an n-ply composite material laminate.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z_0	NSM	SB	F.T.	TREF	GE	LAM	MEM
+	MID1	T1	θ_1	SOUT1	MID2	T2	θ_2	SOUT2	
+	MID3	T3	θ_3	SOUT3	-etc.-				

Example:

1	2	3	4	5	6	7	8	9	10
PCOMP	311	-0.15	6.85		H0FF				
+	86	0.056	0.0	YES			45.0		
			-45.0				90.0		

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	Z_0	Offset from the reference plane to the bottom surface (Real or Blank. Blank if LAM=SYM), (Default = -1/2 the thickness of the element)
4	NSM	Non-structural mass per unit area (Real > 0.0 or Blank).
5	SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress) (Real > 0.0 or Blank).
6	F.T.	Failure theory to be used to test whether the element fails or not (Character, Integer > 0 or blank). If character, the following theories are allowed: HILL for the Hill theory HOFF for the Hoffman theory TSAI for the Tsai-Wu theory STRN for the maximum strain theory If Integer, user supplied failure index equation identification number specified in **FINDEX** or **FINDEXN**.
7	TREF	Reference temperature (Real or Blank).
8	GE	Damping coefficient (Real or Blank).
9	LAM	Symmetric lamination option (Character or Blank). If blank, all plies must be entered. If set to SYM or SYM1, describe only plies on one side of element centerline. If set to SYM, the Z_0 is always 1/2 the total thickness of the element.

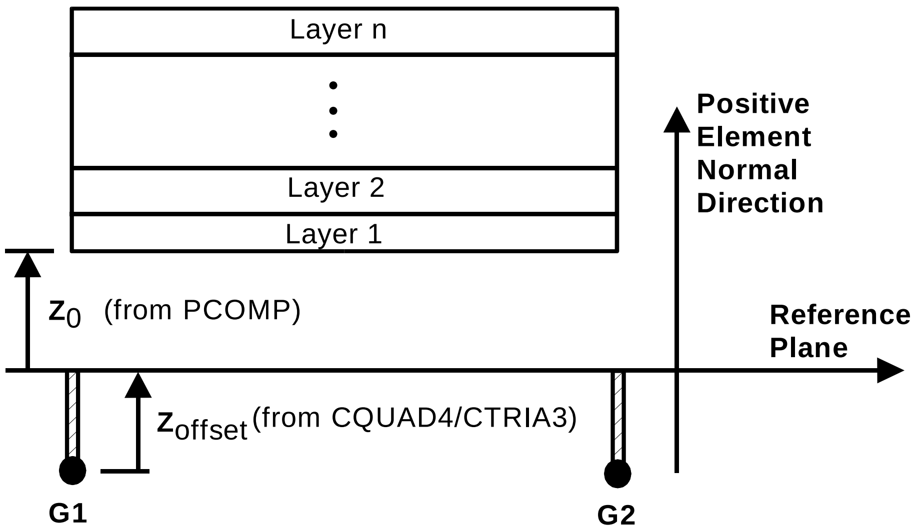
10	MEM	Membrane properties only. (Character or Blank). If set to MEM, then only the membrane stiffness will be used. If this field is blank, then the bending, membrane and bending-membrane coupling stiffness will be used. See Remark 7.
2,6	MIDi	Material ID for the various plies. MID1 > 0, MID2, MID3, ... MIDn > 0 or Blank. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDs must refer to **MAT1**, **MAT2** or **MAT8** data statements. (MID1: Integer > 0. All others, Integer > 0 or Blank). See Remark 1 for defaults.
3,7	Ti	Thickness of the various plies (Real: T1 > 0.0, T2, T3, ... > 0.0 or Blank). See Remark 1 for default.
4,8	θ_i	Orientation angle of the longitudinal direction of each ply with the material axis of the element. If the material angle on the element connection statement is 0.0, the material axis and side 1-2 of the element coincide. The plies are to be numbered serially, starting with 1 at the bottom layer (the bottom layer is defined as the surface with the largest -Z value in the element coordinate system). (Real, Default = 0.0).
5,9	SOUTi	Stress or strain output required (YES) or not (NO) for the various plies or for the maximum ply only (IFMAX). (Character). (Default = NO). See Remark 8.

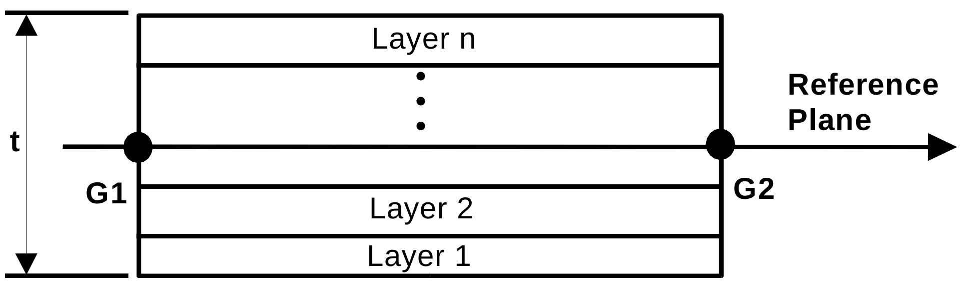
Remarks:

1. The default under MID2, MID3, ... is the last defined MAT statement. In the example shown, MID1 through MID4 are equal to 86. This also applies to the Ti values.
2. At lease one of the four values (MIDi, Ti, θ_i and SOUTi) must be present for a ply to exist. The minimum number of plies is one.
3. TREF given on the PCOMP statement will be used for all plies of the element. It will override values supplied on material data statements for individual plies.
4. GE given on the PCOMP data statement will be used for the element. It will override values supplied on material data statements for individual plies.
5. If transverse shear calculations are required, the G1,Z and G2,Z must be supplied in all MAT8 data referenced by the PCOMP data. If G1,Z and G2,Z in the MAT8 data are left blank, then there is no transverse shear flexibility.
6. PCOMP property identification numbers must be unique with respect to **PCOMPG**, **PSHELL** and **PSKIN** property identification numbers. It is recommended that property identification numbers be unique with respect to all other property identification numbers.
7. If MEM is entered in field 10, then the element stiffness matrix is geometrically condensed to remove the transverse displacement and the two bending rotational degrees of freedom. The drilling rotation is condensed out using static condensation. This method produces a stiffness matrix with two degrees of freedom per node in the element coordinate system. In general, in the global coordinate system, these will be three degrees of freedom per node. PARAM AUTOSPC should be set to YES to eliminate the transverse displacement.

8. When SOUTi = YES:
For the HILL, HOFF, TSAI and **FINDEX** equation options, the failure index is printed in the output file with the composite stresses (STRESS = ALL or any appropriate options). For the STRN or **FINDEXN** equation options, the failure index is printed in the output file with the composite strains (STRAIN = ALL or any appropriate options).
9. See **Composite Elements (CQUAD4, CTRIA3, CQUAD8 and CTRIA6 referencing PCOMP/PCOMPG data)** (Sec. 2.4.6) for a description of the failure theories.
10. A positive value for Z_0 puts the bottom surface of the elements above the reference plane. A negative value for Z_0 puts the bottom surface of the elements below the reference plane. See the figure below:
11. Interlaminar shear stresses and strains are calculated only when G1,Z and G2,Z are nonzero in all MAT8 data referenced by the PCOMP data. Additionally, when SB is nonzero, a bonding material failure index is calculated as $FB = \frac{max(|\sigma_{1z}|, |\sigma_{2z}|)}{SB}$.

6

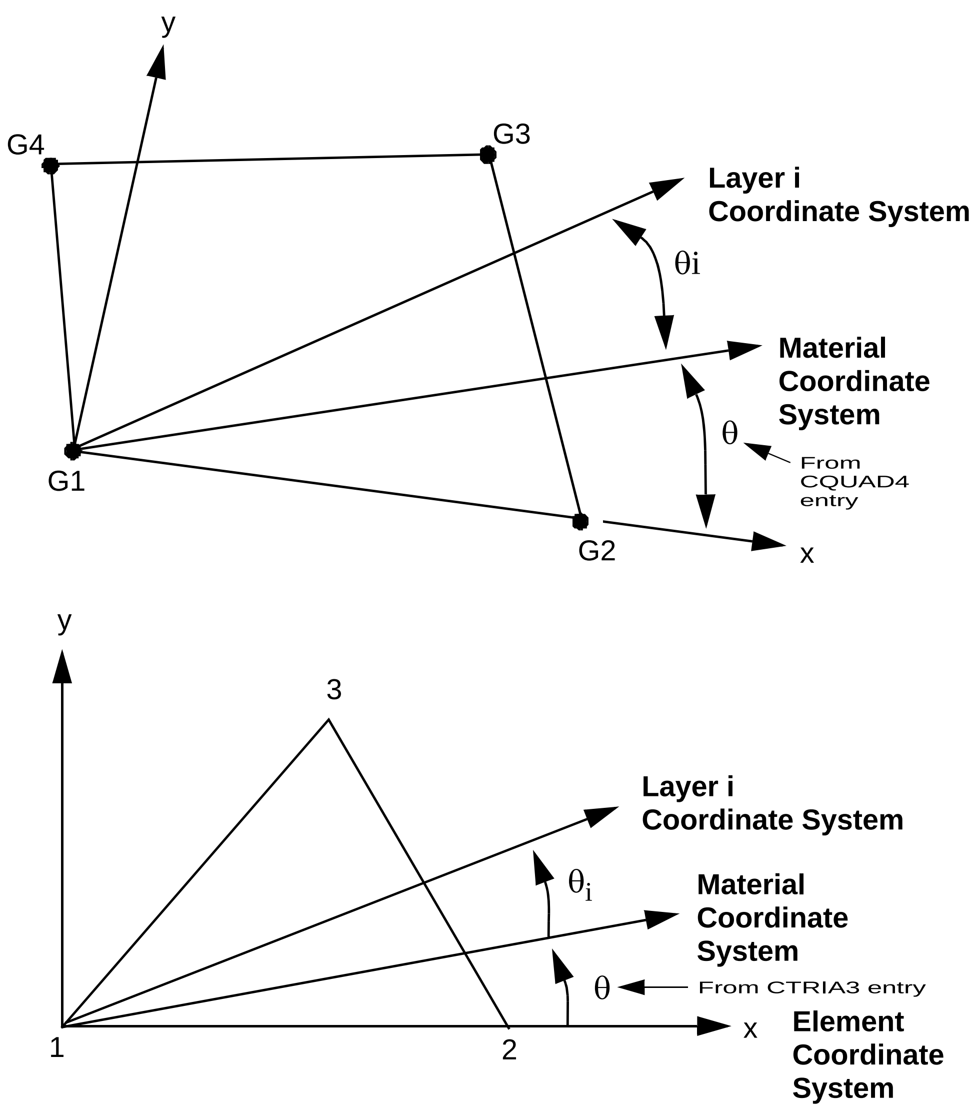




Typical case: Reference Plane = Mid-plane

$Z_0 = -t/2$ $\quad Z_{offset} = 0$

Figure 6-51



y
G4
G3
Layer i
Coordinate System
θi
Material
Coordinate
System
θ
From
CQUAD4
entry
G1
G2
x
y
3
Layer i
Coordinate System
θi
Material
Coordinate
System
θ
From CTRIA3 entry
Element
Coordinate
System
1
2
x

Figure 6-52

6

6.7.113 PCOMPG

Data Entry: **PCOMPG** - Layered Composite Element Property.

Description: Used to define the properties of an n-ply composite material laminate including global ply IDs.

Format:

1	2	3	4	5	6	7	8	9	10
PCOMPG	PID	Z_0	NSM	SB	F.T.	TREF	GE	LAM	MEM
+	GPID1	MID1	T1	θ_1	SOUT1				
+	GPID2	MID2	T2	θ_2	SOUT2				
+	-etc.-								

Example:

1	2	3	4	5	6	7	8	9	10
PCOMPG	311	-0.15	6.85		H0FF				
+	1001	86	0.056	0.0	YES				
	1002			45.0					
	1003			-45.0					
	1004			90.0					

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	Z_0	Offset from the reference plane to the bottom surface (Real or Blank. Blank if LAM=SYM), (Default = -1/2 the thickness of the element)
4	NSM	Non-structural mass per unit area (Real > 0.0 or Blank).
5	SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress) (Real > 0.0 or Blank).
6	F.T.	Failure theory to be used to test whether the element fails or not (Character, Integer > 0 or blank). If character, the following theories are allowed: HILL for the Hill theory HOFF for the Hoffman theory TSAI for the Tsai-Wu theory STRN for the maximum strain theory If Integer, user supplied failure index equation identification number specified in **FINDEX** or **FINDEXN**.
7	TREF	Reference temperature (Real or Blank).
8	GE	Damping coefficient (Real or Blank).

6

9	LAM	Symmetric lamination option (Character or Blank). If blank, all plies must be entered. If set to SYM or SYM1, describe only plies on one side of element centerline. If set to SYM, the Z_0 is always 1/2 the total thickness of the element.
10	MEM	Membrane properties only. (Character or Blank). If set to MEM, then only the membrane stiffness will be used. If this field is blank, then the bending, membrane and bending-membrane coupling stiffness will be used. See Remark 7.
2	GPIDi	Global Ply ID for the various plies. (Integer > 0).
3	MIDi	Material ID for the various plies. MID1 > 0, MID2, MID3, ... MIDn > 0 or Blank. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDs must refer to **MAT1**, **MAT2** or **MAT8** data statements. (MID1: Integer > 0. All others, Integer > 0 or Blank). See Remark 1 for defaults.
4	Ti	Thickness of the various plies (Real: T1 > 0.0, T2, T3, ... > 0.0 or Blank). See Remark 1 for default.
5	θ_i	Orientation angle of the longitudinal direction of each ply with the material axis of the element. If the material angle on the element connection statement is 0.0, the material axis and side 1-2 of the element coincide. The plies are to be numbered serially, starting with 1 at the bottom layer (the bottom layer is defined as the surface with the largest -Z value in the element coordinate system). (Real, Default = 0.0).
6	SOUTi	Stress or strain output required (YES) or not (NO) for the various plies or for the maximum ply only (IFMAX). (Character). (Default = NO). See Remark 8.

Remarks:

1. The default under MID2, MID3, ... is the last defined MAT statement. In the example shown, MID1 through MID4 are equal to 86. This also applies to the Ti values.
2. At lease one of the four values (MIDi, Ti, θ_i and SOUTi) must be present for a ply to exist. The minimum number of plies is one.
3. TREF given on the PCOMPG statement will be used for all plies of the element. It will override values supplied on material data statements for individual plies.
4. GE given on the PCOMPG data statement will be used for the element. It will override values supplied on material data statements for individual plies.
5. If transverse shear calculations are required, the G1,Z and G2,Z must be supplied in all MAT8 data referenced by the PCOMPG data. If G1,Z and G2,Z in the MAT8 data are left blank, then there is no transverse shear flexibility.
6. PCOMPG property identification numbers must be unique with respect to **PCOMP**, **PSHELL** and **PSKIN** property identification numbers. It is recommended that property identification numbers be unique with respect to all other property identification numbers.

7. If MEM is entered in field 10, then the element stiffness matrix is geometrically condensed to remove the transverse displacement and the two bending rotational degrees of freedom. The drilling rotation is condensed out using static condensation. This method produces a stiffness matrix with two degrees of freedom per node in the element coordinate system. In general, in the global coordinate system, these will be three degrees of freedom per node. PARAM AUTOSPC should be set to YES to eliminate the transverse displacement.
8. When SOUTi = YES:
   For the HILL, HOFF, TSAI and **FINDEX** equation options, the failure index is printed in the output file with the composite stresses (STRESS = ALL or any appropriate options). For the STRN or **FINDEXN** equation options, the failure index is printed in the output file with the composite strains (STRAIN = ALL or any appropriate options).
9. See **Composite Elements (CQUAD4, CTRIA3, CQUAD8 and CTRIA6 referencing PCOMP/PCOMPG data)** (Sec. 2.4.6) for a description of the failure theories.
10. A positive value for Z_0 puts the bottom surface of the elements above the reference plane. A negative value for Z_0 puts the bottom surface of the elements below the reference plane. See the figure below:
11. Interlaminar shear stresses and strains are calculated only when G1,Z and G2,Z are nonzero in all MAT8 data referenced by the PCOMPG data. Additionally, when SB is nonzero, a bonding material failure index is calculated as $FB = \frac{max(|\sigma_{1z}|, |\sigma_{2z}|)}{SB}$.

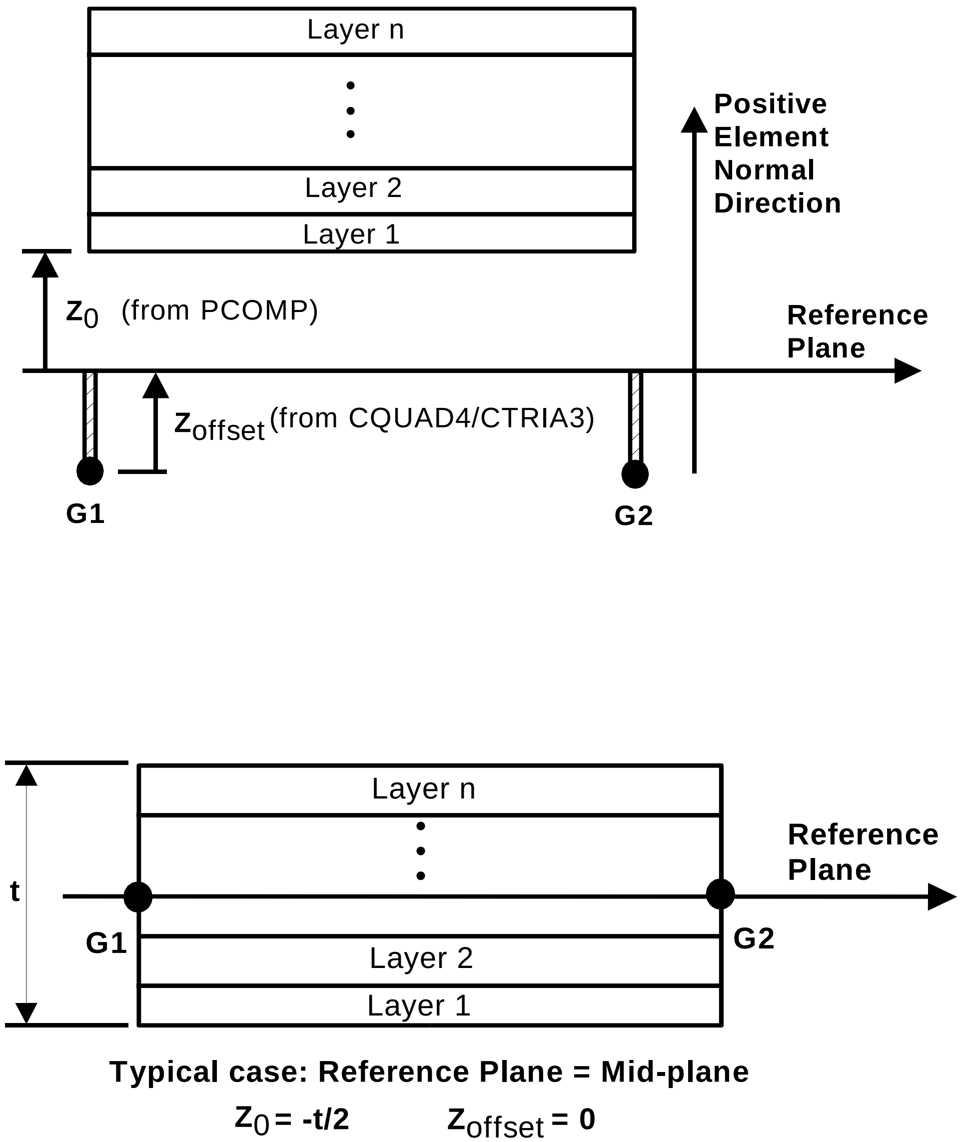
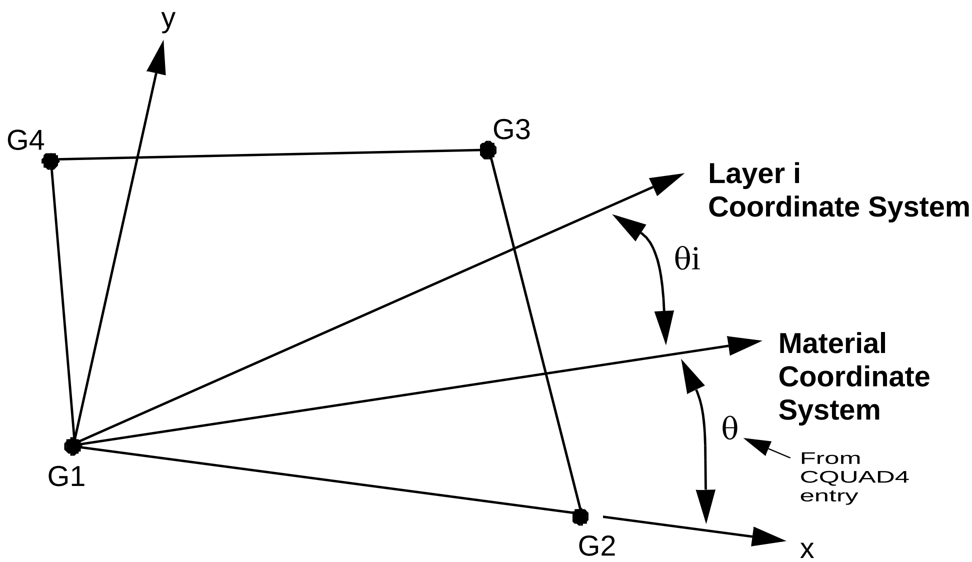

Layer n
Layer 2
Layer 1
Positive
Element
Normal
Direction
Z_0 (from PCOMP)
Reference
Plane
Z_{offset} (from CQUAD4/CTRIA3)
G1
G2
Layer n
Reference
Plane
t
G1
G2
Layer 2
Layer 1
Typical case: Reference Plane = Mid-plane
$Z_0 = -t/2$
$Z_{offset} = 0$

Figure 6-53


y
G4
G3
Layer i
Coordinate System
θi
Material
Coordinate
System
θ
From
CQUAD4
entry
G1
G2
x

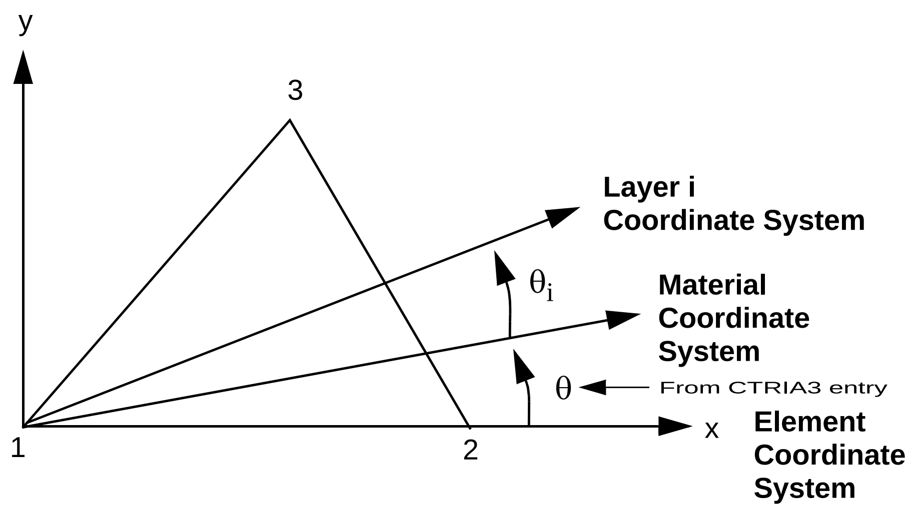

y
3
Layer i
Coordinate System
θi
Material
Coordinate
System
θ
From CTRIA3 entry
1
2
x
Element
Coordinate
System

Figure 6-54

6.7.114 PCONM3

Data Entry: **PCONM3 -** Concentrated Mass Element Property.

Description: Used to define a concentrated mass at a grid point of the structural model which is referenced by **CONM3** data.

Format:

1	2	3	4	5	6	7	8	9	10
PCONM3	PID	CID	M	X1	X2	X3			
+	I11	I21	I22	I31	I32	I33			

Example:

1	2	3	4	5	6	7	8	9	10
PCONM3	11	21	220.	3.1	3.5	1.33			
+	4010.0								

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	CID	Coordinate system identification number (Integer $\geq$ -1 or blank). A value of 0 or blank implies the basic coordinate system (see remark 4).
4	M	Mass value (Real > 0).
5-7	X1,X2,X3	Offset distances from the grid point to the center of gravity of the mass in the coordinate system defined in field 3, unless CID = -1, in which case X1, X2, X3 are the coordinates, not offsets, of the center of gravity of the mass in the basic coordinate system (Real or blank. Default = 0.0).
2-7	Iij	Mass moments of inertia measured at the mass center of gravity in the coordinate system defined by field 3 (Diagonal terms should be Real $\geq$ 0.0. the rest may be Real or blank. Default = 0.0). If CID = -1, the basic coordinate system is used.

Remarks:

1. It is recommended that property identification numbers be unique with respect to all other property identification numbers.
2. The **CONM2** information cannot be updated in the structural optimization because it has no property data. To use masses as design elements, use the **CONM3** data.
3. The continuation may be omitted. This implies zero values for the mass moments of inertia.

6

4. If CID = -1 in field 3, offsets are internally computed as the difference between the grid point location and X1, X2, X3. In this case, the values of Iij are in a coordinate system that parallels the basic coordinate system. The grid point location may be defined in a nonbasic coordinate system.
5. The form of the inertia matrix about its center of gravity is taken as:

$$\begin{bmatrix} M & & & & & \\ & M & & & \text{SYM} & \\ & & M & & & \\ & & & I11 & & \\ & & & -I21 & I22 & \\ & & & -I31 & -I32 & I33 \end{bmatrix}$$

where:

$$M = \int \rho dV$$

$$I11 = \int \rho (x_2^2 + x_3^2) dV$$

$$I22 = \int \rho (x_1^2 + x_3^2) dV$$

$$I33 = \int \rho (x_1^2 + x_2^2) dV$$

$$I21 = \int \rho x_1 x_2 dV$$

$$I31 = \int \rho x_1 x_3 dV$$

$$I32 = \int \rho x_2 x_3 dV$$

and x_1, x_2, x_3 are components of distance of a point in the mass to the center of gravity of the mass. These coordinates are measured in the coordinate system defined in field 3. The negative signs for the off-diagonal terms are supplied by the program.
6. If lumped mass is used (PARAM, COUPMASS, NO) then the mass offset and I_{21}, I_{31} and I_{32} terms will be ignored. If a mass offset is specified, a warning message will be issued.

6

6.7.115 PCONV

Data Entry: **PCONV** - Convection Boundary Condition Property.

Description: Defines the property for convection boundary condition.

Format:

1	2	3	4	5	6	7	8	9	10
PCONV	PCONID	MID							

Example:

1	2	3	4	5	6	7	8	9	10
PCONV	3	2							

Alternate Format:

1	2	3	4	5	6	7	8	9	10
PCONV	PCONID				3	H			

Example:

1	2	3	4	5	6	7	8	9	10
PCONV	4				3	15.63			

Field	Information	Description
2	PCONID	Unique property identification number (Integer > 0).
3	MID	Material identification number (Integer > 0).
6	“3”	Indicates that the alternate format is used. See Remark 3.
7	H	Convective film coefficient (Real). See Remark 3.

6

Remarks:

1. PCONV is referenced by **CONV** entry to define the property for convective boundary condition on CHBDYE, CHBDYG and CHBDYP elements.
2. MID must refer to a **MAT4** entry. The MAT4 should be in its alternate format to provide the convective film coefficient, H.
3. If field 6 is integer 3, MID is ignored and can be blank. The convective film coefficient, H, must be provided in field 7.

6.7.116 PDAMP

Data Entry: **PDAMP** - Scalar Viscous Damper Property

Description: Used to define the damping value of a scalar damper element which is referenced by the **CDAMP1** data.

Format:

1	2	3	4	5	6	7	8	9	10
PDAMP	PID	B	PID	B	PID	B	PID	B	

Example:

1	2	3	4	5	6	7	8	9	10
PDAMP	10	2.3	8	6.2					

Field	Information	Description
2, 4, ...	PID	Unique property identification number (Integer > 0).
3, 5, ...	B	Value of scalar damper (Real).

Remarks:

1. This data defines a damper value. A structural viscous damper, **CVISC**, may also be used for geometric grid points.
2. Up to four damper properties may be defined on a single entry.
3. Property identification numbers must be unique with respect to all other property identification numbers.

6

6.7.117 PELAS

Data Entry: **PELAS** - Scalar Elastic Structural Property

Description: Used to define the stiffness, damping and stress coefficient of a scalar element (spring) which is referenced by the **CELAS1** data.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
PELAS	5	10.0+6	.01	50.0	6	20.0+6	.01	50.0	

Field	Information	Description
2, 6	PID	Unique property identification number (Integer > 0).
3, 7	K	Elastic stiffness property value (Real).
4, 8	GE	Damping coefficient (Real or Blank).
5, 9	SRC	Stress recovery coefficient (Real or blank). Stress = SRC*Force.

