Finite Element Modelling Techniques in MSC.NASTRAN and LS/DYNA

JULY 2011

MAVERICK UNITED CONSULTING ENGINEERS

TABLE OF CONTENTS

ACKNOWLEDGEMENTS

My humble gratitude to the Almighty, to Whom this and all work is dedicated.

A special thank you also to my teachers at Imperial College of Science, Technology and Medicine, London and my fellow engineering colleagues at Ove Arup and Partners London and Ramboll Whitbybird London.

Maverick United Consulting Engineers

1 FINITE ELEMENT MODELLING TECHNIQUES AND MSC.NASTRAN CARDS COMMON TO ALL ANALYSES

NASTRAN was originally developed by NASA for the space program at the end of the 1960' s. It is an extremely sophisticated and complicated general-purpose finite element analysis program.

1.1 Input Deck .dat Format

A NASTRAN input file is a .dat file. There are 3 input formats. In the small-field fixed format, each card of 10 fields has fields of 8 characters each. In the large-field fixed format, each card of 6 fields has fields of 16 characters except the first and the last which is still 8 characters. The large-field format is denoted by having an asterisk (*) immediately following the card name in the first field. The fields in the free-field format are separated by commas and their accuracy is similar to the small-field fixed format. Hence, real numbers greater than 8 characters will lose precision when in the free-field format. The Bulk Data Section can utilize any format, whilst the other sections utilize the free-field format. $ in the very first column denotes a comment line. The entries for the fixed format cards may be anywhere within the Field except for entries in Field 1 and 10 which must be left-justified. The first field gives the card name whilst the last field is reserved as a continuation field. It is important to ensure that the right data type (between Integer, Real or Character) is entered; Integers cannot contain a decimal point, Reals must contain a decimal point and Characters can be alphanumeric although must start with an alpha character and must be equal or less than 8 characters. NASTRAN will not proceed with the analysis if the wrong type of data is detected. Continuation cards are best automatically generated by leaving the last field of the initial card and first field of the next card blank. Automatic continuation cards are not generated for the large-field format, hence must be entered manually. Never leave blank lines as they will just be ignored. Hence, if a deck consists of 3 cards and the second line is left blank, the third card will be taken as the second card. The format of the input deck is shown as follows.

All elements in the model should have unique element IDs, even elements of different types. The formulation of the element stiffness matrix is independent of how the element grid points are numbered. However, each element will have an element coordinate system which is defined by how the element grid points are numbered.

$ EXECUTIVE CONTROL SECTION
ID <8 character string>,<any length character string> TIME <max execution time in CPU minutes>,<max I/O time in seconds> CEND
$ CASE CONTROL SECTION
$ TITLE = < TITLE OF JOB > SUBTITLE = < SUBTITLE OF JOB > LABEL = < LABEL OF JOB > $ …… $
$ BULK DATA
BEGIN BULK $ …… $ ENDDATA

The .f06 file should be checked for `USER FATAL` error messages, which crashes the analysis and `USER WARNING` error messages, which affects accuracy but does not crash the analysis. Also `USER INFORMATION` messages

should be read. The .f04 file contains all data associated with the execution of the solution sequence along with system parameters such as memory and disk usage. It also lists the allocation of files. Generally the most useful information in the .f04 file is the total scratch database size. This should be recorded and used when subsequently running models of a similar size. Search for the string `MAXIMUM DISK USAGE`. The .acct file is the job accounting file and contains useful data such as elapsed time, cpu time, maximum memory used etc. This data can be useful in planning other, similar jobs in future. If the job is very large then the I/O performance can be assessed by looking at the ratio of cpu to elapsed time.

Displacement vector sets in NASTRAN define the equations of motion and are partitioned in the following ways.