Remarks:

1. Property identification numbers must be unique with respect to all other property identification numbers.
2. The user is cautioned to be careful using negative spring values.
3. One or two elastic spring properties may be defined on a single data entry.
4. For heat transfer analysis, use **PELASH** data.
5. To avoid numerical errors caused by ill-conditioning, the stiffness K value is limited by the value of **PARAM ELASSTIF**.

6.7.118 PELASH

Data Entry: **PELASH** - Scalar Conductive Property

Description: Used to define the conductivity of a scalar element which is referenced by the **CELAS1** data.

Format:

1	2	3	4	5	6	7	8	9	10
PELASH	PID	K			PID	K			

Example:

1	2	3	4	5	6	7	8	9	10
PELASH	5	70.0			6	78.0			

Field	Section	Description
2, 6	PID	Unique property identification number (Integer > 0).
3, 7	K	Scalar conduction property value (Real).

Remarks:

1. Property identification numbers must be unique with respect to all other property identification numbers.
2. The user is cautioned to be careful using negative conduction values.
3. One or two scalar conduction properties may be defined on a single data entry.
4. PELASH data is used for heat transfer analysis. For structural analysis, use **PELAS** data.

6

6.7.119 PGAP

Data Entry: **PGAP** - Gap Element Property.

Description: Used to define properties of a gap element.

Format:

1	2	3	4	5	6	7	8	9	10
PGAP	PID	U0	F0	KA	KB	KT	MU1		

Example:

1	2	3	4	5	6	7	8	9	10
PGAP	1001	0.05		2.6E6	1.4E6	4.3E9			

Field	Information	Description
2	PID	Unique property identification number for a **CGAP** element (Integer > 0).
3	U0	Inital gap opening (Real. Default=0.0).
4	F0	Preload force (Real>=0.0. Default=0.0).
5	KA	Axial stiffness values for the closed gap: UA-UB>=U0. KA is measured in the element coordinate system. (Real>0.0)
6	KB	Axial stiffness values for the open gap: UA-UB<U0. KB is measured in the element coordinate system. (Real>=0.0. Default = 1.0E-14*KA).
7	KT	Transverse stiffness values for the closed gap: UA-UB>=U0. (Real>=0.0. Default = MU1*KA)
8	MU1	Coeficient of static friction (Real>=0.0. Default =0.0)

Remarks:

1. Nonlinearity is activated in a static loadcase with the **NLPARM** solution control command.
2. The force in the gap element is determined as (positive is compression):

$$F = \begin{cases} F0 + KA(UA - UB - U0) & UA - UB \geq U0 \\ F0 + KB(UA - UB - U0) & UA - UB < U0 \end{cases}$$

   where UA is the displacement of grid GA in the element x direction and UB is the displacement of grid GB in the element x direction.
3. In linear analysis, the gap element stiffness is computed assuming UA=UB=0.0.

6

6.7.120 PHBDY

Data Entry: **PHBDY** - Heat Boundary Element Property.

Description: Defines the properties of the **CHBDY** and **CHBDYP** element.

Format for use with CHBDY:

1	2	3	4	5	6	7	8	9	10
PHBDY	PID	MID	AF		ALPHA	D1	D2		

Example:

1	2	3	4	5	6	7	8	9	10
PHBDY	17	29	100.0						

Format for use with CHBDYP:

1	2	3	4	5	6	7	8	9	10
PHBDY	PID	AF	D1	D2					

Example:

1	2	3	4	5	6	7	8	9	10
PHBDY	2	35.0							

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	MID	Material identification number (Integer > 0 or Blank). Used for convective film coefficient.
4	AF	Area factor (Real $\geq$ 0 or Blank). Used only for HBDY types “POINT” and “LINE”.”
6	ALPHA	Absorptivity (0.0 $\leq$ Real $\leq$ 1.0 or Blank) Used only for thermal vector flux calculations. Default is 0.0.
7, 8	D1, D2	Diameters of elliptic cylinder. Used for HBDY type “ELCYL.” See the HBDY element description (Real).

Remarks:

1. The referenced material ID must be on a **MAT4** data statement. The statement defines the convective film coefficient **per unit area**. If no material is referenced, the element convection is zero.
2. The area factor, AF, is used to determine the effective area. For a “POINT”, AF=area; for “LINE”, AF=effective width where area=AF*length.

3. Property identification numbers must be unique with respect to all other property identification numbers.

6.7.121 PK2UU

Data Entry: **PK2UU** - K2UU Stiffness Multiplier

Description: Used to define the stiffness multiplier of an element stiffness matrix defined by the **K2UU1** executive control command.

Format:

1	2	3	4	5	6	7	8	9	10
PK2UU	PID	KMULT							

Example:

1	2	3	4	5	6	7	8	9	10
PK2UU	5	1.0							

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	KMULT	Stiffness multiplier (Real).

Remarks:

1. Property identification numbers must be unique with respect to all other property identification numbers.
2. The user is cautioned to be careful using negative values.

6

6.7.122 PLOAD1

Data Entry: **PLOAD1** - Applied Loads on BAR or BEAM Elements.

Description: Defines uniformly distributed or linearly distributed applied loads or a point load to a **CBAR** or **CBEAM** element over part of or the entire length of the element.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	

Example 1: Distributed load over the entire length of the element.

1	2	3	4	5	6	7	8	9	10
PLOAD1	9	3	FZ			10.0		10.0	

Example 2: Distributed load over a part of the length of the element.

1	2	3	4	5	6	7	8	9	10
PLOAD1	9	4	FY	FRPR	0.2	20.0	0.6	20.0	

Example 3: Point load.

1	2	3	4	5	6	7	8	9	10
PLOAD1	9	5	FY	FR	0.5	100.0			

Field	Information	Description
2	SID	Load set identification number (Integer > 0).
3	EID	**CBAR** or **CBEAM** element identification number (Integer > 0).
4	TYPE	Load type, one of the following values: FX, FY, FZ, FXE, FYE, FZE, MX, MY, MZ, MXE, MYE, MZE.
5	SCALE	Scale factor for X1, X2. One of the values: LE, FR, LEPR, FRPR or Blank.
6,8	X1, X2	Distances (or fractional distances) along CBAR or CBEAM from end A. Real $\geq$ 0, X1 $\leq$ X2 or blank.
7,9	P1, P2	Load factors at end A or X1 and B or X2 (Real or blank. Default = 0.0). Both cannot be 0.0 or blank.

Remarks:

1. If P1 = P2, a uniform distributed load of intensity per unit length equal to P1 will be applied from X1 to X2.

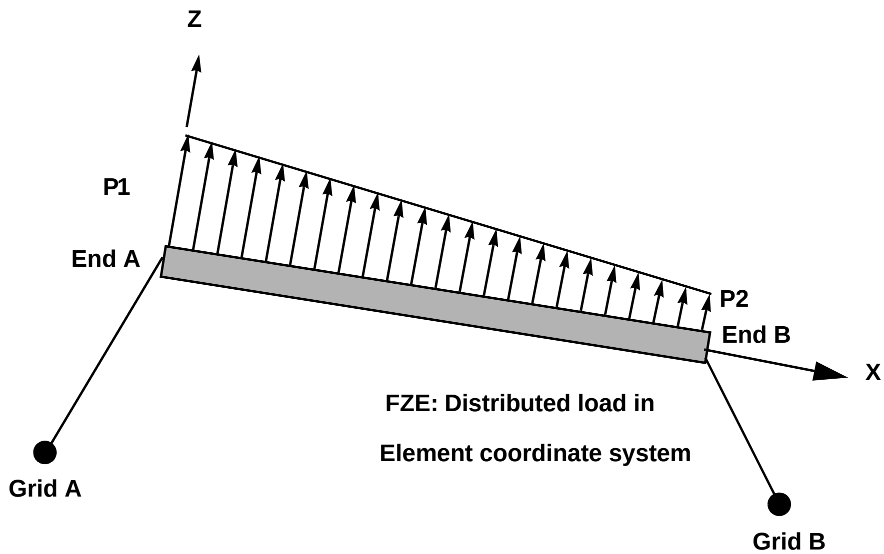
6

2. Load TYPE symbols are used as follows to define loads:

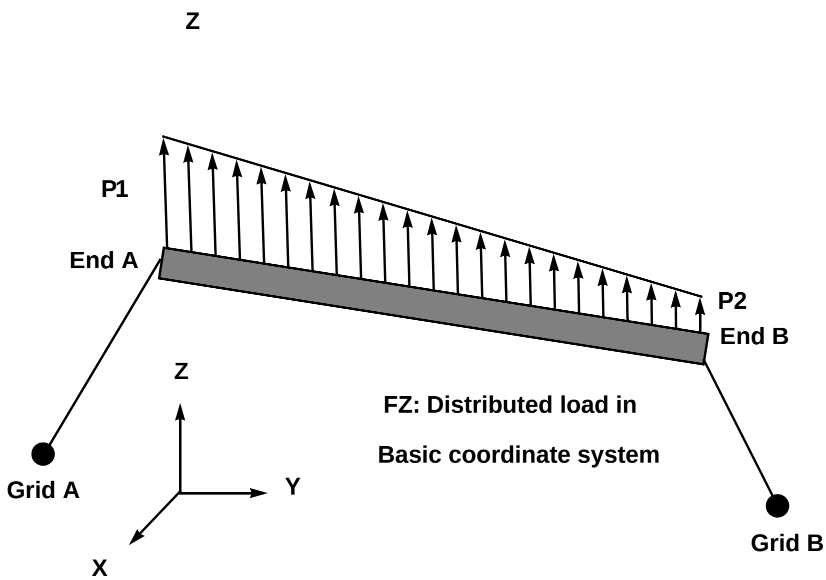
FX, FY or FZ:	Force in the x, y, or z direction of the basic coordinate system.
MX, MY or MZ:	Moment in the x, y, or z direction of the basic coordinate system.
FXE, FYE or FZE:	Force in the x, y, or z direction of the element's coordinate system.
MXE, MYE or MZE:	Moment in the x, y, or z direction of the element's coordinate system.

3. Load sets can be selected in the Solution Control Section (**LOAD** = SID) or through **RLOAD1**, **RLOAD2** or **LOAD** bulk data entries.
4. For distributed loads, the user must be aware that if the length of the bar changes during optimization, the total load applied to it will also change.
5. If SCALE is blank, then the pressure load is applied over the entire beam.
6. SCALE symbols are used to define X1 and X2 as follows.
   - LE: X1 and X2 are actual lengths.
   - FR: X1 and X2 are fractions of the length of the bar.
   - LEPR: X1 and X2 are actual lengths and if $X1 \neq X2$, the distributed load P1 and P2 is input in terms of the projected length of the element.
   - FRPR: X1 and X2 are fractions of the length and if $X1 \neq X2$, the distributed load P1 and P2 is input in terms of the projected length of the element.
7. If X2 is blank or equal to X1, then a concentrated load of value P1 will be used at X1 (or X1*Length).
8. If X1 < X2, a linearly varying distributed load will be applied to the element, starting from X1 (or X1*Length) and ending at X2 (or X2*Length), having an intensity of P1 at X1 (or X1*Length) and P2 at X2 (or X2*Length).
9. Pressure loads are applied in the shear axis of CBEAM.

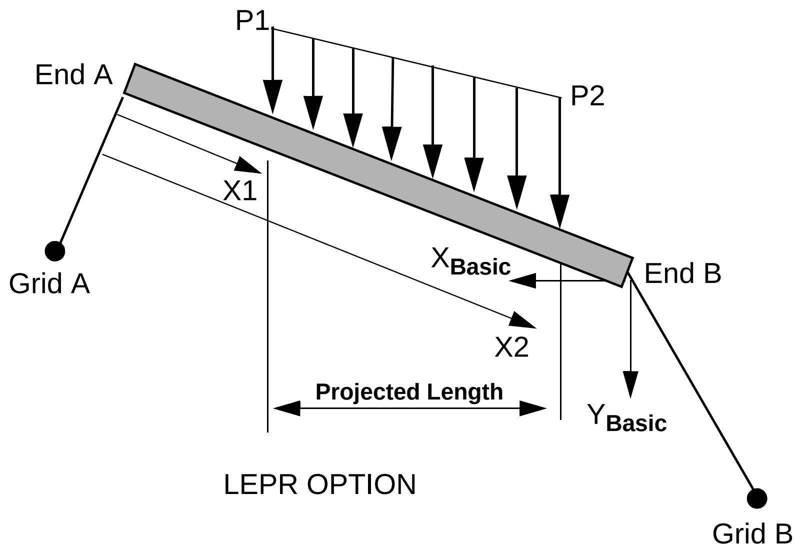
NOTE: Length is the distance between ends A and B, not the distance between grids A and B.



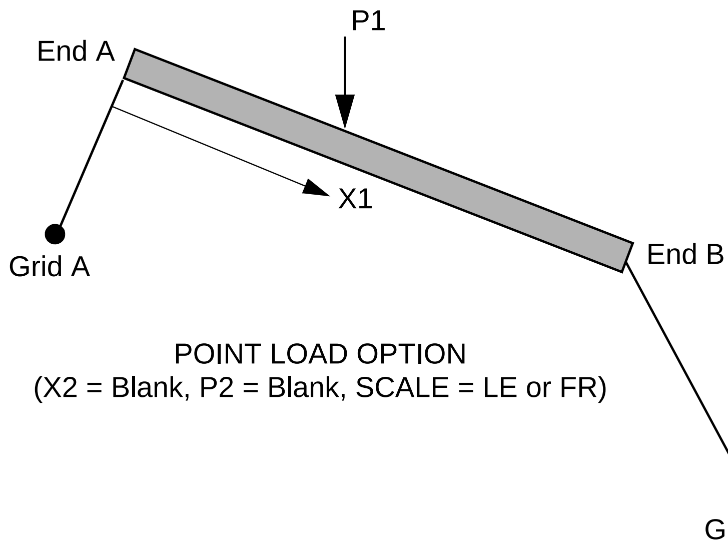
Z
P1
End A
P2
End B
X
FZE: Distributed load in
Element coordinate system
Grid A
Grid B



Z
P1
End A
P2
End B
Z
FZ: Distributed load in
Basic coordinate system
Y
X
Grid A
Grid B



LEPR OPTION



POINT LOAD OPTION
(X2 = Blank, P2 = Blank, SCALE = LE or FR)

6.7.123 PLOAD2

Data Entry: **PLOAD2** - Normal Surface Pressure Load on a Two-Dimensional Structural Element.

Description: Defines a uniform normal static pressure load applied to two-dimensional elements. Only **CTRIA3**, **CQUAD4**, **CTRIA6**, **CQUAD8** or **CSHEAR** elements may have a pressure load applied to them via this data.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD2	SID	P	EID	EID	EID	EID	EID	EID	
+	EID	EID	EID	-etc.-					

Example:

1	2	3	4	5	6	7	8	9	10
PLOAD2	5	800.0	2	4	6	8	10	12	
+	14								

Alternate Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
PLOAD2	1	1100.0	1	THRU	100				

6

Field	Information	Description
2	SID	Load set identification number (Integer > 0).
3	P	Pressure value (Real).
4...	EID	Element identification number (Integer > 0).
4,6	EID1, EID2	Element identification number (Integer > 0. EID1 < EID2).

Remarks:

1. EID after the first omitted entry are not allowed and will produce a fatal error.
2. Load sets can be selected in the Solution Control Section (**LOAD**=SID) or through **RLOAD1**, **RLOAD2** or **LOAD** bulk data entries.
3. The direction of the pressure is computed according to the right-hand rule using the grid point sequence specified on the element data.
4. All elements referenced must exist.

5. Continuation data are allowed.
6. The load intensity is the load per unit surface area.
7. Since this load is per unit surface area, the user must be aware that if the size of the TRIA3, QUAD4, TRIA6, QUAD8 or SHEAR element changes during optimization, the total load applied to it will also change.

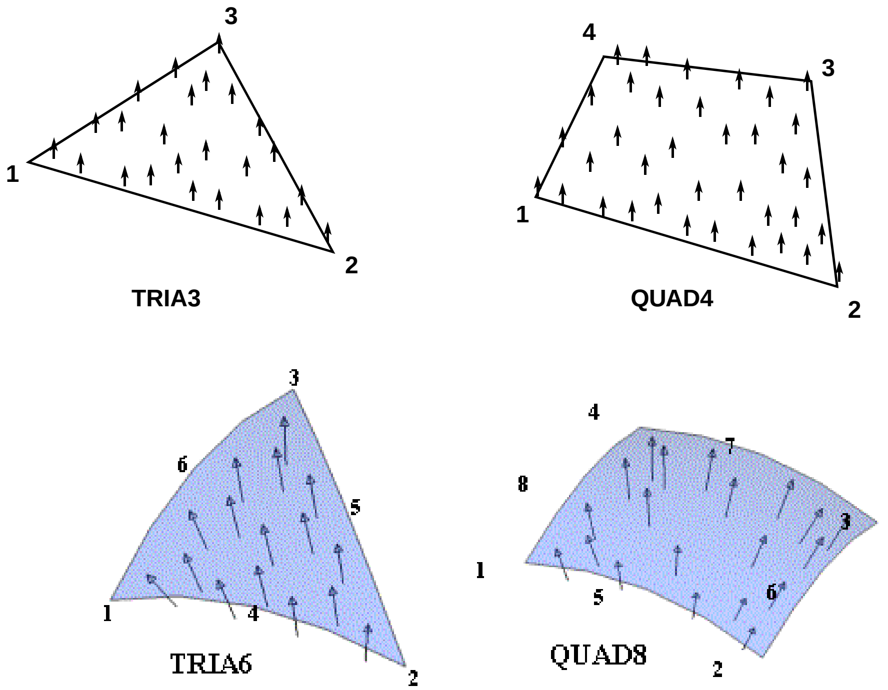


Figure 6-55

6.7.124 PLOAD4

Data Entry: **PLOAD4** - Pressure Loads on Face of 2D or 3D Elements.

Description: Defines a load on a face of a **CTRIA3**, **CQUAD4**, **CTRIA6**, **CQUAD8**,**CSHEAR**, **CHEXA**, **CHEX20**, **CPENTA**,**CPYRA**,**CTETRA** or **CTETRA** element.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P				G1	G3 or G4	
+	CID	N1	N2	N3					

Example:

1	2	3	4	5	6	7	8	9	10
PLOAD4	2	8	600.0				7	9	
+	1	1.0	0.0	0.0					

Alternate Format:

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P				“THRU”	EID2	
+	CID	N1	N2	N3					

Example:

1	2	3	4	5	6	7	8	9	10
PLOAD4	1	18	500.0				THRU	29	
+		1.0	1.0	1.0					

6

Field	Information	Description
2	SID	Load set identification number (Integer > 0).
3	EID	Element identification number (Integer > 0).
4	P	Load per unit surface area (pressure) on the face of the element (Real).
8	G1	Identification number of a **GRID** connected to a corner of the face (Integer > 0).

9	G3	Identification number of a GRID connected to a corner diagonally opposite to G1 on the same face of a CHEXA, CHEX20 or CPENTA element. Required data for quadrilateral faces of CHEXA, CHEX20 and CPENTA elements only (Integer > 0 or blank). G3 must be omitted for a triangular surface on a CPENTA element. For triangular faces of CPYRA elements, G3 is the identification of a GRID shared by both the triangular face and quadrilateral face. For quadrilateral face of CPYRA element, G3 is the identification number of the GRID connected to a corner diagonally opposite to G1 on the quadrilateral face.
9	G4	Identification number of the CTETRA GRID located at the corner not on the face being loaded. This is required data for CTETRA elements only. (Integer > 0 or blank)
2	CID	Coordinate system identification number (Integer $\geq$ 0 or blank).
3-5	N1,N2,N3	Components of vector measured in coordinate system defined by CID (Real or blank. Default = 0.0). If present, at least one component must be non-zero. Used to define the direction (but not the magnitude) of the load intensity.
Alternate Format		
8	"THRU"	(Character or Blank)
9	EID2	Element identification number (Integer > 0 or Blank: EID2 > EID),

Remarks:

1. If field 8 is "THRU" or blank, then EID must reference a CTRIA3, CQUAD4, CTRIA6, CQUAD8 or CSHEAR element. If field 8 is integer, EID must reference a CHEXA, CHEX20, CPENTA, CPYRA or CTETRA element.
2. The continuation data is optional. If fields 2, 3, 4 and 5 of the continuation data are blank, the load is assumed to be a pressure acting normal to the face. If these fields are not blank, the load acts in the direction defined in these fields. The load intensity is the load per unit of surface area, not the load per unit of area normal to the direction of loading.
3. The direction of positive normal pressure (defaulted continuation data) is inward for 3D elements or according to the right-hand rule using the grid point sequence specified on the element data for 2D elements.
4. Equivalent grid point loads are computed by linear (or bilinear) interpolation of load intensity, followed by numerical integration using isoparametric shape functions. Note that a uniform load intensity will not necessarily result in equal equivalent grid point loads.
5. For triangular faces of CPENTA elements, G1 is an identification number of a corner grid point that is on the face being loaded and the G3/G4 field is left blank.

6. For faces of CTETRA elements, G1 is an identification number of the corner grid point that is <u>on</u> the face being loaded, and G4 is an identification number of the corner point that is <u>not on</u> the face being loaded. Since a CTETRA has only four corner points, this point, G4, will be unique and different for each of the four faces of a CTETRA element.
7. Load sets can be selected in the Solution Control Section (**LOAD** = SID) or through **RLOAD1**, **RLOAD2** or **LOAD** bulk data entries.
8. Since this load defines a pressure, the user must be aware that if the size of the face of the element on which the load is applied changes during optimization, the total load applied to it will also change.
9. The “THRU” and EID2 entries are optional.
10. The referenced elements must exist.
11. To use pressure loads that are only applied normal to the surface, the PARAMeter PLOADM should be set to 0. The pressure load is computed then using the component of the pressure vector (N_1,N_2,N_3) that is normal to the surface.

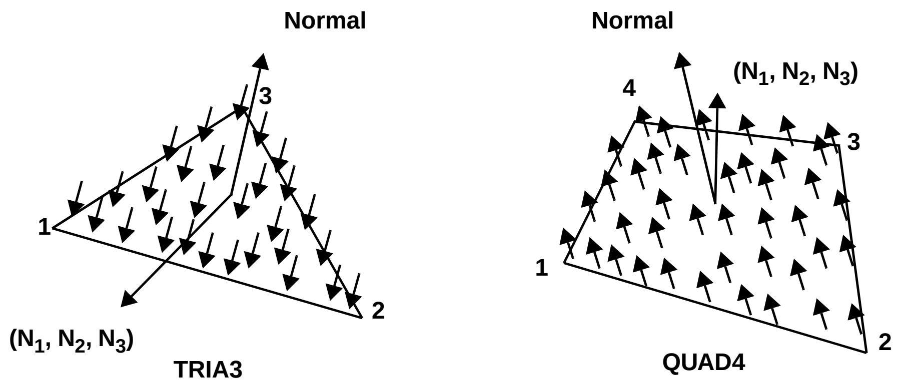


Figure 6-56

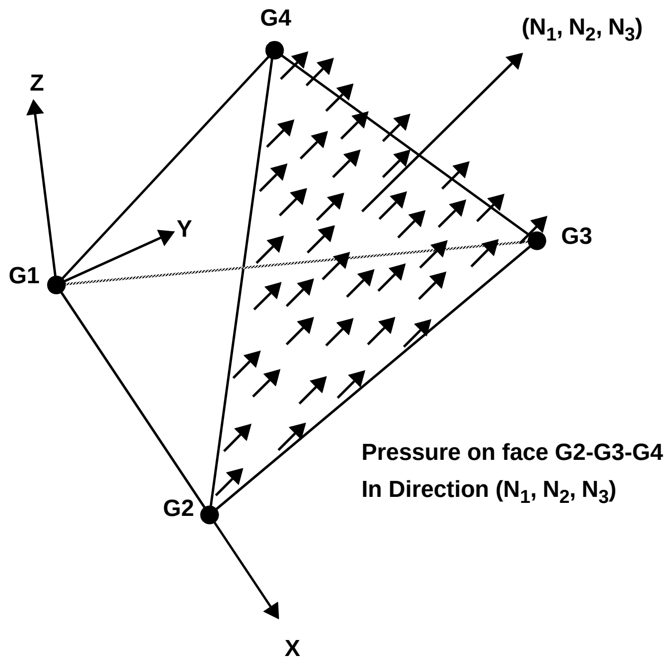

G4
(N_1, N_2, N_3)
Z
Y
G1
G3
G2
X
Pressure on face G2-G3-G4
In Direction (N_1, N_2, N_3)

Figure 6-57

6.7.125 PLOAD5

Data Entry: **PLOAD5** - Pressure Loads on Face of Structural Elements.

Description: Defines loads on the faces of a **CTRIA3**, **CQUAD4**, **CTRIA6**, **CQUAD8** or **CSHEAR** elements.

Format:

1	2	3	4	5	6	7	8	9	10
PLOAD5	SID	EID1	P				"THRU"	EID2	
+	CID	N1	N2	N3					

Example:

1	2	3	4	5	6	7	8	9	10
PLOAD5	1	18	500.0						
+		1.0	1.0	1.0					

Field	Information	Description
2	SID	Load set identification number (Integer > 0).
3	EID1	Element identification number (Integer > 0).
4	P	Load per unit surface area (pressure) on the face of the element (Real).
8	"THRU"	Word "THRU" or Blank.
9	EID2	Element identification number (Integer > 0; EID2 > EID1), (Optional)
2	CID	Coordinate system identification number (Integer $\geq$ 0 or blank).
3-5	N1,N2,N3	Components of vector measured in coordinate system defined by CID (Real or blank). At least one component must be non-zero. Used to define the direction (but not the magnitude) of the load intensity.

Remarks:

1. **PLOAD4** has more capabilities, and is preferred over PLOAD5.
2. The load intensity is the load per unit of surface area, not the load per unit of area normal to the direction of loading.
3. Equivalent grid point loads are computed by linear numerical integration using isoparametric shape functions. Note that the uniform load intensity will not necessarily result in equal equivalent grid point loads.
4. Normal loads cannot be applied to the 2-D elements via this entry. Use the **PLOAD2** entry.

6

5. The continuation entry is required.
6. The “THRU” and EID2 entries are optional.
7. The referenced elements must exist.
8. Loads can be selected in the Solution Control Section (**LOAD** = SID) or through **RLOAD1**, **RLOAD2** or **LOAD** bulk data entries.
9. To use pressure loads that are only applied normal to the surface, the PARAMeter PLOADM should be set to 0. The pressure load is computed then using the component of the pressure vector (N_1,N_2,N_3) that is normal to the surface.
10. Since this load is per unit surface area, the user must be aware that if the size of the face of the CTRIA3, CQUAD4, CTRIA6, CQUAD8 or CSHEAR element on which the load is applied changes during optimization, the total load applied to it will also change.

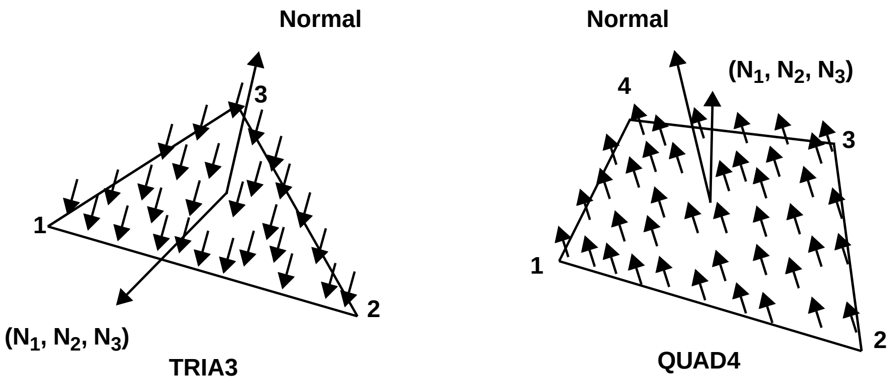


Figure 6-58

6.7.126 PLOADA

Data Entry: **PLOADA** - Pressure Loads on BAR or BEAM Elements.

Description: Defines a uniformly distributed, or linearly distributed applied loads, in any coordinate system to **CBAR** or **CBEAM** elements over the entire length of the element.

Format:

1	2	3	4	5	6	7	8	9	10
PLOADA	SID	EID	CID	X1	X2	X3	PA	PB	

Example:

1	2	3	4	5	6	7	8	9	10
PLOADA	2	10	1000	0.0	1.0	0.0	6.0	6.0	

Field	Information	Description
2	SID	Load set identification number (Integer > 0).
3	EID	Element identification number (Integer > 0).
4	CID	Coordinate system identification number (Integer $\geq$ 0 or blank).
5-7	X1, X2, X3	Components of a vector that define the direction of the applied load (Real or blank). At least one component must be non-zero.
8, 9	PA, PB	Load factors at grids A and B (Real or blank). At least one must be non-zero.