(i) g-set is the unconstrained set of structural equations

(ii) the g-set is partitioned into the m-set pertaining to the dependent set of DOFs and the n-set pertaining to the independent set of DOFs

g-set – m-set = n-set

(iii) the n-set is then partitioned into s-set pertaining to the constrained DOFs and the f-set pertaining to the unconstrained DOFs

n-set – s-set = f-set

(iv) the f-set is partitioned into the o-set pertaining to the DOFs eliminated by static condensation and the a-set pertaining to the a-set (analysis set) pertaining to the DOFs not eliminated by static condensation

f-set – o-set = a-set

(v) the a-set is partitioned into the q-set pertaining to the generalized coordinates in Generalized Dynamic Reduction (GDR) or component mode synthesis and the t-set pertaining to the physical DOFs

a-set – q-set = t-set

(vi) the t-set is partitioned into the r-set pertaining to the SUPORT DOFs and the l-set pertaining to DOFs not placed in a SUPORT entry

t-set – r-set = l-set

(vii) the l-set is partitioned into the c-set pertaining to the free boundary set for component mode synthesis or dynamic reduction and the b-set pertaining to the coordinates fixed for component mode analysis or dynamic reduction

l-set – c-set = b-set

Some additional sets include

(i) the e-set representing the extra dynamic DOFs from control systems and other nonstructural physical variables

(ii) the sa-set representing the permanently constrained aerodynamic points in aeroelastic analysis

(iii) the k-set representing the aerodynamic points in aerodynamic analysis

Hence, some combined sets include

(i) the d-set which is a combination of the a-set and the e-set

a-set + e-set = d-set

(ii) the fe-set which is a combination of the f-set and the e-set

f-set + e-set = fe-set

(iii) the ne-set which is a combination of the n-set and the e-set

n-set + e-set = ne-set

(iv) the p-set which is a combination of the g-set and the e-set

g-set + e-set = p-set

(v) the ps-set which is a combination of the p-set and the sa-set

p-set + sa-set = ps-set

(vi) the pa-set which is a combination of the ps-set and the k-set

ps-set + k-set = pa-set

The output can be printed (into ASCII .f06), punched (into ASCII .pch) or plotted (into binary .op2). The ASCII output can also be outputted in a XY format for simplicity if desired. To print into the ASCII .f06, punch into the ASCII .pch or plot into the binary .op2, the relevant output commands are specified, depicted for example as follows.

```
$ CASE CONTROL SECTION
DISPLACEMENT (<SORT1/SORT2>, <PRINT,PUNCH,PLOT>, <REAL/PHASE>) = ALL/<Grid Set ID>
```

Further, to request binary plotted output, the format of the .op2 file must be specified.

```
PARAM, POST, -2
```

The output coordinate system can be in the basic coordinate system or the global coordinate system.

```
PARAM, OUGCORD, BASIC
```

To request simple XY output to be either printed or punched, the following is specified. Note that within the XYPLOT section, the format of the output (i.e. whether Real and Imaginary or Magnitude and Phase) where relevant is determined by that which is specified in the general output request, here DISP. The output set of the general output request DISP does not need to reference the grids whose component outputs are being plotted out in the XYPLOT, thus only the format is taken. Note however that the general output request must specify PLOT. If only the XYPLOT is desired and not the binary .op2 file, the PARAM, POST should just not be specified. Failure to specify the general output request would result in the default format for the XYPLOT. Also note that the XYPLOT section must be the very last request in the Case Control Section.

```
$ CASE CONTROL SECTION
DISPLACEMENT (<PLOT>,<REAL/PHASE>) = ALL/<Grid Set ID>
$ XY plot output
OUTPUT(XYPLOT)
XYPRINT DISP RESPONSE <subcase>/<Grid ID>(<T1/T2/T3><RM/IP>)
XYPUNCH DISP RESPONSE <subcase>/<Grid ID>(<T1/T2/T3><RM/IP>)
```

A useful output option is **stress sorting**. Stress sorting is useful to limit the amount of output to say stresses in elements that only exceed a certain stress criteria. Stress sorting can be performed in MSC.NASTRAN for **SOL 101**, **SOL 109** and **SOL 112** using a few parameters. To enable stress sorting

```
PARAM, S1, 1 $ Enables stress sorting
```

To limit the printed and plotted stresses to include only those in the highest 400 say elements

```
PARAM, NUMOUT, 400 say
```

If NUMOUT, 0 is specified, then stresses in all element greater than BIGER say 200000 Pa is printed and plotted.

```
PARAM, BIGER, 200000 say
```

PARAM, SRTOPT controls the sorting method based on maximum or minimum magnitude or maximum or minimum algebraic value. The default is to sort based on maximum magnitude. PARAM, SRTELTYP specifies the element type, default being all elements. Stress sorting is based upon different stress parameters for different element types. For instance the default stress quantity for CQUAD4 is von Mises on the Z2 surface. The default stress parameters may be changed using the DTI, INDTA bulk data entry.

1.2 Grid Cards

$ BULK DATA									
GRID	**ID**	**CP**	**X1**	**X2**	**X3**	**CD**	PS	SEID	

CP is the spatial coordinate system (basic coordinate system (0) or a local user-defined coordinate system)
CD is the DOF, output, constraints and solution vector coordinate system (basic coordinate system (0) or a local user-defined coordinate system)

There are three coordinate systems in NASTRAN, namely: -

(a) the basic coordinate system, i.e. an implicitly defined rectangular default coordinate system
(b) local user–defined coordinate systems which can be rectangular, cylindrical or spherical
(c) element coordinate systems

The global coordinate system in NASTRAN is defined as all the coordinate systems defined in the CD field. Hence the global coordinate system, which is the system in which the structural matrices are assembled, is really a collection of basic and local coordinate systems. Classical finite element books state that element stiffness matrices are generated in the element coordinate system and then transformed into a unique coordinate system, which they call the global coordinate system, in NASTRAN the basic coordinate system. NASTRAN on the other hand, generates the element stiffness matrices in the element coordinate system, then instead of transforming that into the unique basic coordinate system, these matrices are transformed into the coordinate system specified on the CD field for the grid points where the element is attached, hence called the global coordinate system. Hence if the CD field of all the grid entries are 0 or left blank, then the global coordinate system will be equal to the basic coordinate system.

The user defined local coordinate systems include: -

Rectangular CORD1R defined by 3 grids
Cylindrical CORD1C defined by 3 grids
Spherical CORD1S defined by 3 grids
Rectangular CORD2R defined by 3 points defined by an independent coordinate system
Cylindrical CORD2C defined by 3 points defined by an independent coordinate system
Spherical CORD2S defined by 3 points defined by an independent coordinate system

If the location of the grids changes then obviously the definitions of the CORD1R, CORD1C and CORD1S local systems also change.

If a grid point is not attached to the structure in any way, there will be not stiffness contribution to it. It may be that it is required to define a local coordinate system or an element orientation vector. In this case, the grid point should be fully constrained, either manually with SPC cards or automatically with AUTOSPC.

1.3 Finite Element Modelling Techniques

1.3.1 Nature of the Finite Element Method

Shape functions must for convergence in the limit: -

(i) permit rigid body motion of the element i.e. zero strain or stress condition (satisfied by having a constant term in the interpolation function)

(ii) have its strains at least defined (constant strain or stress condition), although not necessarily continuous between elements. A constant stress condition is necessary for convergence since, in the limit, if an element is made infinitesimally small, it will then correctly represent the stress at a point, which is constant. For Euler-Bernoulli bending, a continuity of $C^{h-1} = C^{2-1} = C^1$ (where h is the highest derivative order of the displacement in the governing differential equation) is required. For Timoshenko bending, C^0 is sufficient. For plane stress, C^0 is sufficient. C^0 implies that inter-element continuity of displacement is required but not any of its derivatives, hence strains and stresses are not continuous between elements.