Remarks:

1. If PA = PB, a uniform distributed load of intensity per unit length equal to PA will be applied.
2. PLOADA can be selected in the solution control section by **LOAD** = SID or through **RLOAD1**, **RLOAD2** or **LOAD** bulk data entries.
3. Since this load is per unit length, the user must be aware that if the length of the bar element on which the load is applied changes during optimization, the total load applied to it will also change.
4. Pressure loads are applied in the shear axis of **CBEAM**.

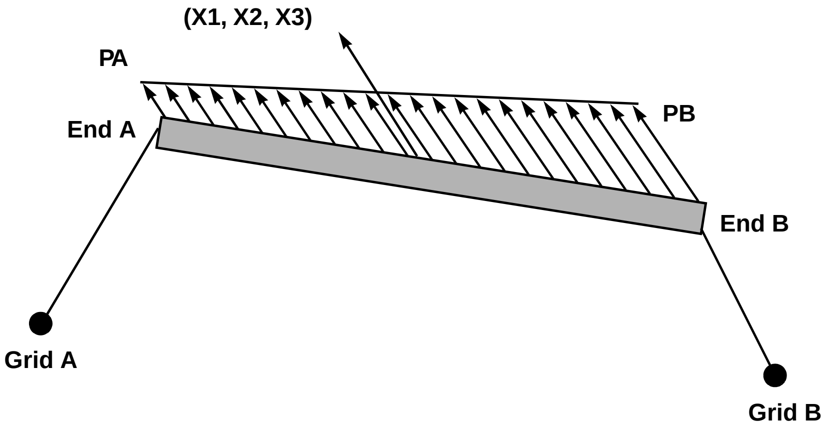


Figure 6-59

6.7.127 PLOADX1

Data Entry: **PLOADX1** - Normal Surface Pressure Loads axisymmetric Elements (TRIAX6).

Description: Defines surface traction to be used with **CTRIAX6** axisymmetric element.

Format:

1	2	3	4	5	6	7	8	9	10
PLOADX1	SID	EID	PA	PB	GA	GB	THETA		

Example:

1	2	3	4	5	6	7	8	9	10
PLOADX1	20	10	3.2	10.5	10	12	30		

Field	Information	Description
2	SID	Load set identification number (Integer > 0). See Remark 1.
3	EID	Element identification number (Integer > 0). See Remark 6.
4	PA	Surface traction at grid point GA. (Real).
5	PB	Surface traction at grid point GB. (Real).
6, 7	GA, GB	Corner grid points. GA and GB are any two adjacent corner grid points of the element (Integer > 0).
8	THETA	Angle between surface traction and inward normal to the line segment. (Real; Default = 0.0).

Remarks:

1. The load set can be selected in the Solution Control Section (**LOAD** = SID) or through **RLOAD1**, **RLOAD2** or **LOAD** bulk data entries.
2. PLOADX1 is intended only for the CTRIAX6 elements.
3. The surface traction is assumed to vary linearly along the element side between GA and GB.
4. The surface traction is input as force per unit area.

6

5. THETA is measured counter-clockwise from the inward normal of the straight line between GA and GB to the vector of the applied load, as shown in the figure below. Positive pressure is in the direction inward normal to the line segment.

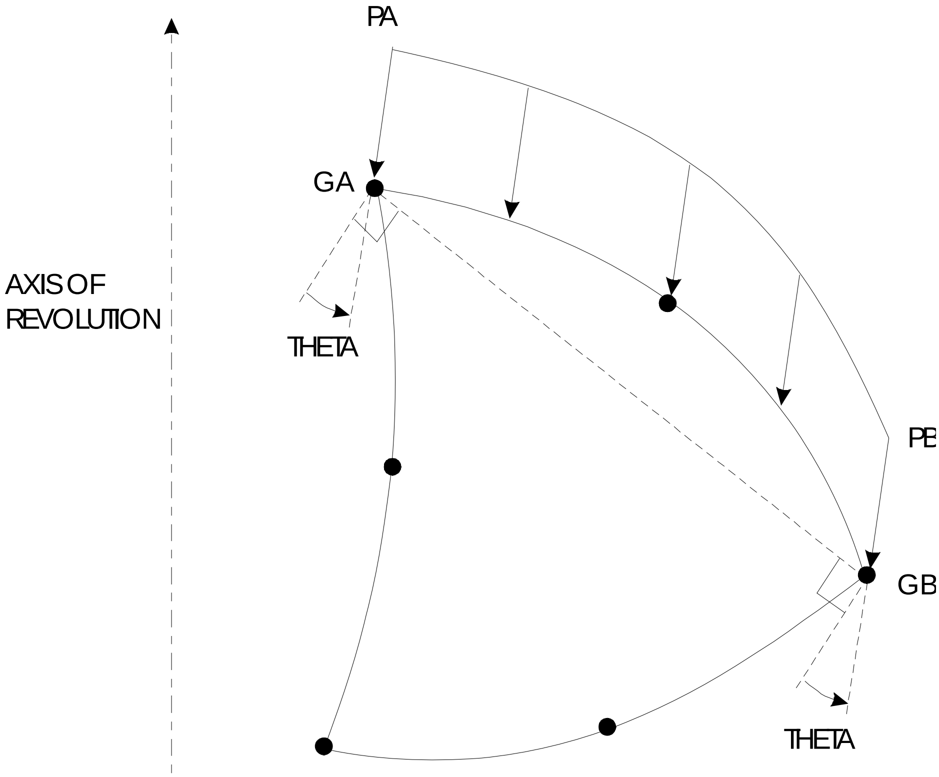


Figure 6-60

6. All elements referenced must exist.
7. Since the load is per unit of surface area, the user must be aware that, if the size of the CTRIAX6 element changes during optimization, the total load applied to it will also change.

6.7.128 PLOTEL

Data Entry: **PLOTEL** - Plot Element Definition.

Description: Defines a one-dimensional plot element.

Format:

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

Example:

1	2	3	4	5	6	7	8	9	10
PLOTEL	8	9	11						

Field	Information	Description
2	EID	Identification number of the plot element (Integer > 0).
3,4	G1,G2	**GRID** identification numbers of connection points (Integer > 0).

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.
2. Plot elements do not have stiffness, mass or any physical property. They do not affect any analysis results.

6.7.129 PMASS

Data Entry: **PMASS** - Scalar Mass Property.

Description: Defines the mass value of a scalar mass element which is defined by means of the **CMASS1** data.

Format:

1	2	3	4	5	6	7	8	9	10
PMASS	PID	M	PID	M	PID	M	PID	M	

Example:

1	2	3	4	5	6	7	8	9	10
PMASS	5	3.45	10	8.42					

Field	Information	Description
2, 4, ...	PID	Unique property identification number (Integer > 0).
3, 5, ...	M	Value of scalar mass (Real).

Remarks:

1. This data entry defines a mass value.
2. Up to four mass values may be defined by this entry.
3. Property identification numbers must be unique with respect to all other property identification numbers.

6

6.7.130 PM2UU

Data Entry: **PM2UU** - M2UU Mass Multiplier

Description: Used to define the mass multiplier of an element defined by the **M2UU1** executive control command.

Format:

1	2	3	4	5	6	7	8	9	10
PM2UU	PID	MMULT							

Example:

1	2	3	4	5	6	7	8	9	10
PM2UU	5	1.0							

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	MMULT	Mass multiplier (Real ≥ 0.0)

Remarks:

1. Property identification numbers must be unique with respect to all other property identification numbers.

6

6.7.131 PROCESS

Data Entry: **PROCESS**

Description: Run an external program that outputs bulk data statements.

Format:

```
PROCESS 'command line'
```

Examples:

```
PROCESS 'bzcat model.bulk.bz2'
```

Option	Meaning
command line	External program and arguments. The user must provide the command line according to the machine installation.

Remarks:

1. The PROCESS data can be anywhere in the bulk data.
2. Multiple PROCESS data are allowed in the bulk data.
3. The command line is limited to 240 characters.
4. If the line does not end with a quote character, additional lines will be read until the closing quote is found. Leading and trailing spaces on continued and continuation lines are discarded.

6.7.132 PROD

Data Entry: **PROD** - Rod Property.

Description: Defines the properties of a rod which is referenced by the **CROD** data.

Format:

1	2	3	4	5	6	7	8	9	10
PROD	PID	MID	A			NSM			

Example:

1	2	3	4	5	6	7	8	9	10
PROD	17	29	1.0			0.0			

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	MID	Material identification number (Integer > 0).
4	A	Area of rod (Real > 0)
7	NSM	Nonstructural mass per unit length (Real $\geq$ 0 or blank).

Remarks:

1. PROD property identification numbers must be unique with respect to **PTUBE** property identification numbers. It is recommended that PROD property identification numbers be unique with respect to all other property identification numbers.
2. PROD data may only reference **MAT1** material data. For heat transfer analysis, only **MAT4** material date can be referenced.
3. When only structural load cases are specified in the solution control, only a structural material needs to be specified. However, when only heat transfer analysis is specified, both a structural and a thermal materials must be provided.

6.7.133 PROPSET

Data Entry: **PROPSET** - Property Set.

Description: Defines a set of element properties.

Format:

1	2	3	4	5	6	7	8	9	10
PROPSET	PSID	TYPE	PID1	PID2	PID3	PID4	PID5	PID6	
+	PID7	-etc.-							

Example:

1	2	3	4	5	6	7	8	9	10
PROPSET	10	PSHELL	2	3					

Field	Information	Description
2	PSID	Property set identification number (Integer > 0).
3	TYPE	Property type for PIDi (Character) One of ‘PAXIS’, ‘PBAR’, ‘PBEAM’, ‘PBUSH’, ‘PCOMP’, ‘PCONM3’, ‘PDAMP’, ‘PELAS’, ‘PELASH’, ‘PGAP’, ‘PHBDY’, ‘PK2UU’, ‘PMASS’, ‘PM2UU’, ‘PROD’, ‘PSHEAR’, ‘PSHELL’, ‘PSOLID’, ‘PVECTOR’ or ‘PVISC’.
4 -	PIDi	Property identification number (Integer > 0)

Remarks:

1. All of the property identification numbers (PIDi) must reference properties of the same type. Properties of different types may be combined by using multiple PROPSET entries with the same PSID.
2. As many continuation lines as desired may appear.

6.7.134 PSHEAR

Data Entry: **PSHEAR** - Shear Element Property.

Description: Defines the elastic properties of a shear panel (**CSHEAR**). The alternate form provides for the internal treatment of the effective extensional area of the shear panel.

Format:

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

Alternate Format:

1	2	3	4	5	6	7	8	9	10
PSHEAR	PID	MID	T	NSM	F1	F2			

Examples:

1	2	3	4	5	6	7	8	9	10
PSHEAR	1	2	0.65	8.5					

1	2	3	4	5	6	7	8	9	10
PSHEAR	1	2	0.05	0.002	1.5	8.2			

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	MID	Material identification number (Integer > 0).
4	T	Membrane thickness (Real $\geq$ 0.0).
5	NSM	Nonstructural mass per unit area (Real $\geq$ 0 or blank. Default = 0.0).
6	F1	Effectiveness factor for extensional stiffness along sides 1-2 and 1-4 (see Remark 3) (Real $\geq$ 0.0 or Blank). Default=0.0.
7	F2	Effectiveness factor for extensional stiffness along sides 2-3 and 3-4 (see Remark 3) (Real $\geq$ 0.0 or Blank). Default=0.0.

Remarks:

1. All PSHEAR data statements must have identification numbers that are unique with respect to all other property identification numbers.
2. PSHEAR data statements may reference only **MAT1** material data.

3. The effective extensional area is treated by means of equivalent rods on the perimeter of the element. If F1 $\leq$ 1.01, the areas of the rods on edges 1-2 and 3-4 are set equal to 0.5(F1)(T)(W_1), where W_1 is the average width of the panel. Thus, if F1 = 1.0, the panel is fully effective for extension in the 1-2 direction. If F1 > 1.01, the areas of the rods on edges 1-2 and 3-4 are each set equal to 0.5(F1)T^2. Thus, if F1=24, the effective width of skin contributed by the panel to the flanges on edges 1-2 and 3-4 is equal to 12T. The significance of F2 for edges 2-3 and 1-4 is the same.
4. Poisson's ratio coupling for extension effects is ignored.
5. The two forms of the PSHEAR data may be used in the same Bulk Data Section.

6.7.135 PSHELL

Data Entry: **PSHELL** - Shell Element Property.

Description: Defines the membrane and bending properties of thin plate/shell elements (**CTRIA3**,**CQUAD4**,**CTRIA6** and **CQUAD8**).

Format:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	D or DF	MID3	TS or TSF	NSM	
+	Z1	Z2			SCSID				

Example:

1	2	3	4	5	6	7	8	9	10
PSHELL	1	1	2.0	1		1			
+	-1.0	1.0							

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	MID1	Material identification number for the membrane and/or conduction (Integer > 0 or blank). See remarks 6 and 11.
4	T	Membrane and/or conduction thickness (Real > 0.0). See remark 15.
5	MID2	Material identification number for bending (Integer > 0 or blank or -1).
6	D or DF	Bending stiffness, D (Real > 0.0 or blank. Default: D= $T^3/12$). or Bending stiffness factor, DF = 12 D/T^3 (Real > 0.0 or blank. Default: DF=1.0). See remark 7.
7	MID3	Material identification number for transverse shear. Must be blank if MID2 is blank (Integer > 0 or blank).
8	TS or TSF	Transverse shear thickness, TS (Real > 0.0 or blank. Default: TS = 0.833333T). or Transverse shear factor, TSF = TS/T (Real > 0.0 or blank. Default: TSF = 0.833333). See remark 7.
9	NSM	Nonstructural mass per unit area (Real ≥ 0 or blank. Default = 0.0).

6

2,3	Z1,Z2	Fiber distances for stress computation. The positive direction is determined by the right-hand rule and the order in which the grid points are listed on the connection data. (Real or blank, see remark 8 for defaults).
6	SCSID	Identification number of force, stress and strain output coordinate system (Integer ≥ -2 or blank. Default = -1). See remark 12.

Remarks:

1. PSHELL property identification numbers must be unique with respect to **PCOMP**, **PCOMPG** and **PSKIN** property identification numbers. It is recommended that PSHELL property identification numbers be unique with respect to all other property identification numbers.
2. The structural mass is computed from the density using the membrane thickness and membrane material properties. If the membrane material is omitted, then the bending material (MID2) will be used along with the membrane thickness.
3. The results of leaving an MID field blank (or MID2 = -1) are:
   MID1 No membrane stiffness
   MID2 No bending or transverse stiffness
   MID3 No transverse shear flexibility
4. The continuation data is not required.
5. This data is used in connection with the CTRIA3, CQUAD4, CTRIA6 and CQUAD8 data.
6. PSHELL data may reference **MAT1**, **MAT2** or **MAT8** material property data. For heat transfer analysis, the PSHELL data may reference **MAT4** or **MAT5** material property data. Also see Remark 10.
7. The meaning of the data in fields 6 and 8 depends on the compatibility format mode of the input file, as specified by the executive control command **SOL**. If the mode is COMPAT0, then field 6 is interpreted as the bending stiffness, D, and field 8 is interpreted as the transverse shear thickness, TS. If the mode is COMPAT1, then field 6 is interpreted as the bending stiffness factor, $DF = 12\ D/T^3$, and field 8 is interpreted as the transverse shear factor, TSF = TS/T.
8. The default for Z1 is -T/2, and Z2 is +T/2. T is the plate thickness, defined by T on this data.
9. For plane strain analysis, set MID2 = -1 and only MAT1 type data is allowed for MID1.
10. If the transverse shear material, MID3, references MAT2 data, then G13, G23 and G33 must be blank. If it references MAT8 data, then G1,Z and G2,Z must not be zero.
11. When only structural loadcases are requested in the solution control, heat transfer material is optional. However, if only heat transfer loadcases are requested, both structural and heat transfer material must be specified.

12. The force, stress and strain output coordinate system may be the element (-1 or blank), material (-2), basic (0) or any defined system (Integer > 0). If the output system is basic or any defined system, then the X-axis is along the projection onto the plane of the element of the X-axis of the specified coordinate system.
13. If automatic generation of stress constraints is requested, the material limits from the material specified by MID1 will be used. If MID1 is blank, the material limits from the material specified by MID2 will be used.
14. KXZ, KYZ and KZZ must be blank on MAT5 data referenced by PSHELL data.
15. If the thickness, T, is less than or equal to the value of analysis parameter **SHL2SKN** then this entry will be automatically converted into a PSKIN entry.

6.7.136 PSKIN

Data Entry: **PSKIN** - Skin Element Property.

Description: Defines the property identification number of skin elements (**CTRIA3**, **CQUAD4**, **CTRIA6** and **CQUAD8**).

Format:

1	2	3	4	5	6	7	8	9	10
PSKIN	PID								

Example:

1	2	3	4	5	6	7	8	9	10
PSKIN	100								

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).

Remarks:

1. PSKIN property identification numbers must be unique with respect to **PCOMP**, **PCOMPG** and **PSHELL** property identification numbers. It is recommended that PSKIN property identification numbers be unique with respect to all other property identification numbers.
2. Skin elements do not have stiffness, mass or any physical property. They do not affect any analysis results and can be considered as the 2-D counterpart to **PLOTEL** elements.
3. Skin element can be use by DTGRID design data to define topography optimization regions and directions of perturabations.

6

6.7.137 PSOLID

Data Entry: **PSOLID** - Properties of Solid Elements.

Defines the properties of solid elements. Referenced by **CHEXA**, **CHEX20**, **CTETRA**, **CPENTA** and **CPYRA** data.

Format:

1	2	3	4	5	6	7	8	9	10
PSOLID	PID	MID	CORDM				FCTN		

Example:

1	2	3	4	5	6	7	8	9	10
PSOLID	1	1							

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	MID	Identification number of a **MAT1**, **MAT4**, **MAT5**, **MAT9**, **MAT10** or **MAT11** data (Integer > 0).
4	CORDM	Identification number of material coordinate system (Integer $\geq$ -1 or blank. Default = 0).
8	FCTN	Fluid element option (“PFLUID” or blank). See Remark 7.

Remarks:

1. Property identification numbers must be unique with respect to all other property identification numbers.
2. Isotropic (MAT1), orthotropic (MAT11) or anisotropic (MAT9) materials may be referenced. For heat transfer analysis, either MAT4 or MAT5 data can be referenced.
3. See the CHEXA, CHEX20, CTETRA, CPENTA or CPYRA data for the definition of the element coordinate system. The material coordinate system may be the basic system (0 or blank), any defined system (Integer > 0) or the element coordinate system (-1).
4. Stress and strain components are output in the material coordinate system at the centroid of the element.
5. When only structural loadcases are requested in the solution control, heat transfer material is optional. However, if only heat transfer loadcases are requested, both structural and heat transfer material, must be specified.
6. Automatic generation of stress constraints will be skipped for elements that reference PSOLID data that references MAT9 or MAT11 materials.

7. If the FCTN field is “PFLUID” then the entry defines a fluid property, and MID must reference a **MAT10** entry. All CHEXA, CPENTA, CPYRA and CTETRA elements that reference a fluid property become fluid elements. If the FCTN field is blank then the entry defines a normal structural solid property, and MID must not reference a MAT10 entry.

6.7.138 PTUBE

Data Entry: **PTUBE** - Tube Property.

Description: Defines the properties of a tube which is referenced by the **CTUBE** data.

Format:

1	2	3	4	5	6	7	8	9	10
PTUBE	PID	MID	DIAM	THICK	NSM				

Example:

1	2	3	4	5	6	7	8	9	10
PTUBE	17	29	10.0	2.0					

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	MID	Material identification number (Integer > 0).
4	DIAM	Outer diameter (Real > 0.0)
5	THICK	Thickness of tube (0.0 < Real ≤ DIAM/2).
6	NSM	Nonstructural mass per unit length (Real ≥ 0.0 or blank).

Remarks:

1. PTUBE property identification numbers must be unique with respect to **PROD** property identification numbers. It is recommended that PTUBE property identification numbers be unique with respect to all other property identification numbers.
2. PTUBE data may only reference **MAT1** material data. For heat transfer analysis, only **MAT4** material date can be referenced.
3. When only structural load cases are specified in the solution control, only a structural material needs to be specified. However, when only heat transfer analysis is specified, both a structural and a thermal materials must be provided.
4. PTUBE data will be converted to equivalent PROD data. The sorted echo will show the generated PROD. The area of the converted rod will be calculated using the values of DIAM and THICK as follows:

$$A = \pi\frac{(DIAM^2 - (DIAM - 2xTHICK)^2)}{4}$$

6.7.139 PVECTOR

Data Entry: **PVECTOR** - Vector spring property.

Description: Used to define the stiffness and damping coefficient of a vector spring element which is referenced by **CVECTOR**.

Format:

1	2	3	4	5	6	7	8	9	10
PVECTOR	PID	K11	K12	K13	K14	K15	K16	K22	
+	K23	K24	K25	K26	K33	K34	K35	K36	
+	K44	K45	K46	K55	K56	K66	GE		

Example:

1	2	3	4	5	6	7	8	9	10
PVECTOR	3	120.		12.0				150.0	
+					130.0				
+	135.0			140.0		150.0	0.1		

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3 to 9	K11, .., K22	Elastic stiffness property values (Real or blank).
2 to 9	K23, .., K36	Elastic stiffness property values (Real or blank).
2 to 7	K44, .., K66	Elastic stiffness property values (Real or blank).
8	GE	Damping coefficient (Real or Blank).

Remarks:

1. Property identification number has to be unique with respect to all other property identification numbers.
2. The user is cautioned to be careful using negative diagonal spring values.
3. Vector elements are ignored in heat transfer loadcases.
4. The properties are in the element coordinate system. If the orientation vector is not defined in the CVECTOR data, then the element coordinate system correspond to the general coordinate system at grid A.

$$K = \begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16} \\ & K_{22} & K_{23} & K_{24} & K_{25} & K_{26} \\ & & K_{33} & K_{34} & K_{35} & K_{36} \\ & & & K_{44} & K_{45} & K_{46} \\ & \mathrm{SYM} & & & K_{55} & K_{56} \\ & & & & & K_{66} \end{bmatrix}$$

6.7.140 PVISC

Data Entry: **PVISC** - Viscous Element Property

Description: Defines the viscous properties of a one-dimensional element which is used to create viscous elements by means of the **CVISC** data.

Format:

1	2	3	4	5	6	7	8	9	10
PVISC	PID	C1	C2		PID	C1	C2		

Example:

1	2	3	4	5	6	7	8	9	10
PVISC	3	6.4	2.97						

Field	Information	Description
2, 6	PID	Unique property identification number (Integer > 0).
3, 7	C1	Viscous coefficient for extension (Real).
4, 8	C2	Viscous coefficient for rotation (Real).

Remarks:

1. Used for both extensional and rotational viscous elements.
2. Has meaning for dynamic response problems only.
3. Viscous properties are material independent; in particular, they are temperature independent.
4. One or two viscous element properties may be defined on a single entry.
5. Property identification numbers must be unique with respect to all other property identification numbers.

6.7.141 PWELD

Data Entry: **PWELD** - CWELD Element Property

Description: Used to define the properties of weld element referenced by **CWELD** data.

Format:

1	2	3	4	5	6	7	8	9	10
PWELD	PID	MID	D					TYPE	
	LDMIN	LDMAX							

Example:

1	2	3	4	5	6	7	8	9	10
PWELD	6	1	2.5						

Field	Information	Description
2	PID	Unique property identification number (Integer > 0).
3	MID	Unique material identification number (Integer > 0).
4	D	Diameter of weld(Real > 0.0).
9	TYPE	Weld Type (Character or blank. Default=blank) For Spot weld, TYPE=”SPOT”
2	LDMIN	Smallest ratio of length to diameter for stiffness calculation (Real > 0.0 or blank. Default=0.2)
3	LDMAX	Largest ratio of length to diameter for stiffness calculation (Real > 0.0 or blank. Default=5.0)

6

Remarks:

1. MID can only refer to the **MAT1** bulk data entry. Material properties, diameter D, and length are used to calculate the stiffness of the connector. The length of the connector is the distance between piercing points GA and GB (see **Figure 6-61**)
2. If TYPE=‘SPOT’ and if the formats PARTPAT, ELPAT, or ELEMID are used on the CWELD entry, then the effective length used in the stiffness calculation is $L_e = (t_A + t_B)/2$, regardless of the distance between GA and GB. t_A and t_B are the thicknesses of shell A and shell B respectively. The effective length is used to avoid excessively stiff or soft connections due to mesh irregularities.

3. If TYPE=blank, the effective length L_e is equal to the true length L, the distance between GA and GB, as long as $LDMIN \leq L/D \leq LDMAX$. If L is below the range, the effective length is set to $L_e = LDMIN \times D$ and if L is above the range, $Le = LDMIN \times D$

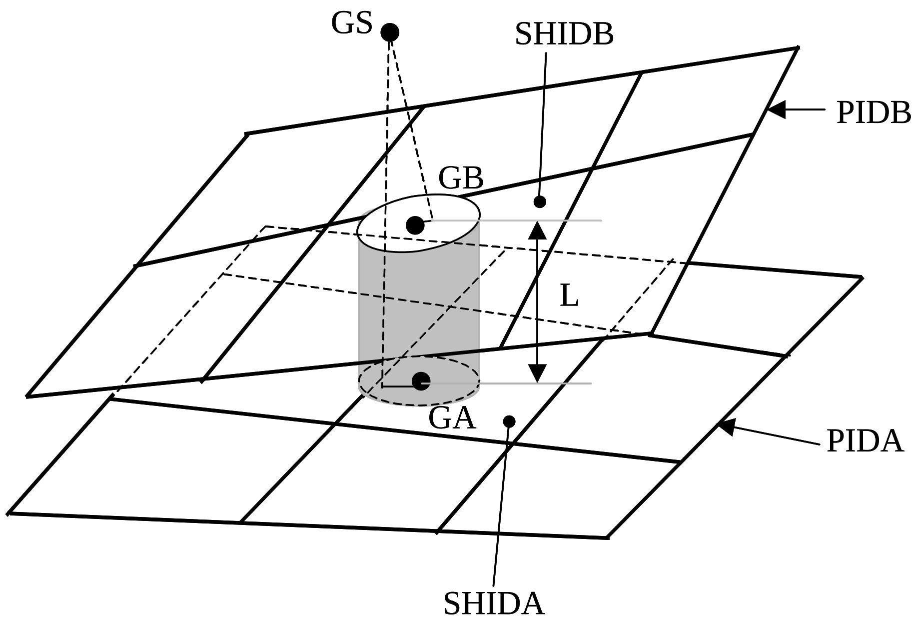


Figure 6-61 Length and Diameter of CWELD element

6

6.7.142 QBDY1

Data Entry: **QBDY1 -** Boundary Heat Flux Load for HBDY Element, Form 1.

Description: Defines a uniform heat flux into **CHBDY** elements.

Format:

1	2	3	4	5	6	7	8	9	10
QBDY1	SID	Q0	EID1	EID2	EID3	EID4	EID5	EID6	

Alternate Format:

1	2	3	4	5	6	7	8	9	10
QBDY1	SID	Q0	EID1	“THRU”	EID2				

Example:

1	2	3	4	5	6	7	8	9	10
QBDY1	2	1.0E-5	2	5	23	88			

Field	Information	Description
2	SID	Heat load set identification number (Integer > 0.
3	Q0	Heat flux into the element (Real > 0).
4-9	EIDi	CHBDY elements (Integer > 0 or Blank or “THRU”).

Remarks:

1. QBDY1 data must be selected in Solution Control (**HEAT** = SID) to be used in heat transfer. The total power into an element is given by the equation: P_{in} = (Effective area) * Q0.
2. Q0 is positive for heat input.
3. If a sequential list of elements is desired, fields 4, 5 and 6 may specify the first element, the string “THRU” and the last element. No subsequent data is allowed with this option.
4. Continuation lines are not allowed.

6

6.7.143 QBDY2

Data Entry: **QBDY2 -** Boundary Heat Flux Load for HBDY Element, Form 2.

Description: Defines a uniform heat flux into **CHBDY** elements.

Format:

1	2	3	4	5	6	7	8	9	10
QBDY2	SID	EID	Q01	Q02	Q03	Q04			

Example:

1	2	3	4	5	6	7	8	9	10
QBDY2	2	465	1.0-5	1.0-5	1.5-5	1.5-5			

Field	Information	Description
2	SID	Heat load set identification number (Integer > 0.
3	EID	Identification number of an CHBDY element (Integer > 0).
4-7	Q0i	Heat flux at the ith grid point on the referenced HBDY element (Real or Blank).

Remarks:

1. QBDY2 data must be selected in Solution Control (**HEAT** = SID) to be used in heat transfer. The total power into each point, i, on the element boundary is given by the equation:
   $P_i = AREA_i * Q0_i$.
2. $Q0_i$ is positive for heat input.
3. If the QBDY2 references a CHBDY type POINT, only Q01 is used. If it references a CHBDY type LINE or ELCYL, only Q01 and Q02 are used. If it references AREA3 or AREA4, the first 3 or 4 $Q0_i$ are used, respectively.

6

6.7.144 QHBDY

Data Entry: **QHBDY** - Boundary Heat Flux Load.

Description: Defines a uniform heat flux into a set of grid points.

Format:

1	2	3	4	5	6	7	8	9	10
QHBDY	SID	FLAG	Q0	AF	G1	G2	G3	G4	

Example:

1	2	3	4	5	6	7	8	9	10
QHBDY	12	LINE	1.0+3	.8	23	34			

Field	Information	Description
2	SID	Heat load set identification number (Integer > 0.
3	FLAG	Type of area involved. Must be one of the following: "POINT," "LINE, "AREA3" or "AREA4."
4	Q0i	Heat flux into area (Real).
5	AF	Area factor depends on type (Real > 0 or Blank).
6-9	G1, G2, G3, G4	Grid point identification of connected points (Integer > 0 or Blank).

Remarks:

1. The heat flux applied to the area is transformed to loads on the points. These points need not correspond to an CHBDY element.
2. The flux is applied to each point, i, by the equation:
   $P_i = AREA_i * Q0$
   where Q0 is positive for heat input, and $AREA_i$ is the portion of the total area associated with point i.
3. In heat transfer, the load is applied with the Solution Control request: HEAT = SID.
4. The number of connected points for the four types are 1(POINT), 2(LINE), 3(AREA3) and 4(AREA4).
5. The area factor, AF, is used to determine the effective area for the POINT and LINE types. It equals the area and the effective width, respectively. It is not used for the other types, which have their area defined implicitly and must be left blank.
6. The type flag defines a surface in the same manner as the CHBDY data. For physical descriptions of the geometry involved, see **CHBDY**.

6.7.145 QSET2

Data Entry: **QSET2 -** Generalized Degrees of Freedoms Selection

Description: Define a set of generalized degrees of freedoms used for Craig-Bampton modes in a Guyan eigenvalue loadcase.

Format:

1	2	3	4	5	6	7	8	9	10
QSET2	SID	G1	C1	G2	C2	G3	C3		
+	G4	C4	G5	C5	-etc.-				

Example:

1	2	3	4	5	6	7	8	9	10
QSET2	10	1	123456	2	123456				

Field	Information	Description
2	SID	Set identification number of QSET (Integer>0).
3, 5, ...	Gi	**GRID** or **SPOINT** identification numbers (integer > 0).
4, 6, ...	Ci	Component number of global coordinate (any unique combination of the digits 1-6 (with no embedded blanks).

Remarks:

1. Degrees of freedoms specified on this entry must not be constrained with **SPC1**, **SPC**, **MPC**, rigid elements or interpolation elements.
2. Degrees of freedoms specified on this entry must not be referenced by any element.
3. QSET sets must be selected in the Solution Control Section (**QSET** = SID) to be used.
4. The component numbers must be blank for SPOINTs.
5. There is no limit in the number of continuation lines.
6. Continuation data is optional.
7. QSET2 is an alternate format to the QSET3 data statement.
8. See **Guyan Reduction** (Sec. 2.13.1) for a general discussion.

6

6.7.146 QSET3

Data Entry: **QSET3 -** Generalized Degrees of Freedoms Selection

Description: Define a set of generalized degrees of freedoms used for Craig-Bampton modes in a Guyan eigenvalue loadcase.

Format:

1	2	3	4	5	6	7	8	9	10
QSET3	SID	C	G1	G2	G3	G4	G5	G6	
+	G7	G8	G9	G10	-etc.-				

Example:

1	2	3	4	5	6	7	8	9	10
QSET3	10	123456	1	2	3	4	5	6	
+	7	8	9	10					

Alternate Format:

1	2	3	4	5	6	7	8	9	10
QSET3	SID	C	G1	THRU	G2				

Example:

	2	3	4	5	6	7	8	9	10
QSET3	10	123456	1	THRU	10				

Field	Information	Description
2	SID	Set identification number of QSET (Integer>0).
3	C	Component number of global coordinate (any unique combination of the digits 1-6,with no embedded blanks).
4, 5, ...	Gi	**GRID** or **SPOINT** identification numbers (integer > 0).

Remarks:

1. Degrees of freedoms specified on this entry must not be constrained with SPC1, SPC, MPC, rigid elements or interpolation elements.
2. Degrees of freedoms specified on this entry must not be referenced by any element.
3. QSET sets must be selected in the Solution Control Section (**QSET** = SID) to be used.
4. The component number must be blank for SPOINTs.
5. There is no limit in the number of continuation lines.
6. Continuation data is optional.
7. QSET3 is an alternate format to the QSET2 data statement.

8. See **Guyan Reduction** (Sec. 2.13.1) for a general discussion.

6.7.147 QVECT

Data Entry: **QVECT** - Thermal Vector Flux Load.

Description: Defines thermal vector flux from a distant source into **CHBDY**, **CHBDYE**, **CHBDYG** and **CHBDYP** elements.

Format:

1	2	3	4	5	6	7	8	9	10
QVECT	SID	Q0	E1	E2	E3	EID1	EID2	EID3	
+	EID4	EID5	-etc.-						

Alternate Format 1:

1	2	3	4	5	6	7	8	9	10
QVECT	SID	Q0	E1	E2	E3	EID1	"THRU"	EID2	

Alternate Format 2:

1	2	3	4	5	6	7	8	9	10
QVECT	SID	Q0			E1	E2	E3		
+	EID1	EID2	EID3	EID4	EID5	-etc.-			

Example:

1	2	3	4	5	6	7	8	9	10
QVECT	12	1.0-2	-1.0	0.0	0.0	34	36	42	
+	65	67							

1	2	3	4	5	6	7	8	9	10
QVECT	3	5.0-1			1.0	0.0	0.0		
+	11	14	THRU	18	21				

6

Field	Information	Description
2	SID	Heat load set identification number (Integer > 0.
3	Q0	Magnitude of thermal flux vector (Real).
4-6	E1, E2, E3	Vector components (in the basic coordinate system) of the thermal vector flux (Rea or Blank). The total flux is given by Q = Q0{E1, E2, E3}
7...	EIDi	Element identification numbers of CHBDY elements irradiated by the distant source (Integer > 0 or Blank or "THRU"). See Remark 3.

Remarks:

1. For heat transfer, the load set is selected in the Solution Control data (**HEAT** = SID). The total power into an element is given by

   $$P_{in} = -\alpha A(\bar{e} \bullet \bar{n})Q0$$

   where α = Absorbtivity
   A = Area of CHBDY element
   $\bar{e}$ = Vector of real numbers, E1, E2, E3
   n = Positive normal vector of element. See CHBDY data description.
   P_{in} = 0 if the vector product is positive (i.e., the flux is coming from behind the element.
2. If the referenced CHBDY element is of TYPE = ELCYL, the power input is an exact integration over the area exposed to the thermal flux vector.
3. If a sequential list of elements is desired, fields 7, 8 and 9 may specify the first element, the string “THRU” and the last element.
4. Alternate format 2 is used with **CHBDYE**, **CHBDYG** and **CHBDYP** elements. In alternate format 2, EIDi can be an integer > 0, or a string “THRU”. If EIDi is “THRU”, it specifies a sequential list of elements.

6.7.148 QVOL

Data Entry: **QVOL** - Volume Heat Addition.

Description: Defines the rate of internal heat generation in an element.

Format:

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

Alternate Format 1:

1	2	3	4	5	6	7	8	9	10
QVOL	SID	QV	EID1	“THRU”	EID2				

Alternate Format 2:

1	2	3	4	5	6	7	8	9	10
QVOL	SID	QV		EID1	EID2	EID3	EID4	EID5	
+	EID6	-etc.-							

Example:

1	2	3	4	5	6	7	8	9	10
QVOL	12	1.0+2	132	THRU	154				

1	2	3	4	5	6	7	8	9	10
QVOL	1	2.0+1		21	23	26	THRU	31	
+	35								

6

Field	Information	Description
2	SID	Heat load set identification number (Integer > 0.
3	QV	Power input per unit volume produced by a heat conduction element (Real $\neq$ 0.0).
4-9	EIDi	A list of heat conduction elements that have volumes (Integer > 0 or “THRU” or Blank). See Remark 4.

Remarks:

1. In heat transfer analysis, the load must be selected by the solution control command (**HEAT** = SID). to be used.

2. If a sequential list of elements is desired, fields 4, 5 and 6 may specify the first element identification number, the string “THRU” and the last element identification number. No subsequent data is allowed with this option.
3. Continuation data is not allowed.
4. **CROD**, **CBAR**, **CBEAM**, **CQUAD4**, **CTRIA3**, **CTRIAX6**, **CHEXA**, **CPENTA**, **CTETRA**, **CHEX20** and **CPYRA** elements may be referenced by this data.
5. In alternate format 2, any EIDi can be a string “THRU” to specify a sequential list of elements.

6.7.149 RANDPS

Data Entry: **RANDPS** - Power Spectral Density Loads Specification.

Description: Defines the power spectral density loads for random response analysis.

$$S_{jk}(f) = (X + iY)G(f)$$

Format:

1	2	3	4	5	6	7	8	9	10
RANDPS	SID	J	K	X	Y	TID			

Example:

1	2	3	4	5	6	7	8	9	10
RANDPS	12	1	2	10.0	20.0	100			

Field	Information	Description
2	SID	Random analysis set identification number (Integer > 0.
3	J	Loadcase identification of the excited load set(Integer > 0).
4	K	Loadcase identification of the applied load set(Integer $\geq$0 ; K$\geq$J).
4	X	Real component of the complex number that define the PSD (Real)
5	Y	Imaginary component of the complex number that define the PSD.(Real or Blank, default=0.0)
TID	Y	TABRND1 set indentification number that defines G(f) (Integer$\geq$0 or Blank, default = 0). See remark 3.

6

Remarks:

1. In random response analysis, **RANDPS** has to be selected by the solution control command (**RANDOM** = SID) to be used.
2. For auto spectral density K=J, X>=0.0 and Y=0.0.
3. If TID = 0 or blank, G(f)=1.0.

6.7.150 RANDT1

Data Entry: **RANDT1** - Time Lag Specification for Autocorrelation Function Evaluation.

Description: Defines time lag constants for random response analysis autocorrelation function.

Format:

1	2	3	4	5	6	7	8	9	10
RANDT1	SID	N	T0	TMAX					

Example:

1	2	3	4	5	6	7	8	9	10
RANDT1	12	10	0.1	20.0					

Field	Information	Description
2	SID	Random analysis set identification number (Integer > 0).
3	N	Number of time lags intervals (Ingeger > 0).
4	T0	Starting time lag (Real $\geq$0.0).
4	TMAX	Maximum time lag. (Real > T0)

Remarks:

1. In random response analysis, **RANDT1** has to be selected by the solution control command **RANDOM** = SID to be used.
2. At least one **RANDPS** data entery must be used with same SID.
3. Time lags are generated using this entry

   are: $Ti = T0 + \frac{(TMAX - T0)}{N}(i-1); i = 1, N+1$

6.7.151 RBAR

Data Entry: **RBAR** - Rigid Bar.

Description: Defines a rigid bar with six degrees of freedom at each end.

Format:

1	2	3	4	5	6	7	8	9	10
RBAR	EID	GA	GB	CNA	CNB	CMA	CMB		

Example:

1	2	3	4	5	6	7	8	9	10
RBAR	5	100	101	1236	34				

Field	Information	Description
2	EID	Identification number of the rigid element (integer > 0).
3,4	GA,GB	**GRID** identification number of connection points (Integer > 0, $GA \neq GB$)
5,6	CNA,CNB	Independent degrees of freedom in the general coordinate system for the element at grid points GA and GB, indicated by any combination of the digits 1 - 6 with no embedded blanks (Integer $\geq$ 0 or blank). See Remark 1.
7,8	CMA,CMB	Component numbers of dependent degrees of freedom in the general coordinate system assigned by the element at grid points GA and GB, indicated by any combination of the digits 1 - 6 with no embedded blanks (Integer $\geq$ 0 or blank). See Remarks 2 and 3.

6

Remarks:

1. The total components in CNA and CNB must equal six; for example, CNA = 1236, CNB = 34. Furthermore, they must jointly be capable of representing any general rigid body motion of the element.
2. If both CMA and CMB are zero or blank, all of the degrees of freedom not in CNA and CNB will be made dependent.
3. Degrees of freedom specified as dependent may not be listed as dependent on other rigid or interpolation elements or **MPC**. Also, dependent degrees of freedom may not be listed on **SPC**, **SPC1**, **ASET2**, **ASET3** or **SUPORT1** data entries.
4. Element identification numbers must be unique with respect to all other element identification numbers.
5. Rigid elements, unlike MPCs are not selected through the Solution Control Section.
6. Forces of constraints are not recovered. **FORCE** will produce no output for this element.

7. Rigid bar elements are ignored in heat transfer analysis.

6.7.152 RBE1

Data Entry: **RBE1** - Rigid Body Element.

Description: Defines a rigid body whose independent degrees of freedom are specified at several grid points and whose dependent degrees of freedom are specified at an arbitrary number of grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RBE1	EID	GN1	CN1	GN2	CN2	GN3	CN3		
+		GN4	CN4	GN5	CN5	GN6	CN6		
+	“UM”	GM1	CM1	GM2	CM2	GM3	CM3		
+		GM4	CM4	-etc.-					

Example:

1	2	3	4	5	6	7	8	9	10
RBE1	100	10	123	20	456				
+	UM	30	246						

Field	Information	Description
2	EID	Identification number of the rigid element (Integer > 0).
3, 5, 7	GNi	**GRID** to which independent degrees of freedom for the element are assigned (Integer > 0).
4, 6, 8	CNi	Independent degrees of freedom in the global coordinate system for the rigid element at grid point(s) GNi. See Remark 1. (Integers 1 through 6 with no embedded blanks).
2	“UM”	Indicates the start of the list of dependent degrees of freedom. (Character).
3, 5, 7	GMi	GRID at which dependent degrees of freedom are assigned (Integer > 0).
4, 6, 8	CMi	Component number of the dependent degrees of freedom in the global coordinate system at grid point GM1, GM2, etc. The components are indicated by any of the digits 1-6 with no embedded blanks (Integer > 0).

Remarks:

1. The total number of components in CN1 to CN6 must equal six. For example, CN1=123, CN2=3, CN3=2, CN4=3. Furthermore, they must jointly be capable of representing any general rigid body motion of the element.
2. The first continuation entry is not required if there are fewer than four GN points.

6

3. Element identification numbers must be unique with respect to all other element identification numbers.
4. Forces of constraint are not recovered. **FORCE** will produce no output for this element.
5. Degrees of freedom specified as dependent may not be listed as dependent on other rigid or interpolation elements or **MPC**. Also, dependent degrees of freedom may not be listed on **SPC**, **SPC1**, **ASET2**, **ASET3** or **SUPORT1** data entries.
6. Rigid body elements are ignored in heat transfer analysis.

6.7.153 RBE2

Data Entry: **RBE2** - Rigid Body Element.

Description: Defines a rigid body whose independent degrees of freedom are specified at a single grid point and whose dependent degrees of freedom are specified at an arbitrary number of grid points.

Format:

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4	GM5	
+	GM6	GM7	GM8	-etc.-					

Example:

1	2	3	4	5	6	7	8	9	10
RBE2	7	6	135	3	9				
+									

Field	Information	Description
2	EID	Identification number of the rigid element (Integer > 0).
3	GN	**GRID** to which all six independent degrees of freedom for the element are assigned (Integer > 0).
4	CM	Component number of the dependent degrees of freedom in the global coordinate system at grid point GM1, GM2, etc. The components are indicated by any of the digits 1-6 with no embedded blanks (Integer > 0).
5,...	GMi	GRID at which dependent degrees of freedom are assigned (Integer > 0, $GM_i \neq GN$).

Remarks:

1. The components indicated by CM are made dependent at all grid points, GMi.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. Rigid elements, unlike MPCs are not selected through the Solution Control Section.
4. Forces of constraint are not recovered. **FORCE** will produce no output for this element.
5. Degrees of freedom specified as dependent may not be listed as dependent on other rigid or interpolation elements or **MPC**. Also, dependent degrees of freedom may not be listed on **SPC**, **SPC1**, **ASET2**, **ASET3** or **SUPORT1** data entries.
6. Rigid body elements are ignored in heat transfer analysis.

6.7.154 RBE3

Data Entry: **RBE3** - Interpolation Constraint Element.

Description: Defines a motion at a “reference” grid point as the weighted average of the motions at a set of other grid points. This element is useful to distribute loads and masses without adding stiffness to the model.

Format:

1	2	3	4	5	6	7	8	9	10
RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
+	G1,3	-etc.-	WT2	C2	G2,1	G2,2	-etc.-	WT3	
+	C3	G3,1	-etc.-.		WT4	C4	G4,1	G4,2	
+	-etc.-								
+	“UM”	GM1	CM1	GM2	CM2	GM3	CM3		
+		GM4	CM4	GM5	CM5	GM6	CM6		

Example:

1	2	3	4	5	6	7	8	9	10
RBE3	16		100	123456	1.0	123	3	9	
+	5	5.3	123	2	4	6	5.2	2	
+	7	8		6.0	1	15	16		
+	UM	100	1456	5	3	7	2		

Field	Information	Description
2	EID	Identification number of the interpolation element (Integer > 0).
4	REFGRID	Reference **GRID** (Integer > 0).
5	REFC	Global components of motion whose values will be computed at the reference grid point. Any of the digits 1, 2, ... 6 with no embedded blanks (Integer > 0).
6	WTi	Weighting factor for components of motion on the following entry at grid points Gi,j (Real).
7	Ci	Global components of motion which have weighting factor WTi at grid points Gi,j. Any of the digits 1, 2, ... 6 with no embedded blanks (Integer > 0).
8, ..	Gi,j	GRID whose components Ci have weighting factor WTi in the averaging equations (Integer > 0).

6

2	"UM"	(Optional) Character string which indicates the start of the data defining the dependent degrees of freedom. The default action is to make only the REFC components of grid REFGRID dependent.
3, 5, 7, ..	GMi	GRID with dependent components. (Integer > 0).
4, 6, 8, ...	CMi	Dependent components of motion at GMi. Any of the digits 1, 2, ..., 6 with no imbedded blanks (Integer > 0).

Remarks:

1. It is recommended that only the translation components 123 be used for Ci. An exception is the case where the Gi,j are colinear. A rotation component should then be added to one grid point to stabilize rigid body rotation about the axis of the element. If rotation components are included, then the weight factors corresponding to the rotation components are scaled by the square of the average of the distances from Gi,j to REFGRID.
2. Blank spaces may be left at the end of a Gi,j sequence.
3. The default for "UM" should be used except in cases where the user wishes to include some or all REFC components on entries where dependent degrees of freedom are not allowed. If the default is not used for "UM":
    a. The total number of dependent degrees of freedom defined by the element must be equal to the number of components in REFC (six components in the above example).
    b. The components specified after "UM" must be a subset of the components specified under {REFGRID,REFC} and {Gi,j, Ci}.
4. Interpolation elements, unlike MPCs, are not selected through the Solution Control Section.
5. RBE3 elements are ignored in heat transfer problems.
6. Degrees of freedom specified as dependent may not be listed as dependent on other rigid or interpolation elements or **MPC**. Also, dependent degrees of freedom may not be listed on **SPC**, **SPC1**, **ASET2**, **ASET3** or **SUPORT1** data entries.
7. Element identification numbers must be unique with respect to all other element identification numbers.
8. Forces of constraints are not recovered. **FORCE** will produce no output for this element.
9. See **RBE3 Element** (Sec. 2.8.1) for a general discussion.
10. If the analysis parameter **RBE3SPC** is set to NO (default), then grids in the Gi,j sequences that are not attached to any non-RBE3 elements will generate a fatal error. Leaving such degrees of freedom can cause spurious modes or other failures in eigenvalue solution. RBE3SPC can be set to YES to ignore the error, or to FIX to automatically remove such degrees of freedom from the element.

6.7.155 RFORCE

Data Entry: **RFORCE** - Rotational force.

Description: Defines a static loading condition due to a centrifugal force field.

Format:

1	2	3	4	5	6	7	8	9	10
RFORCE	SID	GID	CID	A	N1	N2	N3		
+	ACC								

Example:

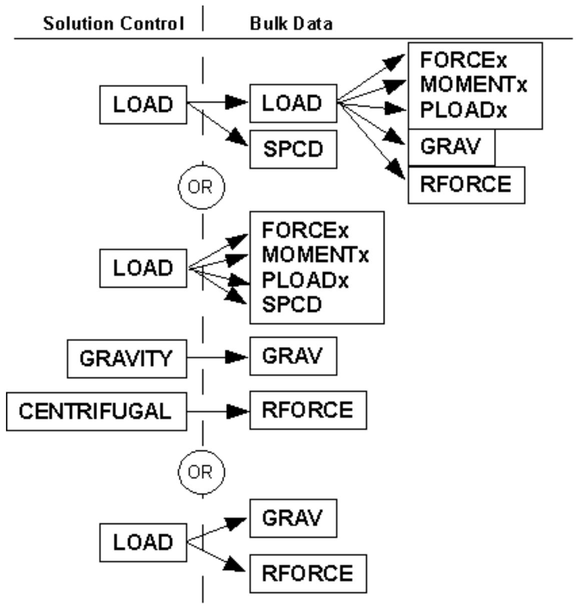
1	2	3	4	5	6	7	8	9	10
RFORCE	2	7	2	-5.3	0.0	0.0	1.0		
+	1.0								

Field	Information	Description
2	SID	Load set identification number (Integer > 0.
3	GID	**GRID** identification number at which the rotation vector acts (Integer $\geq$ 0 or blank. Default = 0).
4	CID	Cartesian coordinate system defining the components of the rotation vector (Integer $\geq$ 0. Default = 0).
5	A	Scale factor of the rotational velocity in revolutions per unit time (Real for blank. Default = 0.0).
6,7,8	N1, N2, N3	Components of the rotation direction vector at grid point GID defined in the CID coordinate system (Real. $N1^2 + N2^2 + N3^2 > 0.0$).
1	ACC	Scale factor of the rotational acceleration in revolutions per unit time squared (Real or blank. Default = 0.0). This line of data is optional.

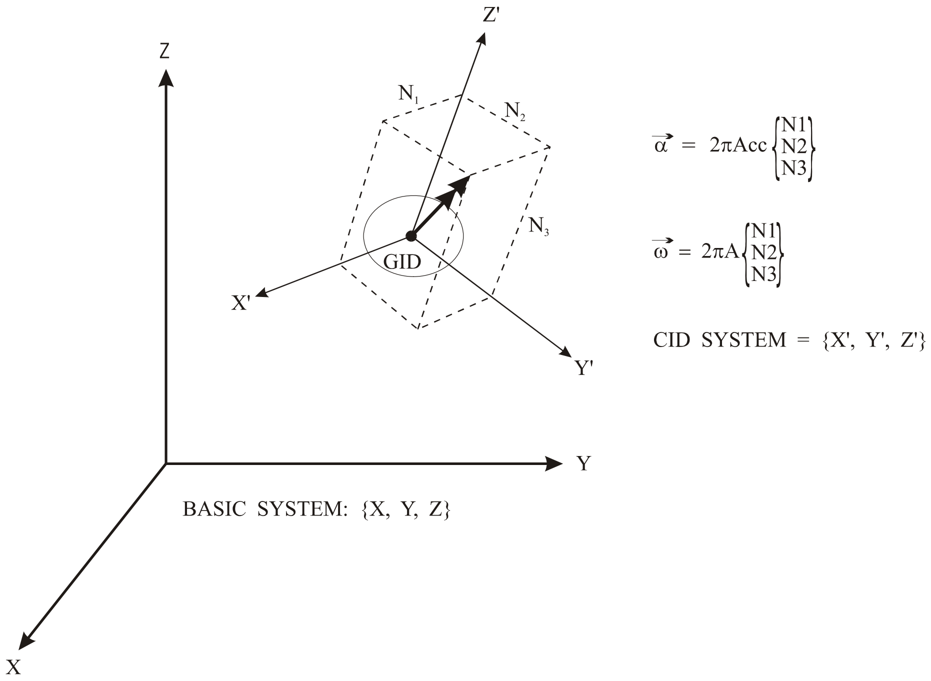
Remarks:

1. GID = 0 or blank means the rotation vector is defined at the origin of the basic coordinate system.
2. CID = 0 or blank signifies that the rotation vector is defined in the basic coordinate system.
3. There can be only one RFORCE entry for each rotational load set identification number.
4. Rotational load set identification numbers must be unique with respect to load sets defined by **FORCE**, **FORCE1**, **MOMENT**, **MOMENT1**, **PLOAD1**, **PLOAD2**, **PLOAD4**, **PLOAD5**, **PLOADA** and **PLOADX1**.

5. Either the **CENTRIFUGAL** solution control command or the **LOAD** bulk data entry can be used to combine centrifugal loading with concentrated and pressure loads sets. Only one RFORCE per LOADCASE is allowed.
6. Rotational load sets can be selected in static load cases with either the Solution Control command **CENTRIFUGAL**=SID or with the Solution Control command **LOAD**=SID.



7. The load vector generated by this entry can be printed with an **OLOAD** request in the Solution Control Section.
8. Centrifugal loads are internally created for all elements with non-zero mass. The only exception is for the **CMASS1** elements that are connected through scalar points, in which case the centrifugal load cannot be defined. When CMASS1 elements are connected with grid points, the centrifugal load is calculated and projected in the direction of the degree of freedom component.


Z
Z'
N1
N2
N3
GID
X'
Y'
Y
X
BASIC SYSTEM: {X, Y, Z}
$\vec{\alpha} = 2\pi Acc \begin{Bmatrix} N1 \\ N2 \\ N3 \end{Bmatrix}$
$\vec{\omega} = 2\pi A \begin{Bmatrix} N1 \\ N2 \\ N3 \end{Bmatrix}$
CID SYSTEM = {X', Y', Z'}

6.7.156 RLOAD1

Data Entry: **RLOAD1** - Frequency Response Dynamic Load, Form 1

Description: Defines frequency dependent dynamic load of the form $P(f) = A[C(f) + iD(f)]e^{i(\theta - 2\pi f\tau)}$ or enforced displacement, velocity or acceleration of the form $u(f)|\dot{u}(f)|\ddot{u}(f) = A[C(f) + iD(f)]e^{i(\theta - 2\pi f\tau)}$ for use in frequency response calculation.

Format:

1	2	3	4	5	6	7	8	9	10
RLOAD1	SID	A	DELAY	DPHASE	TC	TD	TYPE		

Alternate Format 1:

1	2	3	4	5	6	7	8	9	10
RLOAD1	SID				TC	TD	LSID		

Alternate Format 2:

1	2	3	4	5	6	7	8	9	10
RLOAD1	SID				TC	TD		GSID	

Examples:

1	2	3	4	5	6	7	8	9	10
RLOAD1	5	10	8	21	1	3			

1	2	3	4	5	6	7	8	9	10
RLOAD1	6		1.2	0.6	1	4		100	

6

Field	Information	Description
2	SID	Load set identification number (Integer > 0).
3	A	Set identification number of **DAREA** data or of static FORCEi, LOAD, MOMENTi, and/or PLOADi data used to define A or of SPCD data to define the amplitude of enforced displacement, velocity or acceleration. (Integer > 0). See remark 4.
4	DELAY	Set identification number of **DELAY** data which defines τ or the value of τ (Integer $\geq$ 0 or Real or blank). See remark 5.
5	DPHASE	Set identification number of **DPHASE** data which defines θ or the value of θ (Integer $\geq$ 0 or Real or blank). See remark 5.

6	TC	Identification number of a **TABLED1**, **TABLED2**, **TABLED3** or **TABLED4** entry which gives C(*f*) (Integer > 0 or blank; TC+TD > 0).
7	TD	Identification number of TABLEDi data which gives D(*f*) (Integer > 0 or blank; TC+TD > 0).
8	TYPE	Type of dynamic loading (LOAD, DISP, VELO, ACCE or blank). Default is LOAD. See remark 4.
8	LSID	Set identification number of FORCEi, LOAD, MOMENTi, and/or PLOADi data used to define A (Integer > 0).
9	GSID	Set identification number for GRAV data used to define A (Integer > 0).

Remarks:

1. If any DELAY, DPHASE, TC or TD are blank or zero, the corresponding τ, θ, C(*f*) or D(*f*) will be zero.
2. Dynamic load sets are not used unless selected by the Solution Control command **DLOAD** or referenced by selected **DLOAD** bulk data entries.
3. RLOAD1 data can be combined with other RLOAD1, **RLOAD2** or **RLOAD3** data by specifying the same SID.
4. If TYPE (field 8) is LOAD or blank, then A (field 3) can be either a DAREA ID or a static load set ID. If DAREA data with the referenced ID exists, then only that data will be used. Otherwise, static load data defined by **FORCE**, **FORCE1**, **MOMENT**, **MOMENT1**, **PLOAD1**, **PLOAD2**, **PLOAD4**, **PLOAD5**, **PLOADA**, **PLOADX1** or **LOAD** will be used. If TYPE is DISP, VELO or ACCE, then A must be the set ID of **SPCD** data.
5. If A (field 3) specifies DAREA data, then fields 4 and 5 specify DELAY and DPHASE data sets respectively and must integer or blank. If A specifies static load set data or SPCD data, then fields 4 and 5 must be real or blank.

6.7.157 RLOAD2

Data Entry: **RLOAD2 -** Frequency Response Dynamic Load, Form 2

Description: Defines frequency dependent dynamic load of the form $P(f) = AB(f)e^{i\{\phi(f) + \theta - 2\pi f\tau\}}$ or enforced displacement, velocity or acceleration of the form $u(f)|\dot{u}(f)|\ddot{u}(f) = AB(f)e^{i\{\phi(f) + \theta - 2\pi f\tau\}}$ for use in frequency response calculation.

Format:

1	2	3	4	5	6	7	8	9	10
RLOAD2	SID	A	DELAY	DPHASE	TB	TP	TYPE		

Alternate Format 1:

1	2	3	4	5	6	7	8	9	10
RLOAD2	SID				TB	TP	LSID		

Alternate Format 2:

1	2	3	4	5	6	7	8	9	10
RLOAD2	SID				TB	TP		GSID	

Examples:

1	2	3	4	5	6	7	8	9	10
RLOAD2	5	3			7				

1	2	3	4	5	6	7	8	9	10
RLOAD2	6		1.2	0.6	7	4		100	

6

Field	Information	Description
2	SID	Load set identification number (Integer > 0).
3	A	Set identification number of **DAREA** data or of static FORCEi, LOAD, MOMENTi, and/or PLOADi data used to define A or of SPCD data to define the amplitude of enforced displacement, velocity or acceleration. (Integer > 0). See remark 4.
4	DELAY	Set identification number of **DELAY** data which defines τ or the value of τ (Integer $\geq$ 0 or Real or blank). See remark 5.
5	DPHASE	Set identification number of **DPHASE** data which defines θ or the value of θ (Integer $\geq$ 0 or Real or blank). See remark 5.

6	TB	Identification number of a **TABLED1**, **TABLED2**, **TABLED3** or **TABLED4** entry which gives B(*f*) (Integer > 0).
7	TP	Identification number of TABLEDi data which gives $\phi(f)$ (Integer > 0 or blank).
8	TYPE	Type of dynamic loading (LOAD, DISP, VELO, ACCE or blank). Default is LOAD. See remark 4.
8	LSID	Set identification number of FORCEi, LOAD, MOMENTi, and/or PLOADi data used to define A (Integer > 0 or blank).
9	GSID	Set identification number for GRAV data used to define A (Integer > 0 or blank).

Remarks:

1. If any of DELAY, DPHASE or TP are blank or zero, the corresponding τ, θ or $\phi(f)$ will be zero.
2. Dynamic load sets are not used unless selected by the Solution Control command **DLOAD** or referenced by selected **DLOAD** bulk data entries.
3. RLOAD2 data can be combined with other **RLOAD1**, RLOAD2 or **RLOAD3** data by specifying the same SID.
4. If TYPE (field 8) is LOAD or blank, then A (field 3) can be either a DAREA ID or a static load set ID. If DAREA data with the referenced ID exists, then only that data will be used. Otherwise, static load data defined by **FORCE**, **FORCE1**, **MOMENT**, **MOMENT1**, **PLOAD1**, **PLOAD2**, **PLOAD4**, **PLOAD5**, **PLOADA**, **PLOADX1** or **LOAD** will be used. If TYPE is DISP, VELO or ACCE, then A must be the set ID of **SPCD** data.
5. If A (field 3) specifies DAREA data, then fields 4 and 5 specify DELAY and DPHASE data sets respectively and must integer or blank. If A specifies static load set data or SPCD data, then fields 4 and 5 must be real or blank.

6.7.158 RLOAD3

Data Entry: **RLOAD3 -** Frequency Response Dynamic Load, Form 3

Description: Defines frequency dependent dynamic load of the form $P(f) = A$ for use in frequency response calculations.

Format:

1	2	3	4	5	6	7	8	9	10
RLOAD3	SID	GID	C	A					

Example:

1	2	3	4	5	6	7	8	9	10
RLOAD3	100	6	2	7.0					

Field	Information	Description
2	SID	Load set identification number (Integer > 0).
3	GID	**GRID** or **SPOINT** number (Integer > 0).
4	C	Component number (1-6 for grid point, 0 or blank for scalar point).
5	A	Amplitude (Area) of load (Real).

Remarks:

1. Dynamic load sets must be selected by the Solution Control data **DLOAD**.
2. RLOAD3 data can be combined with other **RLOAD1**, **RLOAD2** or RLOAD3 data by specifying the same SID.

6

6.7.159 RROD

Data Entry: **RROD** - Rigid Pin-Ended Rod.

Description: Defines a pin-ended rod that is rigid in extension and compression.

Format:

1	2	3	4	5	6	7	8	9	10
RROD	EID	GA	GB	CMA	CMB				

Example:

1	2	3	4	5	6	7	8	9	10
RROD	8	9	11		2				

Field	Information	Description
2	EID	Identification number of the rigid element (Integer > 0).
3,4	GA,GB	**GRID** identification numbers of connection points (Integer > 0, $GA \neq GB$).
5,6	CMA,CMB	Component number of one and only one dependent translational degree of freedom in the general coordinate system assigned by the user to either GA or GB. (Integer equal to 1, 2, or 3). Either CMA or CMB must contain the integer and the other must be blank.

Remarks:

1. Degrees of freedom specified as dependent may not be listed as dependent on other rigid or interpolation elements or **MPC**. Also, dependent degrees of freedom may not be listed on **SPC**, **SPC1**, **ASET2**, **ASET3** or **SUPORT1** data entries.
2. Element identification numbers must be unique with respect to all other element identification numbers.
3. Rigid elements, unlike MPCs, are not selected through the Solution Control Section.
4. Forces of constraint are not recovered. **FORCE** will produce no output for this element.
5. The degree of freedom selected to be dependent must have a nonzero component along the axis of the rod. This implies that the rod must have finite length.
6. Rigid rod elements are ignored in heat transfer analysis.

6

6.7.160 RSPLINE

Data Entry: **RSPLINE** - Interpolation Constraint Element.

Description: Defines multipoint constraints for the interpolation of displacements at grid points. This element is useful to connect shell meshes with different element density.

Format:

1	2	3	4	5	6	7	8	9	10
RSPLINE	EID	D/L	G1	G2	C2	G3	C3	G4	
+	C4	G5	C5	-etc.-					

Example:

1	2	3	4	5	6	7	8	9	10
RSPLINE	100	0.2	40	50	123456	60		70	
+	123	80	123	90					

Field	Information	Description
2	EID	Unique identification number of the interpolation element with respect to all other elements (Integer > 0).
3	D/L	Ratio of the diameter of the elastic tube which the spline represents to the sum of the lengths of all segments. Default = 0.1 (Real > 0).
4, 5, 7, ...	Gi	**GRID** identification number of the ith grid point (Integer > 0).
6, 8, ...	Ci	Component numbers to be constrained in the ith grid point. Blank or any combination of the integers 1 through 6. See Remark 2.

Remarks:

1. Displacements are interpolated from the equations of an elastic beam passing through the grid points.
2. A blank entry in Ci indicates that all six degrees of freedom at Gi are independent. Since G1 must be independent, no field is provided for C1. Since the last grid point must also be independent, the last entry must be a Gi, not a Ci. For the example shown above, G1, G3 and G6 are independent. G2 has six constrained degrees of freedom, while G4 and G5 each have three constrained degrees of freedom.
3. Degrees of freedom specified as dependent may not be listed as dependent on other rigid or interpolation elements or **MPC**. Also, dependent degrees of freedom may not be listed on **SPC**, **SPC1**, **ASET2**, **ASET3** or **SUPORT1** data entries.
4. Element identification numbers must be unique with respect to all other element identification numbers.

5. Interpolation elements, unlike MPCs, are not selected through the Solution Control Section.
6. RSPLINE elements are ignored in heat transfer analysis.
7. Forces of constraints are not recovered. **FORCE** will produce no output for this element.
8. See **RSPLINE element** (Sec. 2.8.2) for a general discussion.

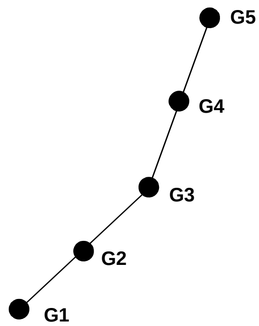


Figure 6-62 RSPLINE

6.7.161 SPC

Data Entry: **SPC** - Single-Point Constraint.

Description: Defines sets of single-point constraints.

Format:

1	2	3	4	5	6	7	8	9	10
SPC	SID	G	C	D	G	C	D		

Example:

1	2	3	4	5	6	7	8	9	10
SPC	1	2	246	0.0					

Field	Information	Description
2	SID	Identification number of single-point constraint set (Integer > 0).
3,6	G	**GRID** or **SPOINT** identification numbers (Integer > 0).
4,7	C	Component number of global coordinate (any unique combination of the digits 1-6 with no embedded blanks). Blank for heat transfer analysis and scalar points.
5,8	D	Constraint for degrees of freedom specified by G and C (Real or blank. Default = 0.0).

Remarks:

1. Continuation data is not allowed.
2. Degrees of freedom listed on this data may not be listed on **ASET2**, **ASET3** or **SUPORT1** entries. Also, they may not be specified as dependent on **RROD**, **RBAR**, **RBE1**, **RBE2**, **RBE3**, **RSPLINE**, **BOLT** and **MPC** data entries.
3. Single-point constraint sets must be selected in the Solution Control Section (**SPC**=SID) or **SPCADD** data.
4. SPC degrees of freedom may be redundantly specified as permanent constraints on the GRID data only if D is zero.
5. The component number must be blank for SPC sets referenced by heat transfer loadcases and for SPOINTs.
6. The value of D is overridden by any value specified by an **SPCD** entry that is selected for the same loadcase.
7. Non-zero values of D can only be used in static or heat transfer loadcases.

6

6.7.162 SPC1

Data Entry: **SPC1** - Single-Point Constraint.

Description: Defines sets of single-point constraints.

Format:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
+	G7	G8	G9	-etc.-					

Example:

1	2	3	4	5	6	7	8	9	10
SPC1	1	2456	2	4	6	8	10	12	
+	14								

Alternate Format:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	GID1	“THRU”	GID2				

Example:

1	2	3	4	5	6	7	8	9	10
SPC1	2	2456	2	THRU	200				

Field	Information	Description
2	SID	Identification number of single-point constraint set (Integer > 0).
3	C	Component number of global coordinate (any unique combination of the digits 1-6 with no embedded blanks). Blank for heat transfer analysis and scalar points.
4,5..	Gi, GIDi	**GRID** or **SPOINT** identification numbers (Integer > 0).

Remarks:

1. Note that enforced non-zero displacements are not available via this data.
2. As many continuation data as desired may appear when “THRU” is not used.
3. Degrees of freedom listed on this data may not be listed on **ASET2**, **ASET3** or **SUPORT1** entries. Also, they may not be specified as dependent on **RROD**, **RBAR**, **RBE1**, **RBE2**, **RBE3**, **RSPLINE**, **BOLT** and **MPC** data entries.
4. Single-point constraint sets must be selected in the Solution Control Section (**SPC**=SID) or **SPCADD** data.

5. SPC degrees of freedom may be redundantly specified as permanent constraints on the GRID data.
6. If the alternate form is used, points in the sequence GID1 through GID2 are not required to exist. Points which do not exist will collectively produce a warning message but will otherwise be ignored.
7. Continuation data entries are optional.
8. The component number must be blank for SPC sets referenced by heat transfer loadcases and for SPOINTs.

6.7.163 SPCADD

Data Entry: **SPCADD** - Single point constraint set combination.

Description: Defines a new SPC set as a union of SPC sets defined on SPC and SPC1 entries.

Format:

1	2	3	4	5	6	7	8	9	10
SPCADD	SID	SID1	SID2	SID3	SID4	SID5	SID6	SID7	
+	SID8	SID9	-etc.-						

Example:

1	2	3	4	5	6	7	8	9	10
SPCADD	10	1	3	7					

Field	Information	Description
2	SID	Unique single-point constraint set Identification number (Integer > 0).
3, 4, 5, ...	SIDi	Single-point constraint set ID used by **SPC** or **SPC1** entries (Integer > 0).

Remarks:

1. Single-point constraint sets can be selected in the solution control section (**SPC** = SID).
2. The single-point constraint IDs (SIDi) must be unique.
3. SPC sets defined by other SPCADD entries may not be referenced.
4. The SPC set ID defined by SPCADD must not also be used by SPC or SPC1 bulk data entries.
5. SPCADD may not combine SPC sets that specify different non-zero enforced displacements for the same degree of freedom.

6

6.7.164 SPCD

Data Entry: **SPCD** - Enforced Displacement or Temperature Value.

Description: Defines an enforced displacement value for static analysis, which is requested as a **LOAD** or an enforced temperature for heat transfer analysis, as requested by **HEAT**.

Format:

1	2	3	4	5	6	7	8	9	10
SPCD	SID	G1	C1	D1	G2	C2	D2		
+		G3	C3	D3	-etc.-				

Example:

1	2	3	4	5	6	7	8	9	10
SPCD	2	100	4	6.0					

Field	Information	Description
2	SID	Identification number of a static load set (Integer > 0).
3,6	Gi	**GRID** or **SPOINT** identification number (Integer > 0).
4,7	Ci	Component number of global coordinate (any unique combination of the digits 1-6 with no embedded blanks) or blank for prescribed temperatures and scalar points.
5,8	Di	Value of enforced displacement for all coordinates designed by Gi and Ci ($\mathrm{Real} \neq 0.0$).

Remarks:

1. The degree of freedom that is enforced with this data must be constrained using a **SPC1** or **SPC** data statement. This SPC1 or SPC data must be requested by an **SPC** control in the same load case that requests a SPCD.
2. Values of Di will override the SPC1 or SPC data entry if the LOAD set is requested.
3. Continuation data entries are optional.
4. The component number must be blank for heat transfer prescribed temperatures and for SPOINTs.
5. The **LOAD** bulk data entry cannot be used to reference SPCD data.

6.7.165 SPOINT

Data Entry: **SPOINT** - Scalar Point List

Description: Defines scalar points of the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
SPOINT	ID	ID	ID	ID	ID	ID	ID	ID	

Example:

1	2	3	4	5	6	7	8	9	10
SPOINT	3	18	1	4	16	2			

Alternate Format:

1	2	3	4	5	6	7	8	9	10
SPOINT	ID1	"THRU"	ID2						

Example:

1	2	3	4	5	6	7	8	9	10
SPOINT	5	THRU	649						

Field	Information	Description
2, 3, ...	ID	SPOINT identification number (Integer > 0).
2, 4	ID1, ID2	SPOINT identification number (Integer > 0; ID1 < ID2).

Remarks:

1. All scalar point identification numbers must be unique with respect to all other grid and scalar points.
2. If the alternate form is used, all scalar points ID1 through ID2 are defined.
3. Multiple SPOINT lists are allowed.

6

6.7.166 SUPORT1

Data Entry: **SUPORT1 -** Reference degrees of freedoms selection for inertia relief analysis.

Description: Define a set of reference degrees of freedoms for a free body for inertia relief analysis.

Format:

1	2	3	4	5	6	7	8	9	10
SUPORT1	SID	G1	C1	G2	C2	G3	C3		

Example:

1	2	3	4	5	6	7	8	9	10
SUPORT1	10	1	123456						

Field	Information	Description
2	SID	Set identification number of SUPORT (Integer>0).
3,5,7	Gi	Grid point identification numbers (integer > 0).
4,6,8	Ci	Component number of Global Coordinate, any unique combination of the digits 1-6 (with no embedded blanks).

Remarks:

1. Degrees of freedoms specified on this data must not be constrained with **SPC1**, **SPC**, **MPC**, rigid body elements or interpolation elements.
2. SUPORT1 sets are activated by the **SUPORT** Solution Control command.
3. Continuation lines are not allowed.
4. See **Inertia Relief** (Sec. 2.12.1) for a general discussion.

6

6.7.167 SWLDPRM

Data Entry: **SWLDPRM** - Override **CWELD** connectivity search parameters

Description: Overrides default parameter values used in search and projection calculations for CWELD connectivity.

Format:

1	2	3	4	5	6	7	8	9	10
SWLDPRM	PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	PARAM4	VAL4	
+	PARAM5	VAL5	PARAM6	VAL6	PARAM7	VAL7			

Example:

1	2	3	4	5	6	7	8	9	10
SWLDPRM	PRTSW	1	NREDIA	2	GSMOVE	2	GMCHK	1	

Field	Information	Description
2,4,6,..	PARAMi	Character string identifying parameter to change (Default=blank; see Remark 4
3,5,7,..	VALi	Parameter value (Real or integer; see Remark 4).

Remarks:

1. Only one SWLDPRM entry is allowed in the Bulk Data section.
2. SWLDPRM changes the default settings of parameters used for generating CWELD connectivity. Connectivity is generated for PARTPAT, ELPAT, ELEMID and GRIDID formats. None of the parameters on this entry are required. The default settings should be changed only for diagnostic and debug purposes.
3. If an error is encountered during connectivity generation, the element is rejected and the progam will loop to the next element until all CWELD elements have been processed.
4. The list of parameters that can be changed with the SWLDPRM entry is given below:

Parameter	Type	Description
GMCHK	Integer >= 0 (Default = 0)	For PARTPAT only. 0 - No geometry checks 1 - Check angle between projected shell elements 2 - Check angle between projected shell elements and if an error is encountered, output candidate shell element pairs

6

Parameter	Type	Description
GSMOVE	Integer >= 0 (Default = 0)	Maximum number of times to move GS if a complete projection of all points has not been found (PARTPAT and ELPAT only)
GSPROJ	Real (Default = 20.0)	Maximum permissible angle between normals of shell A and shell B. If the angle between shell normals is exceeded, no weld element is generated
NREDIA	Integer >=0 (Default = 0)	Maximum number of times to reduce weld diameter if a complete projection of all points has not been found (PARTPAT and ELPAT only)
PROJTOL	Real >= 0.0 (Default = 0.0)	Tolerance to accept projected points GA or GB if the projected point lies outside the shell element but is located within PROJTOL*(dimension of shell element)
GSTOL	Real >=0.0 (Default = 0.0)	For PARTPAT and ELPAT only If GSTOL > 0.0 and the distance between GS and the projected point GA or GB is greater than GSTOL, the weld is rejected
PRTSW	Integer >=0 (Default = 0)	Switch to print diagnostic output 0 = no diagnostic output 1 = print diagnostic output (connectivity information) to .out file 2 = print diagnostic output (connectivity information) to .out file and PLOTEL information to update file if update file has been requested

6.7.168 TABDMP1

Data Entry: **TABDMP1** - Modal Damping Table

Description: Defines modal damping as a tabular function of frequency.

Format:

1	2	3	4	5	6	7	8	9	10
TABDMP1	ID	TYPE							
+	F1	G1	F2	G2	F3	G3	-etc.-		

Example:

1	2	3	4	5	6	7	8	9	10
TABDMP1	5								
+	1.5	0.01142	2.5	0.01607	ENDT				

Field	Information	Description
2	ID	Table identification number (Integer > 0).
3	TYPE	Character data which indicates the type of damping units, G, CRIT, Q or blank. Default is G.
2, 4, ...	Fi	Frequency value in cycles per unit time (Real $\geq$ 0.0).
3, 5, ...	Gi	Damping value (Real).

Remarks:

1. The F_i must be in either ascending or descending order, but not mixed order.
2. Jumps between two points ($F_i = F_{i+1}$) are permitted, but not at the end points.
3. At least two entries must be present.
4. Any F_i, G_i entry may be ignored by placing the character data SKIP in both of the two fields used for that entry.
5. The end of the table is indicated by the existence of the character string ENDT in the first of the two fields following the last entry. An error is detected if any continuation lines follow the line containing the end-of-table flag, ENDT.
6. The TABDMP1 mnemonic infers the use of the algorithm $g = g_T(F)$, where F is input to the table and g is returned. The table look-up $g_T(F)$ is performed using linear interpolation within the table and linear extrapolation outside the table, using the last two end points at the appropriate table end. At jump points, the average $g_T(F)$ is used. There are no error returns from this table look-up procedure.
7. Modal damping tables must be selected by the Solution Control data **SDAMPING** = ID.

8. This form of damping is used only in modal formulation of frequency response analysis.
9. If TYPE is “G” or blank, the damping values g_1, g_2, etc. are in structural damping units, that is, the value of g in (1+ig)K. Therefore, $b_i = g \cdot k_i / \omega_i$. If TYPE is “CRIT”, the damping values g_1, g_2, etc. are in the units of fraction of critical damping, C/C0. If TYPE is “Q”, the damping values g_1, g_2, etc. are in the units of the amplification of quality factor, Q. These constants are related by the following equations;

$$C / C_0 = g / 2$$

$$Q = 1 / (2C / C_0) = 1 / g$$

10. If PARAM KDAMP=1 (Default), then the modal damping is added to the B matrix. If KDAMP=-1, then the modal damping is added to the (1+ig)K matrix.

6.7.169 TABLED1

Data Entry: **TABLED1** - Dynamic Load Tabular Function, Form 1

Description: Defines a tabular function for use in generating frequency dependent dynamic loads.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED1	ID								
+	X1	Y1	X2	Y2	X3	Y3	-etc.-		

Example:

1	2	3	4	5	6	7	8	9	10
TABLED1	25								
+	-1.0	5.9	2.0	5.2	4.0	5.4	ENDT		

Field	Information	Description
2	ID	Table identification number (Integer > 0).
2, 4, ...	Xi	Tabular variable (Real).
3, 5, ...	Yi	Tabular function value (Real).

Remarks:

1. The X_i must be in either ascending or descending order, but not mixed order.
2. Jumps between two points ($X_i = X_{i+1}$) are permitted, but not at the end points.
3. At least two entries must be present.
4. Any X-Y entry may be ignored by placing the character data SKIP in both of the two fields used for that entry.
5. The end of the table is indicated by the existence of the character string ENDT in the first of the two fields following the last entry. An error is detected if any continuation lines follow the line containing the end-of-table flag ENDT.
6. Each TABLEDi mnemonic infers the use of a specific algorithm. For TABLED1 type tables, this algorithm is $Y = y_T(X)$, where X is input to the table and Y is returned. The table look-up, $y_T(x)$, x=X, is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. At jump points, the average $y_T(x)$ is used. There are no error returns from this table look-up procedure.

6.7.170 TABLED2

Data Entry: **TABLED2** - Dynamic Load Tabular Function, Form 2

Description: Defines a tabular function for use in generating frequency dependent dynamic loads. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED2	ID	X1							
+	x1	y1	x2	y2	x3	y3	-etc.-		

Example:

1	2	3	4	5	6	7	8	9	10
TABLED2	25	-12.5							
+	-1.0	-5.9	2.0	-5.2	4.0	2.1	7.0	5.6	
+	SKIP	SKIP	9.0	5.2	ENDT				

Field	Information	Description
2	ID	Table identification number (Integer > 0).
3	X1	Table parameter (Real)
2, 4, ...	xi	Tabular variable (Real).
3, 5, ...	yi	Tabular function value (Real).

Remarks:

1. The x_i must be in either ascending or descending order, but not mixed order.
2. Jumps between two points ($x_i = x_{i+1}$) are permitted, but not at the end points.
3. At least two entries must be present.
4. Any x-y entry may be ignored by placing the character data SKIP in both of the two fields used for that entry.
5. The end of the table is indicated by the existence of the character string ENDT in the first of the two fields following the last entry. An error is detected if any continuation lines follow the line containing the end-of-table flag ENDT.
6. Each TABLEDi mnemonic infers the use of a specific algorithm. For TABLED2 type tables, this algorithm is $Y = y_T(X - X1)$, where X is input to the table and Y is returned. The table look-up, $y_T(x)$, $x=X-X1$, is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. At jump points, the average $y_T(x)$ is used. There are no error returns from this table look-up procedure.

6.7.171 TABLED3

Data Entry: **TABLED3 -** Dynamic Load Tabular Function, Form 3

Description: Defines a tabular function for use in generating frequency dependent dynamic loads. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED3	ID	X1	X2						
+	x1	y1	x2	y2	x3	y3	-etc.-		

Example:

1	2	3	4	5	6	7	8	9	10
TABLED3	25	-12.5	3.5						
+	-1.0	-5.9	2.0	-5.2	4.0	2.1	7.0	5.6	
+	SKIP	SKIP	9.0	5.2	ENDT				

Field	Information	Description
2	ID	Table identification number (Integer > 0).
3	X1	Table parameter (Real)
4	X2	Table parameter (Real≠0)
2, 4, ...	xi	Tabular variable (Real).
3, 5, ...	yi	Tabular function value (Real).

Remarks:

1. The x_i must be in either ascending or descending order, but not mixed order.
2. Jumps between two points ($x_i = x_{i+1}$) are permitted, but not at the end points.
3. At least two entries must be present.
4. Any x-y entry may be ignored by placing the character data SKIP in both of the two fields used for that entry.
5. The end of the table is indicated by the existence of the character string ENDT in the first of the two fields following the last entry. An error is detected if any continuation lines follow the line containing the end-of-table flag ENDT.

6. Each TABLEDi mnemonic infers the use of a specific algorithm. For TABLED3 type tables, this algorithm is $Y = y_T\left(\frac{X - X1}{X2}\right)$, where X is input to the table and Y is returned. The table look-up, $y_T(x)$, $x = \left(\frac{X - X1}{X2}\right)$, is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points at the appropriate table end. At jump points, the average $y_T(x)$ is used. There are no error returns from this table look-up procedure.

6.7.172 TABLED4

Data Entry: **TABLED4 -** Dynamic Load Tabular Function, Form 4

Description: Defines a tabular function for use in generating frequency dependent dynamic loads. Also contains parametric data for use with the table.

Format:

1	2	3	4	5	6	7	8	9	10
TABLED4	ID	X1	X2	X3	X4				
+	A0	A1	A2	A3	A4	A5	-etc.-		

Example:

1	2	3	4	5	6	7	8	9	10
TABLED4	24	0.0	2.0	0.0	10.0				
+	1.8	-0.021	-3.22-4	0.0	5.2-7	ENDT			

Field	Information	Description
2	ID	Table identification number (Integer > 0).
3	X1	Table parameter (Real)
4	X2	Table parameter (Real≠0)
5	X3	Table parameter (Real; X3 < X4)
6	X4	Table parameter (Real; X3 < X4)
2, 3, ...	Ai	Coefficient entries (Real).

Remarks:

1. At least one entry must be present.
2. The end of the table is indicated by the existence of the character string ENDT in the first of the two fields following the last entry. An error is detected if any continuation lines follow the line containing the end-of-table flag ENDT.
3. Each TABLEDi mnemonic infers the use of a specific algorithm. For TABLED4 type tables, this algorithm is $Y = \sum_{i=0}^{N} A_i \left(\frac{X - X1}{X2}\right)^i$, where X is input to the table and Y is returned. Whenever X < X3, use X3 for X; whenever X > X4, use X4 for X. There are N+1 entries in the table. There are no error returns from this table look-up procedure.

6

6.7.173 TABRND1

Data Entry: **TABRND1** - Power Density as a Tabular Function referenced by RANDPS.

Description: Defines a tabular function for use in generating power density values as a function of frequencies

Format:

1	2	3	4	5	6	7	8	9	10
TABRND1	ID								
+	f1	G1	f2	G2	f3	G3	-etc.-		

Example:

1	2	3	4	5	6	7	8	9	10
TABRND1	25								
+	1.0	6.9	2.0	5.2	4.0	5.4	ENDT		

Field	Information	Description
2	ID	Table identification number (Integer > 0).
2, 4, ...	fi	Frequency value in cycles per unit time (Real $\geq$ 0.0).
3, 5, ...	Gi	Power spectra density value (Real).

Remarks:

1. This data is referenced by **RANDPS** entries.
1. The f_i must be in either ascending or descending order, but not mixed order.
2. Jumps between two points ($f_i = f_{i+1}$) are permitted, but not at the end points.
3. At least two entries must be present.
4. Any f-G entry may be ignored by placing the character data SKIP in both of the two fields used for that entry.
5. The end of the table is indicated by the existence of the character string ENDT in the first of the two fields following the last entry. An error is detected if any continuation lines follow the line containing the end-of-table flag ENDT.
6. The TABRND1 mnemonic infers the use of the algorithm $G = G_T(f)$, where f is input to the table and G is returned. The table look-up $G_T(f)$ is performed using linear interpolation within the table and linear extrapolation outside the table, using the last two end points at the appropriate table end. At jump points, the average $G_T(f)$ is used. There are no error returns from this table look-up procedure.

6

6.7.174 TEMP

Data Entry: **TEMP** - Grid Point Temperature Field.

Description: Defines temperature at grid points for determination of thermal loading and stress recovery.

Format:

1	2	3	4	5	6	7	8	9	10
TEMP	SID	G	T	G	T	G	T		

Example:

1	2	3	4	5	6	7	8	9	10
TEMP	8	100	100.0						

Field	Information	Description
2	SID	Temperature set identification number (Integer > 0).
3,5,..	G	**GRID** identification number (Integer > 0). The second and third G can be blank.
4,6,..	T	Temperature (Real). The second and third T can be blank.

Remarks:

1. Temperature load sets must be selected in the Solution Control Section (**TEMPERATURE** = SID or TEMP(LOAD) = SID).
2. From one to three grid point temperatures may be defined on a single data entry.
3. Average element temperatures are obtained as a simple average of the connecting grid point temperatures.
4. If the fifth field is blank, the seventh field must also be blank.

6

6.7.175 TEMPD

Data Entry: **TEMPD -** Grid Point Temperature Field Default.

Description: Defines a temperature value for all grid points of the structural model which have not been given a temperature on a **TEMP** data entry.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPD	SID	T	SID	T	SID	T	SID	T	

Example:

1	2	3	4	5	6	7	8	9	10
TEMPD	3	120.0	4	240.	5	360.0			

Field	Information	Description
2,4,..	SID	Temperature set identification number (Integer > 0).
3,5,..	T	Default temperature value (Real).

Remarks:

1. Temperature load sets must be selected in the Solution Control Section (**TEMPERATURE** = SID).
2. From one to four default temperatures may be defined on a single data entry.
3. Average element temperatures are obtained as a simple average of the connecting grid point temperatures.
4. Only one default temperature may be specified for each temperature load set.

6.7.176 USET

Data Entry: **USET** - User Degrees of Freedom Set Definition.

Description: Defines a user set of degrees of freedom.

Format:

1	2	3	4	5	6	7	8	9	10
USET	NAME	G	C	G	C	G	C		

Example:

1	2	3	4	5	6	7	8	9	10
USET	U6	2	246						

Field	Information	Description
2	NAME	Set name (Character)
3,5,7	G	**GRID** or **SPOINT** identification numbers (Integer > 0).
4,6,8	C	Component number of global coordinate (Any unique combination of the digits 1-6 with no embedded blanks. Blank for scalar points.)

Remarks:

1. Continuation data is not allowed.
2. If a USET named “U6” is defined, then degrees of freedom in that set will be used to create target vectors for residual vector calculations in natural frequency and/or modal frequency response loadcases.

6.7.177 USET1

Data Entry: **USET1 -** User Degrees of Freedom Set Definition.

Description: Defines a user set of degrees of freedom.

Format:

1	2	3	4	5	6	7	8	9	10
USET1	NAME	C	G1	G2	G3	G4	G5	G6	
+	G7	G8	G9	-etc.-					

Example:

1	2	3	4	5	6	7	8	9	10
USET1	U6	2456	2	4	6	8	10	12	
+	14								

Alternate Format:

1	2	3	4	5	6	7	8	9	10
USET1	NAME	C	GID1	“THRU”	GID2				

Example:

1	2	3	4	5	6	7	8	9	10
USET1	U6	2456	2	THRU	12				

Field	Information	Description
2	NAME	Set name (Character)
3	C	Component number of global coordinate (Any unique combination of the digits 1-6 with no embedded blanks. Blank for scalar points.)
4,5..	Gi, GIDi	**GRID** or **SPOINT** identification numbers (Integer > 0).

Remarks:

1. If a USET named “U6” is defined, then degrees of freedom in that set will be used to create target vectors for residual vector calculations in natural frequency and/or modal frequency response loadcases.
2. Continuation data entries are optional.

CHAPTER 7

Output Files

7

7.1 Summary of *GENESIS* Analysis Files

INFORMATION	CONTROL	FILE NAME
Input Data	created by user	*file*.dat
User Supplied Stiffness Matrix	created by user	Defined by K2UU command
User Supplied Mass Matrix	created by user	Defined by M2UU command
Output Data	created always	*file*.out
Run Log	created always	*file*.log
Post-Processing File (OUTPUT2 format)	STRESS=POST, etc.	*pname*xx.op2 *pname*xx.op2.gz *pname*=xxxx=.op2 *pname*=xxxx=.op2.gz
Post-Processing File (PUNCH format)	STRESS=POST, etc.	*pname*xx.pch *pname*=xxxx=.pch
Post-Processing File (BINARY, FORMAT and PLOT formats)	STRESS=POST, etc.	*pname*xx.PST *pname*=xxxx=.PST
Post-Processing File (IDEAS format)	STRESS=POST, etc.	*pname*xx.unv *pname*=xxxx=.unv
Post-Processing Files (PATRAN format)	STRESS=POST, etc.	*pname*xxyy.[dis, rxn,gpf,tmp,els,eln, elf,gps] *pname*xxyyzz.eig, *pname*xxyyzz.---
Reduced Matrix and Recovery MPC File	ALOAD=DMIG, KAA=DMIG, MAA=DMIG, K4AA=DMIG PARAM,SEMPC	*pname*xx.DMIG *pname*=xxxx=.DMIG
Guyan Reduced Stiffness Matrices File	KAA=POST	*pname*xxyy.KAA *pname*xxyyyyyyyy.KAA
Guyan Reduced Mass Matrices File	MAA=POST	*pname*xxyy.MAA *pname*xxyyyyyyyy.MAA
Eigen Database File	PARAM,MODERST	*pname*.EIGDB

xx = Design Cycle number (0-99), xxxx = Design Cycle number (> 99).

yy = Loadcase number modulo 100, yyyyyyyy = Loadcase number.

zz = Loading frequency number

7.2 Program Output

The name of the program output file depends on how *GENESIS* was installed on your system. On most systems the results go to a file with the same base name as the input file and with the extension ".out". On other systems it will be the file name assigned to unit 6. If *GENESIS* stops because of an error, a detailed error message will be printed in the output file. The output file contains up to four sections.

The number of lines per page and the number of characters per line in the output file can be controlled by the solution control command **LINE**.

In addition to the output file, Genesis will also create a log file that contains statistics about the system environment as well as any output that was printed to the terminal console. The log file also contains the exact start and stop times for the Genesis process, from which the total elapsed execution time can be calculated.

Unsorted Input Data

This section contains the input data exactly as it appears in the input data file. This data is produced by the solution control section command **ECHO**=UNSORT or ECHO=BOTH. Portions of the input can be selected or deselected for printing in this section by using the **ECHOON** and **ECHOOFF** statements.

Sorted Input Data

This section contains a summary of the input data after it has been processed by *GENESIS*. This data is produced by the solution control section command ECHO=SORT or ECHO=BOTH.

Model Summary

This section contains tables of the analysis and design problem sizes and load case summary. The analysis table contains the number of grids, elements, degrees of freedom, etc. The load case table contains a summary with type and number of load cases. These tables are printed with the solution control command **SUMMARY** = YES (the default).

Analysis Results

This section contains the analysis results requested in the solution control section of the input data.

7.3 Post-Processing Data

Data for plotting grid point displacements, velocities, accelerations, element forces, reaction forces, applied loads, stresses, strains, and mode shapes and temperatures can be generated by *GENESIS*. A separate file is written for each design cycle in which the analysis results have been requested. Only results that have been requested in the solution control section of the input data are written to the files. For example, to get stress data, one of the commands; STRESS=ALL, STRESS=n or STRESS=POST must appear.

The format of the post-processing files is determined by the executive control command **POST**. Post-processing files can be binary (POST=BINARY), 80 column ASCII formatted (POST=FORMAT), formatted with the structure definition data (POST=PLOT), PATRAN neutral file format (POST=PATRAN), NASTRAN OUTPUT2 format (POST=OUTPUT2), NASTRAN PUNCH format (POST=PUNCH) or IDEAS universal file format (POST=IDEAS).

The files have the name *pname*xx.*ext*, where *pname* is set to the base of the input filename, xx is the design cycle number, and *ext* is `op2` for OUTPUT2 format, `pch` for PUNCH format, `unv` for IDEAS format, `PST` for BINARY, FORMAT, or PLOT format. For PATRAN format, the files are named *pname*xxyy.[dis,rxn,gpf,tmp,els,eln,elf,gps], *pname*xxyyzz.eig, *pname*xxyyzz.---, where xx is the design cycle number, yy is the LOADCASE number and zz is the loading frequency number.

Note that if the design cycle is greater than 99, the filename pattern changes so that the design cycle is 4 digits bracketed by '=' characters: *pname*=xxxx=.*ext.*

Displacements, velocities, accelerations and mode shapes are written out in the basic coordinate system. If results should be in the general coordinate system, the Solution Control Command **POSTOUTPUT** = GENERAL must be used. Applied loads and reaction forces are always in the general coordinate system.

7.3.1 GENESIS Format Post-processing Files

If **POST**=BINARY, FORMAT or PLOT, each post processing file is written using the following procedure:

LOOP through result type
    LOOP through load cases (in internal order)
        LOOP through loading frequencies
            LOOP through element types or mode numbers
                WRITE Header
                LOOP through element ID or grid ID (in internal order)
                    WRITE Results
                END LOOP
            END LOOP
        END LOOP
    END LOOP
END LOOP

Each result set has a header consisting of five integers: IRTYPE, IETYPE or IMODE or ILFREQ, LOAD, NREC, and NWORD. These are defined as follows:

IRTYPE - The result type:
1=displacement
2=stress
3=strain
4=force
5=mode shape
6=grid point stress
7=reaction forces
8=applied loads
9=temperature
10=dynamic displacement
11=dynamic velocity
12=dynamic acceleration
13=dynamic stress
14=dynamic strain
15=dynamic force
16=dynamic grid point stress
17=grid mass
18=list of ASET grids
19=element strain energy
0= end of file

IETYPE or IMODE or ILFREQ - The element type or mode number or loading frequency number. See **Postprocessing File with Structure Definition** for element type codes. IETYPE=0 for grid displacements, grid point stresses, reaction forces, applied loads, temperatures and grid mass. IETYPE=User ASET Set ID for list of ASET grids. ILFREQ is the loading frequency number for dynamic displacement, velocity, acceleration and grid point stress results, LOAD - The user defined load case ID.

NREC - The number of records of results:

= number of grids for displacements, grid point stresses, reaction forces, applied loads, temperatures and grid mass.

= number of elements of one type for stress, strain, and force.

= number of elements multiplied by the maximum number of layers for which results are requested for composite element stresses and strains.

= number of grids plus one for mode shapes.

= number of grids plus one for dynamic displacements, velocities, accelerations and grid point stresses.

= number of elements plus one for dynamic stresses, strains and forces.

= number of ASET grids for list of ASET grids

NWORD - The number of real (double precision) numbers (words) per record.

= 6 for displacements, mode shapes, reaction forces and applied loads.

= the number of items per element for stress, strain, and force results. See **Number of Element Results**.

= 13 for grid point stresses.

= 1 for grid point temperatures and grid mass.

= 12 for dynamic displacements, velocities and accelerations.

= two times the number of items per element for dynamic stress, strain and force results.

= 26 for dynamic grid point stresses.

= 0 for list of ASET grids.

Each line of results data has 1 integer value (the user defined grid or element ID) followed by NWORD real (double precision) values.

For mode shape output, the first line has a grid ID=0 and the first real number is the frequency in cycles/time.

For dynamic analysis results, the first line has grid or element ID=0, and the first real number is the loading frequency. For each item, the real and imaginary component or magnitude and phase are output, depending on the solution control command DYNOUTPUT.

To get formatted post processing data, use the executive section command POST=FORMAT or just POST. The two format statements used to write the output are:

Header:

```
FORMAT(5I8)
```

Data:

```
FORMAT(I8,5(1X,E13.7)/(8X,5(1X,E13.7)))
```

To get binary output use the executive section command POST=BINARY. The supplied program “RPOST.FOR” is an example of how to read this data.

7

It is suggested that the user first use the formatted output to become familiar with the output format. After the output format is understood it is suggested that RPOST.FOR be modified and used to read the binary output. The binary output will be more efficient for larger problems.

Number of Element Results

Element Class	# of Forces	# of Stresses	# of Strains
ROD	2	2	0
BAR	12	8*	0
BEAM	8-88**	5-55**	0
BUSH	6	6	6
PSHELL	8	14	22
SOLID	0	13	13
ELAS1/2	1	1	0
SHEAR	16	6	0
DAMP1/2	1	0	0
VISC	2	0	0
PCOMP	0	9	10
PWELD	8	7	0

* BAR elements defined in the design element library have different numbers of stresses depending on the element type. See Element Library Design Variable to Property Relationships (DVPROP3) in the Design Reference manual for the number of stresses calculated for a particular element.

** The number of BEAM force results is 8 times the maximum number of sections defined on any PBEAM in the input file. The number of BEAM stress results is 5 times the maximum number of sections of any PBEAM. Elements referencing PBEAM entries with fewer than the maximum number of sections are padded at the end with zeroes so that all beam elements have the same number of results.

The order of all the element results, except BEAM and PCOMP, is the same as the item code order for responses. The item code order for each element type is listed in the DRESP1 data description. For composite element stresses and strains, the orders are;

1	Layer ID	Layer ID
2	σ_1	ε_1
3	σ_2	ε_2
4	τ_{12}	γ_{12}
5	σ_I	ε_I
6	σ_{II}	ε_{II}
7	τ_{max}	γ_{max}
8	σ_{VM}	γ_{VM}
9	FP	FP
10		FMODE

For BEAM forces, the result order is:

1	X/XB
2	Axial force
3	Shear 1
4	Shear 2
5	Torque
6	Moment 2
7	Moment 1
8	Bimoment
9 - 8*$N_{Stations}$	Repeat 1-8 for each station

7

For BEAM stresses, the result order is;

1	X/XB
2	Bending + Axial Stress at C
3	Bending + Axial Stress at D
4	Bending + Axial Stress at E
5	Bending + Axial Stress at F
6 - 5*$N_{Stations}$	Repeat 1-5 for each station

Sample Program to Read Binary Post File (RPOST.FOR)

```
      PROGRAM RPOST
C
C     **************************************************************
C     EXAMPLE PROGRAM FOR READING BINARY POST DATA
C     **************************************************************
C
      IMPLICIT DOUBLE PRECISION (A-H,O-Z)
C
      DOUBLE PRECISION STRESS(22)
C
C     OPEN BINARY POST DATA FILE
C
      OPEN (20, FILE='DES01.PST', STATUS='OLD', FORM='UNFORMATTED',
     * IOSTAT=KERR, ERR=997)
C
C     READ HEADER
C
100   READ(20) ICODE,IETYPE,LOAD,NREC,NWORD
C
C     CHECK FOR END OF FILE
C
      IF (ICODE.EQ.0) GO TO 997
C
      IF (ICODE.EQ.1) THEN
C
C        ICODE=1 = DISPLACEMENTS
C        READ IN TRANSLATIONAL DISPLACEMENTS AND WRITE TO SCREEN
C
         WRITE(6,*) ' DISPLACEMENTS FOR LOADCASE ',LOAD
         WRITE(6,*) ' GRID ID X Y Z'
         DO 200 I = 1,NREC
```

7

```
            READ(20) IGID,X,Y,Z
            WRITE(6,*) IGID,X,Y,Z
200      CONTINUE
C
      ELSE IF (ICODE.EQ.2 .AND. IETYPE.EQ.7) THEN
C
C     READ IN  ELEMENT (IETYPE=7) STRESSES (ICODE=2) AND
C     WRITE VON MISES STRESS (LOCATION NUMBER 7) TO SCREEN
C
         WRITE(6,*) '  VON MISES STRESSES FOR LOADCASE ',LOAD
        WRITE(6,*) ' ELEM ID STRESS'
DO       300 I = 1,NREC
            READ(20) IEID,(STRESS(J),J=1,NWORD)
            WRITE(6,*) IEID,STRESS(7)
300      CONTINUE
C
      ELSE
C
C     READ OVER UNWANTED OUTPUT
C
         DO 400 I = 1,NREC
            READ(20) IDUMMY
400      CONTINUE
      END IF
C
C     GO TO 100 AND READ NEXT HEADER
C
      GO TO 100
C
997   IF (KERR.NE.0) WRITE(6,*) ' CAN'T OPEN POST FILE, REASON:',KERR
C
      END
```

7

Postprocessing File with Structure Definition

Sometimes it is convenient to have the structure definition in the analysis results file. This can be achieved using the command POST=PLOT. In this case the grid point locations and element connectivities will be placed ahead of the analysis results in the ".PST" file. The analysis data will be in the same format that is generated with the command POST=FORMAT. The first two lines of data contain the TITLE and SUBTITLE respectively, and are written using

```
FORMAT (A72)
```

The third line contains the number of grids, the number of elements, the total number of static loadcases and loadcoms, the total number of requested frequencies, the total number of heat transfer loadcases, the total number of dynamic loading frequencies, and the number of local coordinate systems, and is written using

```
FORMAT (7I8)
```

This is followed by the grid point locations in the basic coordinate system. Each record of data contains the grid point ID, the three coordinates, and the grid point results coordinate system ID, and is written using

```
FORMAT (I8,3(1X,E13.7),I8)
```

If the Solution Control command POSTOUTPUT = BASIC, the default, then the coordinate system ID is always 0.

Following the grid points are the element connectivities. Each record contains the element ID, element type code, element property ID, maximum number of grids used to define the element, the associated grid points, and the element results coordinate system for 2D (PSHELL) and 3D (PSOLID) elements, and is written using

```
FORMAT (10I8)
```

Note that the missing grids on the CHEX20, CELAS1, CDAMP1, CMASS1, CBUSH and CHBDY elements have zero ID's. See PSHELL and PSOLID data descriptions for the meaning of the element results coordinate system codes.

Following the elements are the local coordinate systems. Each record contains the coordinate system ID, type (1: rectangular, 2: cylindrical and 3: spherical), location of the origin, and 3X3 transformation matrix, and is written using

```
FORMAT (2I8/3(1X,E13.7)/3(1X,E13.7)/3(1X,E13.7)/3(1X,E13.7))
```

The following table lists the element codes and maximum number of grids per element.

ELEMENT TYPE	CODE	MAXIMUM NUMBER OF GRIDS
ROD	1	2
BAR	2	2
BEAM	27	2
BUSH	28	2
QUAD4 (PSHELL)	3	4
QUAD4 (PCOMP)	20	4
TRIA3 (PSHELL)	4	3
TRIA3 (PCOMP)	21	3
QUAD8 (PSHELL)	5	8
QUAD8 (PCOMP)	33	8
TRIA6 (PSHELL)	6	6
TRIA6 (PCOMP)	34	6
HEXA (8 NODES)	7	8
PENTA(6 NODES)	8	6
(9-21 NODES) HEX20	9	21
PENTA(15 NODES)	35	15
PYRA(5 NODES)	41	5
PYRA(13 NODES)	42	13
ELAS1/2	10	2
CONM2	11	1
TETRA (4 NODES)	12	4
TETRA (10 NODES)	19	10
TRIAX6	22	6

7

CONM3	13	1
HBDY	14	4
SHEAR	15	4
DAMP1/2	16	2
MASS1/2	17	2
VISC	18	2

Following each BAR and BEAM element connectivity information is the element's orientation vector in the basic coordinate system. This is written using:

```
FORMAT (3(1X,E13.7))
```

7.3.2 PATRAN 2.5 Format Results Files

Results files that can be used directly with PATRAN can be generated using the command POST=PATRAN. When this command is used the results are written to separate files for each result type and each load case (and each dynamic loading frequency), instead of the one large file generated with the FORMAT, BINARY, and PLOT commands. The last two digits of the **LOADCASE** or **LOADCOM** ID is included in the file name. Because of this the LOADCASE and LOADCOM ID's must be less than 100 when using the POST=PATRAN command.

To get grid point results in the general coordinate system, use the solution control command **POSTOUTPUT** = GENERAL.

When dynamic analysis results are requested, the real and imaginary component or magnitude and phase of each item are written out. The form of the output can be controlled by the solution control command **DYNOUTPUT**.

Displacement results are written to files with the name *pname*xxyy.dis, where xx is the design cycle number and yy is the user defined LOADCASE or LOADCOM ID.

Reaction forces are written to files with the name *pname*xxyy.rxn, where xx is the design cycle number and yy is the user defined LOADCASE ID.

Applied loads are written to files with the name *pname*xxyy.gpf, where xx is the design cycle number and yy is the user defined LOADCASE ID.

Temperature results are written to files with the name *pname*xxyy.tmp, where xx is the design cycle number and yy is the user defined LOADCASE ID.

Element stresses are written to files with the name *pname*xxyy.els where xx is the design cycle number and yy is the user defined LOADCASE or LOADCOM ID. The column numbers of the stress results correspond to the stress item codes found with DRESP1 for all elements except composites, for which the codes are listed on p. 718.

Element strains are written to files with the name *pname*xxyy.eln where xx is the design cycle number and yy is the user defined LOADCASE or LOADCOM ID. The column numbers of the strain results correspond to the strain item codes found with DRESP1 for all elements except composites, for which the codes are listed on p. 718.

Element forces are written to files with the name *pname*xxyy.elf where xx is the design cycle number and yy is the user defined LOADCASE or LOADCOM ID. The column numbers of the force results correspond to the force item codes found with DRESP1.

Grid point stresses are written to files with the name *pname*xxyy.gps where xx is the design cycle number and yy is the user defined LOADCASE or LOADCOM ID. The column numbers of the solid element grid point stresses correspond to the grid stress item codes found with DRESP1.

Mode shapes are written to files with the name *pname*xxyyzz.eig where xx is the design cycle number, yy is the user defined LOADCASE ID, and zz is the mode number. Note that modes greater than 99 cannot be output. Also beware that the file name is quite long, which may cause problems on computers that have a filename length limit. The LOADCASE LABEL is replaced by the mode frequency value in the file header.

The file names for dynamic analysis results are *pname*xxyyzz.---, where xx is the design cycle number, yy is the user defined LOADCASE ID and zz is the loading frequency number. The value of the loading frequency is included at the end of the LOADCASE LABEL in the file header.

7.3.3 NASTRAN OUTPUT2 Format Results Files

Results files that can be used by software that reads NASTRAN OUTPUT2 files can be generated using the command **POST** = OUTPUT2. One file is written per design cycle. In *GENESIS*, the default is to write the NASTRAN “TAPE LABEL” in the first eight records (NASTRAN OUTPUT2 parameter ITAPE=-1). To **not** write the NASTRAN “TAPE LABEL” (NASTRAN OUTPUT2 parameter ITAPE=0), use PARAM, **TAPELBL**, 0 in the bulk data section of the input data. The data block names, approach and device codes, and record contents of the files are summarized below.

For dynamic analysis results, the magnitude/phase or real/imaginary components are output, depending on the solution control command **DYNOUTPUT**.

To get grid point results in the general coordinate system, use the Solution Control Command **POSTOUTPUT** = GENERAL.

OUTPUT2 files can be automatically compressed to reduce disk space requirements. The PARAMeter **OP2ZIP** controls the compression. OP2ZIP can take a value of 0 to 6. For values > 0, the OUTPUT2 files will be compressed in the standard "gzip" format and their file extension will be "`.op2.gz`".

OP2ZIP value	Effect
0	No compression will be used.
1	Lossless compression.
2	Compression with rotation components on displacement, velocity, acceleration and eigenvector results set to zero.
3, 4, 5, 6	Compression with rotation components on displacement, velocity, acceleration and eigenvector results set to zero and all element and grid result values reduced in precision by truncating the floating point representation, with higher values resulting in a greater loss of digits. At OP2ZIP = 4, there will be error in the fifth significant digit. At OP2ZIP = 6 there will be error in the third significant digit.

While higher values typically result in smaller file sizes, this comes at the price of loss of information. It is not recommended to use values above 1 if the result data is intended for any purposes other than visualization.

The PARAMeter **ZIPLVL** controls the aggressiveness of the compressor engine when writing compressed OUTPUT2 files. Using a higher level will typically result in smaller file sizes at the expense of increased CPU time. This parameter is only effective when parameter OP2ZIP > 0.

Data Block Names

OUGV1	Displacements
OES1	Element Stresses
OEF1	Element Forces
OSTR1	Element Strains
ONRGY1	Element Strain Energies
OPHIG	Eigenvectors
OGS1	Grid Point Stresses
OQG1	Reaction Forces
OPG1	Applied Loads
OCU1	Contact Clearances
OCF1	Contact Forces
OCP1	Contact Pressures
TOUGV1	Temperatures
OUPVC1	Dynamic Displacements, Velocities and Accelerations
OESC1	Dynamic Stresses
OSTRC1	Dynamic Strains
OEFC1	Dynamic Forces
OGSC1	Dynamic Grid Point Stresses

Notes:

1. The device code is 2 for POST only and 3 for both POST and output file.
2. OUGV1, OQG1, OPG1 and TOUGV1 are the same as with NASTRAN. The approach code is always 1. OUPVC1 is the same as with NASTRAN, and the approach code is 5.
3. OPHIG is the same as with NASTRAN. The approach code is always 2.
4. OES1 and OSTR1 are the same as with NASTRAN except as shown below. The approach code is always 1.
5. OEF1 is the same as with NASTRAN except as shown below. The approach code is always 1.
6. OESC1, OSTRC1 and OEFC1 are the same as with NASTRAN except as shown below. The approach code is always 5.
7. OGS1 is the same as with the NASTRAN volume principal stress format except that the approach code is always 1 and the volume ID is always 1.

8. OGSC1 is the same as NASTRAN Data Block OGS1, except that word 2 of record 1 is 1027, the volume ID is always 1, the approach code is always 5, word 6 of record 1 is the loading frequency, and the format code is 3 for magnitude/phase and 2 for real/imaginary.
   Record 2 has the form:

Word	Information
2	Normal-x RM
3	Normal-y RM
4	Normal-z RM
5	Shear-xy RM
6	Shear-yz RM
7	Shear-zx RM
8	Normal-x IP
9	Normal-y IP
10	Normal-z IP
11	Shear-xy IP
12	Shear-yz IP
13	Shear-zx IP

9. If a read error occurs when using the *GENESIS* OUTPUT2 file, try using PARAM, TAPELBL, 0 in the bulk data section of the input data.

Contents of Record 2 for Stresses and Strains

ELEMENT		WORD	*GENESIS* ITEM	NASTRAN ITEM
TYPE	NAME			
1	CROD (no strain)	2	Axial stress at end A	Axial stress
		3	0.0	Axial safety margin
		4	0.0	Torsional stress
		5	0.0	Torsional safety margin
2	CBEAM (no strain)	2	Grid ID	Grid ID
		3	X/XB	X/XB
		4	Bending + Axial at C	Longitudinal Stress at C
		5	Bending + Axial at D	Longitudinal Stress at D
		6	Bending + Axial at E	Longitudinal Stress at E
		7	Bending + Axial at F	Longitudinal Stress at F
		8	Maximum of 4-7	Maximum Stress
		9	Minimum of 4-7	Minimum Stress
		10	0.0	Margin of Safety in Tension
		11	0.0	Margin of Safety in Compression
		12-111	Repeat items 2-11 for each station. All zeroes are used for non-existent stations	
4	CSHEAR (no strain)	2	Maximum Shear	Maximum Shear
		3	Average Shear	Average Shear
		4	0.0	Safety Margin
11	CELAS1 (no strain)	2	Stress	Stress

Contents of Record 2 for Stresses and Strains

33	CQUAD4	2	- Half thickness	Z1 fiber distance
		3	Normal-x at Z1	Normal-x at Z1
		4	Normal-y at Z1	Normal-y at Z1
		5	Shear-xy at Z1	Shear-xy at Z1
		6	Shear angle at Z1	Shear angle at Z1
		7	Major principle at Z1	Major principle at Z1
		8	Minor principle at Z1	Minor principle at Z1
		9	von Mises at Z1	Max shear at Z1
		10	Half thickness	Z2 fiber distance
		11	Normal-x at Z2	Normal-x at Z2
		12	Normal-y at Z2	Normal-y at Z2
		13	Shear-xy at Z2	Shear-xy at Z2
		14	Shear angle at Z2	Shear angle at Z2
		15	Major principle at Z2	Major principle at Z2
		16	Minor principle at Z2	Minor principle at Z2
		17	von Mises at Z2	Max shear at Z2

Contents of Record 2 for Stresses and Strains

34	CBAR (no strain)	2	Bending + axial at point C - end A	SA1
		3	Bending + axial at point D - end A	SA2
		4	Bending + axial at point E - end A	SA3
		5	Bending + axial at point F - end A	SA4
		6	Average of bending + axial at points C, D, E and F at end A	Axial
		7	Max bending + axial at end A	Max SA
		8	Min bending + axial at end A	Min SA
		9	0.0	Torsional safety margin
		10	Bending + axial at point C - end B	SB1
		11	Bending + axial at point D - end B	SB2
		12	Bending + axial at point E - end B	SB3
		13	Bending + axial at point F - end B	SB4
		14	Max bending + axial at end B	Max SB
		15	Min bending + axial at end B	Min SB
		16	0.0	Safety margin in compression

Contents of Record 2 for Stresses and Strains

39	CTETRA	2	0	Stress coordinate system
		3	'GRID'	Coordinate type
		4	4	Number of active points
		5	Grid ID or 0 for center	Grid ID or 0 for center
		6	Normal-x	Normal-x
		7	Shear-xy	Shear-xy
		8	Major principle	First principle
		9	0.0	First principle cos-x
		10	0.0	Second principle cos-x
		11	0.0	Third principle cos-x
		12	Mean pressure (delta volume)	Mean pressure
		13	von Mises	Octahedral
		14	Normal-y	Normal-y
		15	Shear-yz	Shear-yz
		16	Minor principle	Second principle
		17	0.0	First principle cos-y
		18	0.0	Second principle cos-y
		19	0.0	Third principle cos-y
		20	Normal-z	Normal-z
		21	Shear-zx	Shear-zx
		22	Intermediate principle	Third principle
		23	0.0	First principle cos-z
		24	0.0	Second principle cos-z
		25	0.0	Third principle cos-z
		26-109	Repeat items 5-25 for four corners Center stress values are used at corners	

7

Contents of Record 2 for Stresses and Strains

64	CQUAD8	2	'CEN/'	'CEN/'
		3	Grid ID or no. of nodes for center	Grid ID or no. of nodes for center
		4	- Half thickness	Z1 fiber distance
		5	Normal-x at Z1	Normal-x at Z1
		6	Normal-y at Z1	Normal-y at Z1
		7	Shear-xy at Z1	Shear-xy at Z1
		8	Shear angle at Z1	Shear angle at Z1
		9	Major principle at Z1	Major principle at Z1
		10	Minor principle at Z1	Minor principle at Z1
		11	von Mises at Z1	Max shear at Z1
		12	Half thickness	Z2 fiber distance
		13	Normal-x at Z2	Normal-x at Z2
		14	Normal-y at Z2	Normal-y at Z2
		15	Shear-xy at Z2	Shear-xy at Z2
		16	Shear angle at Z2	Shear angle at Z2
		17	Major principle at Z2	Major principle at Z2
		18	Minor principle at Z2	Minor principle at Z2
		19	von Mises at Z2	Max shear at Z2
		20-87	Repeat items 3-19 for four corner grids Center stress values are used at corners	

Contents of Record 2 for Stresses and Strains

66	CHEX20 or 9-21 noded C	2	0	0
		3	Normal-x	Normal-x
		4	Shear-xy	Shear-xy
		5	Major principle	First principle
		6	0.0	First principle cos-x
		7	0.0	Second principle cos-x
		8	0.0	Third principle cos-x
		9	Mean pressure (delta volume)	Mean pressure
		10	von Mises	Octahedral
		11	Normal-y	Normal-y
		12	Shear-yz	Shear-yz
		13	Minor principle	Second principle
		14	0.0	First principle cos-y
		15	0.0	Second principle cos-y
		16	0.0	Third principle cos-y
		17	Normal-z	Normal-z
		18	Shear-zx	Shear-zx
		19	Intermediate principle	Third principle
		20	0.0	First principle cos-z
		21	0.0	Second principle cos-z
		22	0.0	Third principle cos-z
67	CHEXA (8 nodes)	Same as CTETRA except that items 5-25 are repeated for eight corners for a total of 193 words.		
68	CPENTA	Same as CTETRA except that items 5-25 are repeated for six corners for a total of 151 words		
255	CPYRA	Same as CTETRA except that items 5-25 are repeated for five corners for a total of 130 words		
74	CTRIA3	Same as CQUAD4		

7

Contents of Record 2 for Stresses and Strains

86	CGAP (no strains)	2	Normal x force	Normal x force
		3	Shear y force	Shear y force
		4	Shear z force	Shear z force
		5	Delta-u displacement	Delta-u displacement
		6	Delta-v displacement	Delta-v displacement
		7	Delta-w displacement	Delta-w displacement
95	CQUAD4 (Composite)	2	Lamina Number	Lamina Number
		3	Normal-1	Normal-1
		4	Normal-2	Normal-2
		5	Shear-12	Shear-12
		6	Layer thickness	Shear-1Z
		7	Layer angle	Shear-2Z
		8	Total thickness	Shear Angle
		9	Major Principle	Major Principle
		10	Minor Principle	Minor Principle
		11	Von Mises	von Mises
96	CQUAD8 (Composite)	Same as CQUAD4 (Composite)		
97	CTRIA3 (Composite)	Same as CQUAD4 (Composite)		
98	CTRIA6 (Composite)	Same as CQUAD4 (Composite)		
102	CBUSH	2	Translation - x	Translation - x
		3	Translation - y	Translation - y
		4	Translation - z	Translation - z
		5	Rotation - x	Rotation - x
		6	Rotation - y	Rotation - y
		7	Rotation - z	Rotation - z

Contents of Record 2 for Stresses and Strains

117	CWELD (ELEMID/ GRIDID) (no strains)	2	Axial	Axial
		3	SA maximum	SA maximum
		4	SA minimum	SA minimum
		5	SB maximum	SB maximum
		6	SB minimum	SB minimum
		7	Maximum shear	Maximum shear
		8	Bearing	Bearing
118	CWELD (ELPAT, PARTPAT) (no strains)	Same as CWELD (ELEMID, GRIDID)		
200	CWELD (ALIGN) (no strains)	Same as CWELD (ELEMID, GRIDID)		

Contents of Record 2 for Solid Element Grid Point Stresses

WORD	*GENESIS*	NASTRAN
2	First principle cos-x	First principle cos-x
3	Second principle cos-x	Second principle cos-x
4	Third principle cos-x	Third principle cos-x
5	First principle cos-y	First principle cos-y
6	Second principle cos-y	Second principle cos-y
7	Third principle cos-y	Third principle cos-y
8	First principle cos-z	First principle cos-z
9	Second principle cos-z	Second principle cos-z
10	Third principle cos-z	Third principle cos-z
11	First principle	First principle
12	Second principle	Second principle
13	Third principle	Third principle
14	Mean pressure	Mean pressure
15	von Mises	Octahedral

Contents of Record 2 for Forces

ELEMENT		WORD	*GENESIS* ITEM	NASTRAN ITEM
TYPE	NAME			
1	CROD	2	Axial force	Axial force
		3	0.0	Torque
2	CBEAM	2	Grid ID	Grid ID
		3	X/XB	X/XB
		4	Bending moment in plane 1	Bending moment in plane 1
		5	Bending moment in plane 2	Bending moment in plane 2
		6	Shear force in plane 1	Web Shear in plane 1
		7	Shear force in plane 2	Web Shear in plane 2
		8	Axial Force	Axial Force
		9	Torque	Total torque
		10	Bimoment	Warping Torque
		11-110	Repeat items 2-9 for each station. Zeroes are used for non-existent stations	

Contents of Record 2 for Forces

4	CSHEAR	2	Force 4 to 1	Force 4 to 1
		3	Force 2 to 1	Force 2 to 1
		4	Force 1 to 2	Force 1 to 2
		5	Force 3 to 2	Force 3 to 2
		6	Force 2 to 3	Force 2 to 3
		7	Force 4 to 3	Force 4 to 3
		8	Force 3 to 4	Force 3 to 4
		9	Force 1 to 4	Force 1 to 4
		10	Kick Force on 1	Kick Force on 1
		11	Shear 12	Shear 12
		12	Kick Force on 2	Kick Force on 2
		13	Shear 23	Shear 23
		14	Kick Force on 3	Kick Force on 3
		15	Shear 34	Shear 34
		16	Kick Force on 4	Kick Force on 4
		17	Shear 41	Shear 41
11	CELAS1/2	2	Force	Force
33	CQUAD4	2	Membrane-x	Membrane-x
		3	Membrane-y	Membrane-y
		4	Membrane-xy	Membrane-xy
		5	Bending-x	Bending-x
		6	Bending-y	Bending-y
		7	Bending-xy	Bending-xy
		8	0.0	Transverse shear-x
		9	0.0	Transverse shear-y

Contents of Record 2 for Forces

34	CBAR	2	Bending moment in plane 1 at end A	Bending moment A1
		3	Bending moment in plane 2 at end A	Bending moment A2
		4	Bending moment in plane 1 at end B	Bending moment B1
		5	Bending moment in plane 2 at end B	Bending moment B2
		6	Shear force in plane 1 at end A	Shear 1
		7	Shear force in plane 2 at end A	Shear 2
		8	Axial force at end A	Axial force
		9	Torque at end A	Torque
64	CQUAD8	2	’CEN/’	’CEN/’
		3	Grid ID or no. of nodes for center	Grid ID or no. of nodes for center
		4	Membrane-x	Membrane-x
		5	Membrane-y	Membrane-y
		6	Membrane-xy	Membrane-xy
		7	Bending-x	Bending-x
		8	Bending-y	Bending-y
		9	Bending-xy	Bending-xy
		10	0.0	Transverse shear-x
		11	0.0	Transverse shear-y
		12-47	Repeat items 3-11 for four corner grids Center force values are used at corners	
74	CTRIA3	Same as CQUAD4		
75	CTRIA6	Same as CQUAD8 except that items 3-11 are repeated for three corner grids for a total of 38 words		

7

Contents of Record 2 for Forces

95	CQUAD4 (Composite)	2-3	Theory	Theory
		4	Lamina Number	Lamina Number
		5	FP	FP
		6	Failure Mode	Failure Mode
		7	FB or -1	FB or -1
		8	MAX of FP, FB or -1	MAX of FP, FB or -1
		9	Failure Flag	Failure Flag
96	CQUAD8 (Composite)	Same as CQUAD4 (Composite)		
97	CTRIA3 (Composite)	Same as CQUAD4 (Composite)		
98	CTRIA6 (Composite)	Same as CQUAD4 (Composite)		
102	CBUSH	2	Translation - x	Translation - x
		3	Translation - y	Translation - y
		4	Translation - z	Translation - z
		5	Rotation - x	Rotation - x
		6	Rotation - y	Rotation - y
		7	Rotation - z	Rotation - z
117	CWELD (ELEMID, GRIDID)	2	mz bending end A plane 1	mz bending end A plane 1
		3	my bending end A plane 2	my bending end A plane 2
		4	mz bending end B plane 1	mz bending end B plane 1
		5	my bending end B plane 2	my bending end B plane 2
		6	fy shear force plane 1	fy shear force plane 1
		7	fz shear force plane 2	fz shear force plane 2
		8	fx axial force	fx axial force
		9	mx torque	mx torque
118	CWELD (ELPAT, PARTPAT)	Same as CWELD (ELEMID, GRIDID)		
200	CWELD (ALIGN)	Same as CWELD (ELEMID, GRIDID)		

7

Contents of Record 2 for Complex Stresses and Strains

ELEMENT		WORD	*GENESIS* ITEM	NASTRAN ITEM	Real/Mag Imag/phase
TYPE	NAME				
1	CROD	2	Axial stress	Axial stress	RM
		3	Axial stress	Axial stress	IP
		4	0.0	Torsional stress	RM
		5	0.0	Torsional stress	IP
2	CBEAM	2	Grid ID	Grid ID	
		3	X/XB	X/XB	
		4	Bending + Axial at C	Longitudinal Stress at C	RM
		5	Bending + Axial at D	Longitudinal Stress at D	RM
		6	Bending + Axial at E	Longitudinal Stress at E	RM
		7	Bending + Axial at F	Longitudinal Stress at F	RM
		8	Bending + Axial at C	Longitudinal Stress at C	IP
		9	Bending + Axial at D	Longitudinal Stress at D	IP
		10	Bending + Axial at E	Longitudinal Stress at E	IP
		11	Bending + Axial at F	Longitudinal Stress at F	IP
		12-111	Repeat items 2-11 for each station. All zeroes are used for non-existent stations		
4	CSHEAR	2	Maximum Shear	Maximum Shear	RM
		3	Maximum Shear	Maximum Shear	IP
		4	Average Shear	Average Shear	RM
		5	Average Shear	Average Shear	IP
11	CELAS1	2	Stress	Stress	RM
		3	Stress	Stress	IP

Contents of Record 2 for Complex Stresses and Strains

33	CQUAD4	2	0.0	Z1 fiber distance 1	
		3	Normal-x at Z1	Normal-x at Z1	RM
		4	Normal-x at Z1	Normal-x at Z1	IP
		5	Normal-y at Z1	Normal-y at Z1	RM
		6	Normal-y at Z1	Normal-y at Z1	IP
		7	Shear-xy at Z1	Shear-xy at Z1	RM
		8	Shear-xy at Z1	Shear-xy at Z1	IP
		9	0.0	Z2 fiber distance 2	
		10	Normal-x at Z2	Normal-x at Z2	RM
		11	Normal-x at Z2	Normal-x at Z2	IP
		12	Normal-y at Z2	Normal-y at Z2	RM
		13	Normal-y at Z2	Normal-y at Z2	IP
		14	Shear-xy at Z2	Shear-xy at Z2	RM
		15	Shear-xy at Z2	Shear-xy at Z2	IP

Contents of Record 2 for Complex Stresses and Strains

34	CBAR	2	Bending + axial at point C - end A	SA1	RM
		3	Bending + axial at point D - end A	SA2	RM
		4	Bending + axial at point E - end A	SA3	RM
		5	Bending + axial at point F - end A	SA4	RM
		6	Average of bending + axial at points C, D, E and F at end A	Axial	RM
		7	Bending + axial at point C - end A	SA1	IP
		8	Bending + axial at point D - end A	SA2	IP
		9	Bending + axial at point E- end A	SA3	IP
		10	Bending + axial at point F - end A	SA4	IP
		11	Average of bending + axial at points C, D, E and F at end A	Axial	IP
		12	Bending + axial at point C - end B	SB1	RM
		13	Bending + axial at point D - end B	SB2	RM
		14	Bending + axial at point E - end B	SB3	RM
		15	Bending + axial at point F - end B	SB4	RM
		16	Bending + axial at point C - end B	SB1	IP
		17	Bending + axial at point D - end B	SB2	IP
		18	Bending + axial at point E - end B	SB3	IP
		19	Bending + axial at point F - end B	SB4	IP

Contents of Record 2 for Complex Stresses and Strains

39	CTETRA	2	0	Stress coordinate system	
		3	'GRID'	Coordinate type	
		4	4	Number of active points	
		5	Grid ID or 0 for center	Grid ID or 0 for center	
		6	Normal-x	Normal-x	RM
		7	Normal-y	Normal-y	RM
		8	Normal-z	Normal-z	RM
		9	Shear-xy	Shear-xy	RM
		10	Shear-yz	Shear-yz	RM
		11	Shear-zx	Shear-zx	RM
		12	Normal-x	Normal-x	IP
		13	Normal-y	Normal-y	IP
		14	Normal-z	Normal-z	IP
		15	Shear-xy	Shear-xy	IP
		16	Shear-yz	Shear-yz	IP
		17	Shear-zx	Shear-zx	IP
		18-69	Items 5 through 17 repeated for 4 corners	Items 5 through 17 repeated for 4 corners	

Contents of Record 2 for Complex Stresses and Strains

64	CQUAD8	2	'CEN/'	'CEN/'	
		3	Grid ID or no. of nodes for center	Grid ID or no. of nodes for center	
		4	0.0	Z1 fiber distance 1	
		5	Normal-x at Z1	Normal-x at Z1	RM
		6	Normal-x at Z1	Normal-x at Z1	IP
		7	Normal-y at Z1	Normal-y at Z1	RM
		8	Normal-y at Z1	Normal-y at Z1	IP
		9	Shear-xy at Z1	Shear-xy at Z1	RM
		10	Shear-xy at Z1	Shear-xy at Z1	IP
		11	0.0	Z2 fiber distance 2	
		12	Normal-x at Z2	Normal-x at Z2	RM
		13	Normal-x at Z2	Normal-x at Z2	IP
		14	Normal-y at Z2	Normal-y at Z2	RM
		15	Normal-y at Z2	Normal-y at Z2	IP
		16	Shear-xy at Z2	Shear-xy at Z2	RM
		17	Shear-xy at Z2	Shear-xy at Z2	IP
		18-75	Repeat items 3-17 for four corner grids Center stress values are used at corners		

7

Contents of Record 2 for Complex Stresses and Strains

66	CHEX20	2	0	0	
		3	Normal-x	Normal-x	RM
		4	Normal-y	Normal-y	RM
		5	Normal-z	Normal-z	RM
		6	Shear-xy	Shear-xy	RM
		7	Shear-yz	Shear-yz	RM
		8	Shear-zx	Shear-zx	RM
		9	Normal-x	Normal-x	IP
		10	Normal-y	Normal-y	IP
		11	Normal-z	Normal-z	IP
		12	Shear-xy	Shear-xy	IP
		13	Shear-yz	Shear-yz	IP
		14	Shear-zx	Shear-zx	IP
67	CHEXA (8 Nodes)	Same as CTETRA except that items 5-17 are repeated for eight corners for a total of 121 words.			
68	CPENTA	Same as CTETRA except that items 5-17 are repeated for six corners for a total of 95 words			
255	CPYRA	Same as CTETRA except that items 5-17 are repeated for five corners for a total of 82 words			
74	CTRIA3	Same as CQUAD4			
75	CTRIA6	Same as CQUAD8 except that items 3-17 are repeated for three corner grids for a total of 62 words			
95	CQUAD4 (Composite)	Undefined			
96	CQUAD8 (Composite)	Undefined			
97	CTRIA3 (Composite)	Undefined			
98	CTRIA6 (Composite)	Undefined			

7

Contents of Record 2 for Complex Stresses and Strains

102	CBUSH	2	Translation - x	Translation - x	RM
		3	Translation - y	Translation - y	RM
		4	Translation - z	Translation - z	RM
		5	Rotation - x	Rotation - x	RM
		6	Rotation - y	Rotation - y	RM
		7	Rotation - z	Rotation - z	RM
		8	Translation - x	Translation - x	IP
		9	Translation - y	Translation - y	IP
		10	Translation - z	Translation - z	IP
		11	Rotation - x	Rotation - x	IP
		12	Rotation - y	Rotation - y	IP
		13	Rotation - z	Rotation - z	IP
117	CWELD (ELEMID, GRIDID) (no strains)	2	Axial	Axial	RM
		3	SA maximum	SA maximum	RM
		4	SA minimum	SA minimum	RM
		5	SB maximum	SB maximum	RM
		6	SB minimum	SB minimum	RM
		7	Maximum Shear	Maximum Shear	RM
		8	Bearing	Bearing	RM
		9	Axial	Axial	IP
		10	SA maximum	SA maximum	IP
		11	SA minimum	SA minimum	IP
		12	SB maximum	SB maximum	IP
		13	SB minimum	SB minimum	IP
		14	Maximum shear	Maximum shear	IP
		15	Bearing	Bearing	IP
118	CWELD (ELPAT, PARTPAT)	Same as CWELD (ELEMID, GRIDID)			
200	CWELD (ALIGN)	Same as CWELD (ELEMID, GRIDID)			

Contents of Record 2 for Complex Forces

1	CROD	2	Axial force	Axial force	RM
		3	Axial force	Axial force	IP
		4	0.0	Torque	RM
		5	0.0	Torque	IP
2	CBEAM	2	Grid ID	Grid ID	
		3	X/XB	X/XB	
		4	Bending moment Plane 1	Bending moment Plane 1	RM
		5	Bending moment Plane 2	Bending moment Plane 2	RM
		6	Shear force Plane 1	Shear force Plane 1	RM
		7	Shear force Plane 2	Shear force Plane 2	RM
		8	Axial force	Axial force	RM
		9	Torque	Total Torque	RM
		10	Bimoment	Warping torque	RM
		11	Bending moment Plane 1	Bending moment Plane 1	IP
		12	Bending moment Plane 2	Bending moment Plane 2	IP
		13	Shear force Plane 1	Shear force Plane 1	IP
		14	Shear force Plane 2	Shear force Plane 2	IP
		15	Axial force	Axial force	IP
		16	Torque	Total Torque	IP
		17	Bimoment	Warping torque	IP
		18-177	Repeat items 2-17 for each station. Zeroes are used for non-existent stations.		

Contents of Record 2 for Complex Forces

4	CSHEAR	2	Force 4 to 1	Force 4 to 1	RM
		3	Force 2 to 1	Force 2 to 1	RM
		4	Force 1 to 2	Force 1 to 2	RM
		5	Force 3 to 2	Force 3 to 2	RM
		6	Force 2 to 3	Force 2 to 3	RM
		7	Force 4 to 3	Force 4 to 3	RM
		8	Force 3 to 4	Force 3 to 4	RM
		9	Force 1 to 4	Force 1 to 4	RM
		10	Force 4 to 1	Force 4 to 1	IP
		11	Force 2 to 1	Force 2 to 1	IP
		12	Force 1 to 2	Force 1 to 2	IP
		13	Force 3 to 2	Force 3 to 2	IP
		14	Force 2 to 3	Force 2 to 3	IP
		15	Force 4 to 3	Force 4 to 3	IP
		16	Force 3 to 4	Force 3 to 4	IP
		17	Force 1 to 4	Force 1 to 4	IP
		18	Kick Force on 1	Kick Force on 1	RM
		19	Shear 12	Shear 12	RM
		20	Kick Force on 2	Kick Force on 2	RM
		21	Shear 23	Shear 23	RM
		22	Kick Force on 3	Kick Force on 3	RM
		23	Shear 34	Shear 34	RM
		24	Kick Force on 4	Kick Force on 4	RM
		25	Shear 41	Shear 41	RM
		26	Kick Force on 1	Kick Force on 1	IP
		27	Shear 12	Shear 12	IP
		28	Kick Force on 2	Kick Force on 2	IP
		29	Shear 23	Shear 23	IP
		30	Kick Force on 3	Kick Force on 3	IP
		31	Shear 34	Shear 34	IP
		32	Kick Force on 4	Kick Force on 4	IP
		33	Shear 41	Shear 41	IP

Contents of Record 2 for Complex Forces

11	CELAS1/2	2	Force	Force	RM
		3	Force	Force	IP
20	CDAMP1/2	Same as CELAS1/2			
24	CVISC	2	Axial Force	Axial Force	RM
		3	Axial Force	Axial Force	IP
		4	Torque	Torque	RM
		5	Torque	Torque	IP
33	CQUAD4	2	Membrane Force-x	Membrane Force-x	RM
		3	Membrane Force-y	Membrane Force-y	RM
		4	Membrane Force-xy	Membrane Force-xy	RM
		5	Bending Moment-x	Bending Moment-x	RM
		6	Bending Moment-y	Bending Moment-y	RM
		7	Bending Moment-xy	Bending Moment-xy	RM
		8	Shear-x	Shear-x	RM
		9	Shear-y	Shear-y	RM
		10	Membrane Force-x	Membrane Force-x	IP
		11	Membrane Force-y	Membrane Force-y	IP
		12	Membrane Force-xy	Membrane Force-xy	IP
		13	Bending Moment-x	Bending Moment-x	IP
		14	Bending Moment-y	Bending Moment-y	IP
		15	Bending Moment-xy	Bending Moment-xy	IP
		16	Shear-x	Shear-x	IP
		17	Shear-y	Shear-y	IP

Contents of Record 2 for Complex Forces

34	CBAR	2	Bending moment A1	Bending moment A1	RM
		3	Bending moment A2	Bending moment A2	RM
		4	Bending moment B1	Bending moment B1	RM
		5	Bending moment B2	Bending moment B2	RM
		6	Shear 1	Shear 1	RM
		7	Shear 2	Shear 2	RM
		8	Axial force	Axial force	RM
		9	Torque	Torque	RM
		10	Bending moment A1	Bending moment A1	IP
		11	Bending moment A2	Bending moment A2	IP
		12	Bending moment B1	Bending moment B1	IP
		13	Bending moment B2	Bending moment B2	IP
		14	Shear 1	Shear 1	IP
		15	Shear 2	Shear 2	IP
		16	Axial force	Axial force	IP
		17	Torque	Torque	IP

Contents of Record 2 for Complex Forces

64	CQUAD8	2	'CEN/'	'CEN/'	
		3	Grid ID or no. of nodes for center	Grid ID or no. of nodes for center	
		4	Membrane Force-x	Membrane Force-x	RM
		5	Membrane Force-y	Membrane Force-y	RM
		6	Membrane Force-xy	Membrane Force-xy	RM
		7	Bending Moment-x	Bending Moment-x	RM
		8	Bending Moment-y	Bending Moment-y	RM
		9	Bending Moment-xy	Bending Moment-xy	RM
		10	Shear-x	Shear-x	RM
		11	Shear-y	Shear-y	RM
		12	Membrane Force-x	Membrane Force-x	IP
		13	Membrane Force-y	Membrane Force-y	IP
		14	Membrane Force-xy	Membrane Force-xy	IP
		15	Bending Moment-x	Bending Moment-x	IP
		16	Bending Moment-y	Bending Moment-y	IP
		17	Bending Moment-xy	Bending Moment-xy	IP
		18	Shear-x	Shear-x	IP
		19	Shear-y	Shear-y	IP
		20-87	Repeat items 3-19 for four corner grids Center force values are used at corners		
74	CTRIA3	Same as CQUAD4			
75	CTRIA6	Same as CQUAD8 except that items 3-19 are repeated for three corner grids for a total of 70 words			
95	CQUAD4 (Composite)	Undefined			
96	CQUAD8 (Composite)	Undefined			
97	CTRIA3 (Composite)	Undefined			
98	CTRIA6 (Composite)	Undefined			

7

Contents of Record 2 for Complex Forces

102	CBUSH	2	Translation - x	Translation - x	RM
		3	Translation - y	Translation - y	RM
		4	Translation - z	Translation - z	RM
		5	Rotation - x	Rotation - x	RM
		6	Rotation - y	Rotation - y	RM
		7	Rotation - z	Rotation - z	RM
		8	Translation - x	Translation - x	IP
		9	Translation - y	Translation - y	IP
		10	Translation - z	Translation - z	IP
		11	Rotation - x	Rotation - x	IP
		12	Rotation - y	Rotation - y	IP
		13	Rotation - z	Rotation - z	IP

Contents of Record 2 for Complex Forces

117	CWELD (ELEMID, GRIDID)	2	mz bending end A plane 1	mz bending end A plane 1	RM
		3	my bending end A plane 2	my bending end A plane 2	RM
		4	mz bending end B plane 1	mz bending end B plane 1	RM
		5	my bending end B plane 2	my bending end B plane 2	RM
		6	fy shear force plane 1	fy shear force plane 1	RM
		7	fz shear force plane 2	fz shear force plane 2	RM
		8	fx axial force	fx axial force	RM
		9	mx torque	mx torque	RM
		10	mz bending end A plane 1	mz bending end A plane 1	IP
		11	my bending end A plane 2	my bending endA plane 2	IP
		12	mz bending end B plane 1	mz bending end B plane 1	IP
		13	my bending end B plane 2	my bending end B plane 2	IP
		14	fy shear force plane 1	fy shear force plane 1	IP
		15	fz shear force plane 2	fz shear force plane 2	IP
		16	fx axial force	fx axial force	IP
		17	mx torque	mx torque	IP
118	CWELD (ELPAT, PARTPAT)	Same as CWELD (ELEMID, GRIDID)			
200	CWELD (ALIGN)	Same as CWELD (ELEMID, GRIDID)			

7.3.4 NASTRAN PUNCH Format Results Files

Results files that can be read by software that reads NASTRAN PUNCH files can be generated using the command **POST**=PUNCH.

A header is written for each loadcase for each result type. The header write statement formats for displacements (DISPLACEMENTS), velocities (VELOCITY), accelerations (ACCELERATION), reaction forces (SPCF), applied loads (OLOADS), grid point stresses (GPSTRESS) and temperatures (THERMAL) are:

FORMAT(‘$TITLE =’,A62) title

FORMAT(‘$SUBTITLE=’,A62) subtitle

FORMAT(‘$LABEL =’,A62) label

FORMAT(‘$’,A30) result type

FORMAT(‘$REAL OUTPUT’)

FORMAT(‘$SUBCASE ID =’,4X,I8) subcase ID

The header write statement formats for ELEMENT STRESSES, ELEMENT STRAINS, ELEMENT FORCES and ELEMENT STRAIN ENERGY are:

FORMAT(‘$TITLE =’,A62) title

FORMAT(‘$SUBTITLE=’,A62) subtitle

FORMAT(‘$LABEL =’,A62) label

FORMAT(‘$’,A30) result type

FORMAT(‘$REAL OUTPUT’)

FORMAT(‘$SUBCASE ID =’,4X,I8) subcase ID

FORMAT(‘$ELEMENT TYPE =’,4X,I8) element type code

The header write statement formats for mode shapes [EIGENVECTOR (SOLUTION SET)] are:

FORMAT(‘$TITLE =’,A62) title

FORMAT(‘$SUBTITLE=’,A62) subtitle

FORMAT(‘$LABEL =’,A62) label

FORMAT(‘$’,A30) result type

FORMAT(‘$REAL OUTPUT’)

FORMAT(‘$SUBCASE ID =’,4X,I8) subcase ID

FORMAT(‘$EIGENVALUE = ‘E14.7,2X,’MODE =’,I6) eigenvalue, mode number

The title, subtitle, and subcase label are from the solution control data. The result types are: DISPLACEMENTS, VELOCITY, ACCELERATION, SPCF, OLOADS, GPSTRESS, ELEMENT STRESSES, ELEMENT STRAINS, ELEMENT FORCES, EIGENVECTOR (SOLUTION SET), ESE, CONTACT DISPLACEMENTS, CONTACT FORCES and CONTACT PRESSURES. The result type for temperatures is DISPLACEMENT as per the NASTRAN convention. The element types are the same as NASTRAN element types and are:

Element Type	Element
1	CROD
2	CBEAM
4	CSHEAR
11	CELAS1
20	CDAMP1
24	CVISC
33	CQUAD4
34	CBAR
36	CTETRA
53	CTRIAX6
64	CQUAD8
66	CHEX20
67	CHEXA
68	CPENTA
74	CTRIA3
75	CTRIA6
86	CGAP
95	CQUAD4 (Composite)
96	CQUAD8 (Composite)
97	CTRIA3 (Composite)
98	CTRIA6 (Composite)
102	CBUSH

117	CWELD (ELEMID, GRIDID)
118	CWELD (ELPAT, PARTPAT)
200	CWELD (ALIGN)
255	CPYRA

The analysis results are written after each header. For static displacements, reaction forces, applied loads and mode shapes, the grid ID and 6 components are written using the format:

FORMAT(2X,I8,7X,'G',1PE13.6,2(5X,E13.6):/('-CONT-',12X,3(5X,E13.6)))

For temperatures, the grid ID and temperature are written using the format:

FORMAT('TEMP*'18X,'1',8X,I8,3X,1PE13.6)

For element stresses, element strains, and element forces for all elements except HEXA, PENTA, TETRA and PYRA, the element ID and element results are written using the format:

FORMAT(2X,I8,13X,1PE13.6,2(5X,E13.6):/('-CONT-',12X,3(5X,E13.6)))

where the element results are in the same order as shown in the tables for the NASTRAN OUTPUT2 results in the proceeding Subsection. For element stresses and strains for HEXA, PENTA, TETRA and PYRA elements, the centroidal stresses are written using the format:

FORMAT(2X,I8,25X,A1,2(5X,1PE13.6):/('-CONT-',12X,3(5X,E13.6)))

where the first record (line) for each element contains the element ID, 'R', 0.0 and 0.0. After the first line for each element are the 21 stress or strain values described in words 5-25 for the CTETRA element as shown in the tables for the NASTRAN OUTPUT2 results in the preceding subsection.

For dynamic analysis results, the header line $REAL OUTPUT is replaced by either $REAL-IMAGINARY OUTPUT or $MAGNITUDE-PHASE OUTPUT, depending on the solution control command DYNOUTPUT. The dynamic displacements, velocities and accelerations contain the 6 real/magnitude components, followed by the 6 imaginary/phase components. The element results are in the same order shown in the tables for the NASTRAN OUTPUT2 results in the preceding subsection. An additional line is added to the end of the header that contains the loading frequency value in hertz and is written with the format:

FORMAT ($FREQUENCY=;2X,E14.7) frequency

For element strain energy, in static loadcases, the subcase ID in the header will be multiplied by 1000. For element strain energy, in the frequency loadcases, the subcase ID in the header will be multiplied by 1000 and the mode number will be added. This method introduce a limitation on the subcases id to be less than 100000. If a loadcase ID is larger that 99999 a warning message will be issued. The element ID and element stress energies are written using the following format:

FORMAT(2X,I8,13X,1PE13.6,2(5X,E13.6))

To get grid results in the general coordinate system, use the Solution Control Command **POSTOUTPUT** = GENERAL.

7.3.5 IDEAS Format Results Files

Results files that can be read into the I-deas program can be generated using the command **POST**=IDEAS. These are ASCII formatted files.

The blocks of data that are used by I-deas for post processing are called Universal Datasets. The *GENESIS* post processing file ".unv" actually contains many Universal Datasets. One Dataset is written for each result type for each load case (or each dynamic loading frequency).

Analysis results for elements (stress, strain, and force) are written to a separate Dataset for each result type for each load case (or dynamic loading frequency). The Dataset type is 2413. This Dataset contains element results for each grid point of each element. The CROD and CBAR element forces and stresses are output for each end of the element. For the two dimensional and solid elements the centroidal result is used as the grid point result for each grid point of the element. For the plate/shell elements (CQUAD4 and CTRIA3) results for both the bottom and top surfaces are output. The header Dataset type is 2413.

Element forces are only written out for CROD and CBAR elements. There are no element force results for CQUAD4, CTRIA3, CSHEAR, CDAMP1, CVISC and CELAS1 elements.

There are no element stress results for CELAS1 elements or for CQUAD4 and CTRIA3 elements that reference PCOMP data.

To get grid results in the general coordinate system, use the Solution Control Command **POSTOUTPUT** = GENERAL.

Note that, if the SCSID data in the PSHELL data is not equal to -1 (the default), then the force, stress and strain Master series results for elements that reference this PSHELL data will be wrong. If the CORDM data in the PSOLID data is not equal to 0 (the default), then the stress and strain I-deas results for elements that reference this PSOLID data will be wrong.

7.4 Reduced Matrices and Recovery MPC

The reduced matrix file is named *pname*xx.DMIG. The reduced matrix file is created if any loadcase has the **ALOAD** = DMIG, **KAA** = DMIG, **K4AA** = DMIG and/or **MAA** = DMIG solution control commands. The matrices are written using the **DMIG** bulk data format.

Note that if the design cycle is greater than 99, the filename pattern changes so that the design cycle is 4 digits bracketed by '=' characters: *pname*=xxxx=.DMIG.

If running in **REDUCE** mode and the analysis parameter **SEMPC** is not zero and there is a **DISPLACEMENT** or **SVECTOR** output request, then **MPC** entries will be generated for the three translational displacements of each grid in the output set. The MPC entries will be written to the DMIG post file using the value of SEMPC as the MPC set ID. These MPC entries can be used in a subsequent run of the residual model to recover degrees of freedom omitted by the superelement reduction.

7.5 Guyan Reduced Stiffness Matrix

The Guyan reduced stiffness matrix file is named *pname*xxyy.KAA or *pname*xxyyyyyyyy.KAA. Files are created for every Guyan reduction loadcase that has the **KAA** = POST solution control command using the following procedure:

```
FOR Every design cycle:
LOOP over all guyan reduction loadcases with KAA = POST
   OPEN the PNAMEXXYY.KAA file
   WRITE(LUN9) KEY,NEQR,ICYCLE,LOADID
   WRITE(LUN9) (IWORK(1,J),IWORK(2,J),J=1,NEQR)
   WRITE OUT REDUCED MATRIX BY COLUMNS OF LOWER TRIANGLE
   ILAST = 0
   DO 10 J = 1,NEQR
      NROW = NEQR - J + 1
      IFIRST = ILAST + 1
      ILAST = IFIRST + NROW - 1
      WRITE(LUN9) (KMAAL(I),I=IFIRST,ILAST)
10 CONTINUE
   CLOSE the PNAMEXXYY.KAA file
CONTINUE
```

Where,

PNAME is the project name.

XX indicates the design cycle number.

YY indicates the loadcase number, when the loadcase id is 99 or less.

YYYYYYYY indicates the loadcase number, when the loadcase id is 100 or higher.

LUN9 is the unit number use to write the unformatted sequential access file

PNAMEXXYY.KAA or PNAMEXXYYYYYYYY.KAA is the unformatted sequential access file that contains the Guyan reduced stiffness matrix.

KEY number zero.

NEQR is the number of ASET degrees of freedoms.

ICYCLE is the design cycle.

LOADID is the load case number.

IWORK(1,J) contains the grid number (J=1,NEQR).

IWORK(2,J) contains the component number (J=1,NEQR).

KMAAL is a double precision array that contains the guyan reduced stiffness matrix.

If MPRINT=FIRST the same procedure is used except that only the reduced stiffness matrices for design cycle zero are printed.

7.6 Guyan Reduced Mass Matrix

The Guyan reduced mass matrix file is named *pname*xxyy.MAA or *pname*xxyyyyyyyy.MAA. Files are created for every Guyan reduction loadcase that has the **MAA** = POST solution control command using the following procedure:

```
FOR Every design cycle:
LOOP over all guyan reduction loadcases with MAA = POST
   OPEN the PNAMEXXYY.MAA file
   WRITE(LUN9) KEY,NEQR,ICYCLE,LOADID
   WRITE(LUN9) (IWORK(1,J),IWORK(2,J),J=1,NEQR)
   WRITE OUT REDUCED MATRIX BY COLUMNS OF LOWER TRIANGLE
   ILAST = 0
   DO 10 J = 1,NEQR
      NROW = NEQR - J + 1
      IFIRST = ILAST + 1
      ILAST = IFIRST + NROW - 1
      WRITE(LUN9) (KMAAL(I),I=IFIRST,ILAST)
10 CONTINUE
   CLOSE the PNAMEXXYY.MAA file
CONTINUE
```

Where,

PNAME is the project name as define in the ID executive control command.

XX indicates the design cycle number.

YY indicates the loadcase number, when the loadcase id is 99 or less.

YYYYYYYY indicates the loadcase number, when the loadcase id is 100 or higher.

LUN9 is the unit number use to write the unformatted sequential access file.

PNAMEXXYY.MAA or PNAMEXXYYYYYYYY.MAA is the unformatted sequential access file that contains the Guyan reduced mass matrix.

KEY number zero.

NEQR is the number of ASET degrees of freedoms.

ICYCLE is the design cycle.

LOADID is the load case number.

IWORK(1,J) contains the grid number (J=1,NEQR).

IWORK(2,J) contains the component number (J=1,NEQR).

KMAAL is a double precision array that contains the guyan reduced mass matrix.

If MPRINT=FIRST the same procedure is used except that only the reduced stiffness matrices for design cycle zero are printed.

7.7 Eigen Database File

The eigen database file is named *pname*.EIGDB. The eigen database file is created if there are natural frequency loadcases and the analysis parameter **MODERST** is set to WRITE. This file is useful for saving time in subsequent modal frequency response runs. In the subsequent runs, change the parameter MODERST to READ, and instead of performing the eigenanalysis again, the program will simply read the existing eigen database file. This process can only work if the subsequent runs do not change the structural model. The only acceptable modifications are changes to viscous or structural damping. However, the only check that the program can perform is to ensure that the subsequent runs have the same number of degrees of freedom as the initial run. If aspects of the model other than damping are changed, and PARAM,MODERST,READ is used, the imported eigenvalues and eigenvectors will not match the changed model, and therefore the modal frequency response results will not be correct.

7

7.8 Scratch Files

GENESIS uses several scratch files for storage. These files have the project name with a special extension, and after the run is finished, they are deleted. While the contents of these files are unimportant for the user, their size is sometimes important, especially when running large problems where the available disk space is relatively small. To see the size of the scratch files, use DIAG=992.

The user can distribute the scratch files among several disks to allow for more space than is available in a single directory. To define directories for scratch files, use the **DIRALL**, **DIRDAF**, **DIRSAF** and/or **DIRSMS** Executive Control Commands.

APPENDIX A

Diagnostic Information

A.1 Diagnostic Information

This chapter describes the diagnostic information available by including the **DIAG** command in the Executive Control section of the data.

A.2 The DIAG Command

Diagnostic information is available though the use of the DIAG command in the executive control section of *GENESIS*. The DIAG command has the format:

```
DIAG = d1,d2,d3,d4,...
```

where the di are the diagnostic switches. This command is used to tell *GENESIS* to write the contents of internal scalars, tables, and matrices into the output data file. This main reason this option is available is for program debugging. However, certain items may be of value to the user. An abbreviated list of the result of each diagnostic switch is presented here. In most cases each diagnostic switch also turns on all other diagnostic switches with the have a smaller ones digit and the same tens and hundreds digit. For example 145 also turns on 141 through 144.

Following is a list of diagnostic numbers and the information that they produce.

Overall

11 CPU time spent in each module.

12 CPU and elapsed wall clock time spent in each module.

Data Manager

21 Print the data manager dictionary when over write is detected.

23 Check the void tables.

25 Initialize all the data blocks that are created without initialization to a special value; 99999.

28 Check data block unrelease by enforcing every released data block to be stored out of core only.

29 Check read only unrelease by comparing the data block with its out of core version when releasing.

Input/Output

32 Print information about the status of files open with unknown status.

33 Print information about every file’s opening and closing.

36 Print the elemental stiffness and mass matrices.

Program Progress

The following information is directed to the error unit number and can be written to the terminal console, LOG file, or output file.

```
81   Print design cycle number, objective function value
     and maximum constraint violation for each design
     cycle, as well as the convergence flag and completion
     code.
82   Print message after the execution of each program
     module that includes CPU time and at the start of each
     design cycle.
83   Print design cycle history.
84   Print wall clock time spent in each module.
85   Print the problem summary.
87   Print all of the above plus more detailed summaries.
```

Header Format

```
91   Set the date and time to zero in the header.
```

Warning Message Control

```
111  Print all warning messages rather than just the first
     10 of each warning code.
```

Input Data

```
121  Print messages before and after renumbering.
125  Solution control, analysis, and design data blocks
     before   renumbering.
126  Solution control, analysis, and design data blocks
     after renumbering.
```

Static Analysis Data Preprocessing

```
141  Print the dimensions of the analysis problem, which
     includes the total number of equations, the size of
     the [K], maximum column height, root mean square column
     height, number of blocks, and the dimension of each
     block.
142  Print some internal constants that are used in setting
     up the dimensions printed with DIAG=141.
144  Print some of the newly generated data blocks that are
     relatively small.
145  Print all the newly generated data blocks.
148  Print the data blocks that are referenced in this
     module.
```

Heat Transfer Data Preprocessing

171 Print the dimensions of the analysis problem, which includes the total number of equations, the size of the [K], maximum column height, root mean square column height, number of blocks, and the dimension of each block.

172 Print some internal constants that are used in setting up the dimensions printed with DIAG=141.

174 Print some of the newly generated data blocks that are relatively small.

175 Print all the newly generated data blocks.

178 Print the data blocks that are referenced in this module.

Block Equation Solver (Structural Analysis)

301 Print header indicating the beginning and ending of the solution process.

302 Print the boundary condition number before triangularization and the coefficients of the multipoint constraints.

303 Print the total energy, residual energy, and their ratio for each static load case. Print a list of the degrees of freedom that have no mass. (Check the normalization of the eigenvector with respect to the mass matrix.)

304 Print the newly created data blocks. (The displacement vectors in both the global and basic coordinate systems.) (The eigenvalues of each dynamic loadcase.)

305 Print the progress of the assembly and solution process, the displacement vectors in terms of equation numbers, the reaction force vectors, and the eigenvectors in the basic coordinate system. (Check the difference between the diagonal elements of the [K] that are assembled in FE module and that are assembled here in solver.) (Check the accuracy of the eigenvectors of the reduced problem in subspace iteration.)

306 Print the iteration number and the eigenvalues in subspace iteration loop for the eigenvalue solution.

307 Check the accuracy of the eigenvalues and eigenvectors by comparing the eigenvalue with PHI*K*PHI/PHI*M*PHI. Check the residual energy by [K]{u} - {F}. (Note: this requires saving the original untriangularized stiffness matrix.)

308 Print the data blocks that are referenced in this module, and print the residual vectors that are created when checking the residual energy by [K]{u} - {F}.

309 Print the eigenvector changes in the subspace iteration.

Block Equation Solver (Heat Transfer Analysis)

371 Print header indicating the beginning and ending of the solution process.

372 Print the boundary condition number before triangularization and the coefficients of the multipoint constraints.

373 Print the total energy, residual energy, and their ratio for each static load case.

374 Print the newly created data blocks. (The displacement vectors in both the global and basic coordinate systems.

375 Print the progress of the assembly and solution process, the displacement vectors in terms of equation numbers, the reaction force vectors, and the eigenvectors in the basic coordinate system. (Check the difference between the diagonal elements of the [K] that are assembled in FE module and that are assembled here in solver.

377 Check the residual energy by [K]{u} - {F}. (Note: this requires saving the original untriangularized stiffness matrix.)

378 Print the data blocks that are referenced in this module, and print the residual vectors that are created when checking the residual energy by [K]{u} - {F}.

Sparse Matrix Equation Solver (Structural Analysis)

301 Print header indicating the beginning and ending of the solution process.

302 Print the boundary condition number before triangularization and the coefficients of the multipoint constraints.

303 Print the total energy, residual energy, and their ratio for each static load case. Print a list of the degrees of freedom that have no mass.)

304 Print the newly created data blocks. (The displacement vectors in both the global and basic coordinate systems.) (The eigenvalues of each dynamic loadcase.)

305 Print the progress of the assembly and solution process, the displacement vectors in terms of equation numbers, the reaction force vectors, and the eigenvectors in the basic coordinate system. (Check the difference between the diagonal elements of the [K] that are assembled in FE module and that are assembled here in solver.) (Check the accuracy of the eigenvectors of the reduced problem in subspace iteration.

306 Print the iteration number and the eigenvalues in subspace iteration loop for the eigenvalue solution.

308 Print the data blocks that are referenced in this module.

309 Print the eigenvector changes in the subspace iteration.

321 Print the progress of the solution process when DIAGnostics 11, 12, 82, or 84 are also turned on. Print information about the determination of the required working storage.

322 Turn the sparse matrix solver message level to 1 and print more detailed information about the solution process and determination of the required working storage.

324 Turn the sparse matrix solver message level to 2.

326 Turn the sparse matrix solver message level to 3.

328 Turn the sparse matrix solver message level to 4.

Sparse Matrix Equation Solver (Heat Transfer Analysis)

371 Print header indicating the beginning and ending of the solution process.

372 Print the boundary condition number before triangularization.

373 Print the total energy, residual energy, and their ratio for each static load case.

374 Print the newly created data blocks. (The displacement vectors in both the global and basic coordinate systems.

375 Print the progress of the assembly and solution process, the displacement vectors in terms of equation numbers, the reaction force vectors, and the eigenvectors in the basic coordinate system. (Check the difference between the diagonal elements of the [K] that are assembled in FE module and that are assembled here in solver.

378 Print the data blocks that are referenced in this module.

321 Print the progress of the solution process when DIAGnostics 11, 12, 82, or 84 are also turned on. Print information about the determination of the required working storage.

322 Turn the sparse matrix solver message level to 1 and print more detailed information about the solution process and determination of the required working storage.

324 Turn the sparse matrix solver message level to 2.

326 Turn the sparse matrix solver message level to 3.

328 Turn the sparse matrix solver message level to 4.

RBE3

791 Use the old (version 10.0 or earlier) RBE3 formulation (rotation component weights are used as input without any scaling).

Sparse Matrix Ordering Selection

821 Force use of the METIS nested dissection ordering method.

822 Force use of the LSI hybrid ordering method.

823 Force use of the multi-minimum degree ordering method.

824 Force use of the SMOOTH hybrid ordering method.

825 Force use of the SMOOTH nested dissection ordering method.

Mesh Smoothing

901 Use the old (version 10.1 or earlier) mesh smoothing algorithm. Print information about element distortion before and after mesh smoothing.

902 Use the old (version 10.1 or earlier) mesh smoothing algorithm. Correct connectivity of 2D planar meshes so that all elements have normals in the same direction.

903 Use the old (version 10.1 or earlier) mesh smoothing algorithm. Smooth mesh even if less than 3% of the elements are distorted.

904 Same as 902 plus 903.

961 Print statistics about the mesh quality measures during mesh smoothing.

Finish Up

992 Print maximum disk space used by SCRATCH files. Scratch files are the files used by *GENESIS* that are deleted after a successful run.

APPENDIX B

VR&D Client Support

- Product Sales and Support
- VR&D Corporate Profile
- Software Products

B.1 Product Sales and Support

We take great pride in creating the most advanced design optimization products available anywhere. However, we recognize that few people are trained in this technology at the universities. We are strongly committed to assisting you with any questions you may have, whether they relate to the basic theory of analysis and optimization, details of the program input or output, difficulties in interpreting results, presenting the technology or applications to management, or suggestions for improvement, we want to hear from you. We are as close as your phone, facsimile, email or post office.

In addition to our technical staff at VR&D, we have developed a network of associates, distributors and partners to assist you. In some cases, our associates are also new to this technology, and so may have some difficulty with detailed questions. We have asked them to contact us in this case so we can help all concerned. You, as a user, should remember that VR&D engineers at the corporate office are always available to help if your first source is not able to completely answer your questions.

If you need the general purpose optimization softwares DOT or VisualDOC, they may be obtained from us. These software can be coupled with almost any analysis program to perform design optimization studies.

You may reach sales and technical support by calling:

VANDERPLAATS R&D Corporate Headquarters
1767 South 8th Street, Suite 200
Colorado Springs, CO 80905

(719) 473-4611
FAX (719) 473-4638
email sales@vrand.com

VANDERPLAATS R&D Michigan Office
41700 Gardenbrook, Suite 115
Novi, MI 48375

(248) 596-1611
FAX (248) 596-1911
email genesis.support@vrand.com

B.2 VR&D Corporate Profile

Company Profile

VR&D is perhaps the first corporation (other than academic consulting companies) created for the sole purpose of performing research in, and producing commercial software products for design optimization. We were incorporated in 1984 to develop, support and advance the state of the art in design optimization. Our staff is highly trained in all aspects of this advanced technology and the vast majority of our budget is devoted to R&D and user support. Most of our product sales come from referrals. Thus, we are able to concentrate on expanding the state of the art in this exciting technology.

B.3 Software Products

The GENESIS software developed by VR&D is the most advanced structural optimization software available anywhere.

GENESIS is a fully integrated finite element analysis and optimization program which uses DOT or BIGDOT (also from VR&D) as the optimization engine. The analysis is based on the finite element method and is similar to many other commercial analysis programs. The analysis data is similar to NASTRAN data to ease the conversion task. The design capabilities are the latest approximation techniques, many of which were developed by VR&D engineers. GENESIS was developed from the beginning to be a design program, not just an analysis program with design added later.

Our software is fully documented. Additionally, user support is as close as your phone, facsimile machine or email. Finally, we offer both public and in-house short courses and public courses to help you better understand our products and technology.

B.3.1 GENESIS Structural Optimization

GENESIS is a fully integrated structural analysis/design software package, written by leading experts in structural optimization. Analysis is based on the finite element method for static, normal modes, direct and modal frequency analysis, and heat transfer calculations. Design is based on the advanced approximation concepts approach to find an optimum design efficiently and reliably. An approximate problem, generated using analysis and sensitivity information, is used for the actual optimization, which is performed by the well established DOT™ or BIGDOT™ optimizer from VR&D. When the optimum of the approximate problem has been found, a new finite element analysis is performed and the process is repeated until the solution has converged to the true optimum. This process typically requires less than ten detailed finite element analyses, even for large and complex design tasks.

Analysis/Design Features

- No fixed problem size limits
- Blocked profile and sparse matrix equation solvers with automatic bandwidth optimizer
- Subspace iteration and Lanczos eigenvalue solution algorithms
- Design sensitivities calculated analytically in most cases. Sensitivities are calculated semi-analytically by central difference at the element level for some shape sensitivity calculations
- Optimization is performed using the latest approximation methods for maximum efficiency. These methods are coupled with the DOT and BIGDOT numerical optimizers to insure reliability
- Matching analysis results. This provides a method to tune the analysis model to match analysis results with measured or experimental results.
- Structural design variables control the shape, as well as the member dimensions
- No special knowledge of optimization technology is required

Analysis Modeling

GENESIS uses well established finite element technology as the analysis basis for design synthesis. The basic elements, equation solvers and eigensolvers are efficient and reliable. These established methods are embedded in a new program structure to make best use of modern computer technology and to gain maximum optimization efficiency.

- Extensive element library
  - Rod elements
  - Bar elements
  - Beam elements
  - Membrane elements
  - Plate elements
  - Composite elements
  - Shear panel elements
  - Axisymmetric elements
  - Solid elements
  - Scalar spring elements
  - Vector spring elements
  - Mass elements
  - Damping elements
  - Rigid elements
  - Interpolation elements
  - General element (user supplied element)
- Multiple loading conditions
  - Point, pressure, thermal, gravity, centrifugal and cyclic loads
- Frequency calculations
- Multiple boundary conditions
- Multiple materials
  - Isotropic
  - Orthotropic
  - General anisotropic
  - Layered composites
- Rectangular, cylindrical and spherical coordinate systems
- Single and multipoint constraints
- Analysis Options
  - Linear static analysis
  - Inertia relief analysis
  - Dynamic normal mode analysis
  - Direct dynamic response analysis
  - Modal dynamic response analysis
  - Buckling analysis

- Heat transfer analysis. Results of heat transfer analysis can be used automatically as a static thermal load.

Design

GENESIS was written from the beginning as a design optimization program, not just an analysis program with optimization added. The analysis and optimization are fully integrated to provide a high degree of efficiency and reliability. The optimization uses the latest approximation techniques. These methods utilize intermediate variables and responses to create a very high quality approximation to the original finite element analysis. Sensitivity analysis is fully integrated and constraint deletion techniques are used to insure that sensitivity calculations are limited to those necessary for optimization. The approximate problem is generated using the analysis and sensitivity information. This approximate analysis is very fast and is used for the actual optimization, which is performed by the well established DOT™ and BIGDOT™ optimizers from Vanderplaats R&D. When the approximate optimum has been found, a new finite element analysis is performed and the process is repeated until the solution has converged to the true optimum. This repetitive process typically requires less than ten detailed finite element analyses, even if the analysis model is large and complex and there are large numbers of design variables.

Shape and Sizing Design Capabilities

GENESIS provides extensive design capabilities. A brief description of these is;

- Simultaneous design of member dimensions and grid locations.
- Design is the default program control. Analysis only, and sensitivity analysis only options are available with a single control statement.
- Member dimensions (not just section properties)
- Grid coordinates
- Linear design variable linking.
- Nonlinear design variable/property (Synthetic) relationships.
  Synthetic relationships are created by providing a FORTRAN like equation in the input data which relates the property to the design variables.
- Direct and Synthetic responses.
- Matching measured data.
- Mode tracking.
- Automatic basis vector generation.

A wide range of objective functions are available for minimization or maximization.

- Mass
- Volume
- Area
- Length
- Distance
- Angle
- Inertia
- Stress
- Strain
- Force
- Displacement
- Velocity
- Acceleration
- Random root mean square displacement
- Random root mean square Velocity
- Random root mean square Acceleration
- Frequency
- Buckling load factor
- Strain energy
- Temperature
- Synthetic (User defined) responses
- Synthetic responses are created by providing a FORTRAN like equation in the input data which relates the response to the design variables, grid locations and direct responses.

Many responses can be simultaneously constrained to be within specified bounds

- Mass
- Volume
- Area
- Length
- Distance
- Angle
- Inertia
- Stress
- Strain
- Force
- Displacement
- Velocity
- Acceleration
- Random root mean square displacement
- Random root mean square Velocity
- Random root mean square Acceleration

- Frequency
- Buckling load factor
- Strain energy
- Temperature
- Synthetic (User defined) responses
- Basis designs to provide smooth shapes.
  Basis designs are created by generating different shapes for the same FEA model. GENESIS then finds the optimum combination of these shapes.

A beam element library of common cross sections is available. All design variable to property relationships are created automatically.

- Square
- Rectangle
- Circle
- Tube
- Spar
- Three dimension box
- Four dimension box
- I Beam
- Rail
- Tee
- Angle

A plate element library of common cross sections is available. All design variable to property relationships are created automatically.

- Solid Plate
- Sandwich Plate
- Two Thickness Sandwich Plate
- User supplied beam/plate shapes. Users routines are linked with GENESIS.
- User supplied responses and sensitivities for use in optimization. Users routines and analysis programs can be linked with GENESIS.

With GENESIS, an optimum design is found for a cost less than just finding an acceptable design by traditional “cut and try” methods.

Topology Design Capabilities

- Automatic generation of design variables
- Enforce symmetry
- Direct responses
- Compliance index objective function
- Mode tracking

Response available for minimization or maximization

- Mass fraction

- Displacement
- Velocity
- Acceleration
- Random root mean square displacement
- Random root mean square Velocity
- Random root mean square Acceleration
- Strain energy
- Frequency

Many responses can be simultaneously constrained to be within specified bounds

- Mass fraction
- Displacements
- Velocity
- Acceleration
- Random root mean square displacement
- Random root mean square Velocity
- Random root mean square Acceleration
- Strain energy
- Frequency

User Interfaces

- Analysis data is very similar to NASTRAN™ Bulk Data.

  This greatly simplifies conversion of data from NASTRAN to GENESIS.
- Design Studio for Genesis is a design interface developed specially for Genesis by VR&D.
- Available output.

  Line printer

  Design history summary file

  Analysis model history file

  Plot files (binary or ASCII)

  NASTRAN OUTPUT2 format file

  PATRAN neutral file format

  NASTRAN PUNCH file format

  IDEAS Universal Dataset format

  Sensitivity Output file

  Guyan reduced stiffness, mass and their sensitivities

  Basis vectors

  Topology Isodensity surfaces

  Solid Representation of shell models

APPENDIX C

Notices

C.1 Third-Party Open-Source Software

The *GENESIS* software may include components derived from third-party open-source software. The following acknowledgments are given to satisfy licensing requirements of third-party software copyright holders.

Lua

ZLib

Minizip

Libxlsxwriter

Index

GENESIS

Analysis Manual

A

B

C

D

E

F

G

H

I

K

L

M

N

O

P

Q

R

S

T

U

V