It has been said that as a mesh of displacement based compatible finite elements is refined, the unknown nodal displacement estimates will converge monotonically. If the form of loading is by point forces or surface pressures, then the finite element solution will converge from a stiffer response to the exact solution in the limit. Conversely, if the loading is in terms of applied displacement or strains (such as support movements or thermal loads), then the finite element response will converge from a more flexible response to the exact solution in the limit.

The exact solution of the differential equation satisfies equilibrium and compatibility everywhere. The nature of the finite element solution on the other hand is as follows.

(i) Compatibility satisfies at nodes naturally when the elements are connected at common nodes

(ii) Equilibrium satisfied at nodes as the equation of static or dynamic equilibrium requires; that for a linear static analysis is $\{P\} = [K] \{U\}$

(iii) Compatibility satisfied within the elements because of the continuous polynomial interpolation, unless under-integration of the element stiffness matrix is made.

(iv) Compatibility should be satisfied across inter-element boundaries barring poor user meshing. For compatibility, the displacement function must be continuous across element boundaries, i.e. at least C^0 continuity at these boundaries. Inter-element C^0 continuity is satisfied for standard TRIA3, TRIA6, QUAD4, QUAD8, HEX8 and HEX20 elements. Compatibility is not satisfied when a QUAD4 is connected to the mid-side nodes of a QUAD8 or a TRIA3 to the mid-side nodes of a QUAD8 and other poorly formed connections or mesh transitions. Note however that QUAD8 connected to a TRIA6 will ensure compatibility. A good mesh transition for refinement is simply to use QUADs (instead of TRIAs) throughout the transition.

(v) Equilibrium is usually NOT satisfied within the element

(vi) Equilibrium is NOT satisfied at or across element boundaries of C^0 elements but is across C_1 elements.

With mesh refinement, the results will converge to the true solution for displacements and stresses. Stress analysis requires a much finer mesh density than static displacement response. This is because of the rate of convergence of displacement response being much greater than that of the stress.

1.3.2 Finite Element Displacement Interpolation Function, Output Stress Variation, and Corresponding Order of Error in Displacement and Stress

Discretization of the continuum structure with appropriate finite elements is crucial. Interpolation function or the shape functions of an element define the variation of displacement within the element. It is of paramount importance to understand the nature of deformations that can be modeled by certain finite elements.

For a simple beam element, the axial deformation differential equation is

$$F = AE\frac{du}{dx} \qquad \textbf{(Engineers Axial Theory } \sigma = F/A = Edu/dx\textbf{)}$$

The bending deformation differential equation is

$$EI\frac{d^2u}{dx^2} = M \qquad \textbf{(Engineers Bending Theory } \sigma/y = M/I = E/R = Ed^2u/dx^2\textbf{)}$$

The shear deformation differential equation is

$$\frac{du}{dx} = \frac{V}{GA_s} \qquad \textbf{(Engineers Shear Theory } \tau = VQ/(Ib) \textbf{ or on average } V/A_s = Gdu/dx\textbf{)}$$

The torsion deformation differential equation is

$$GJ\frac{d\phi}{dx} = T \qquad \textbf{(Engineers Torsion Theory } \tau/r = T/J = Gd\phi/dx\textbf{)}$$

These are also epitomized in the **virtual work** (unit load method) expression to find deflections of systems.

$$\begin{aligned}
\Delta_1 &= \text{external real kinematic due to unit external real action} \\
&= f_1'\delta_1 \\
&= \int (M_1)_V\left(\frac{M_1}{EI}ds\right)_R + \int (P_1)_V\left(\frac{P_1}{EA}ds\right)_R \text{ or } (P_1)_V\left(\frac{P_1L}{EA}\right)_R + \int (T_1)_V\left(\frac{T_1}{GK_t}ds\right)_R + \int (V_1)_V\left(\frac{V_1Q}{GIb}ds\right)_R \\
&+ \left(P_{1spring}\right)_V\left(\frac{P_{1spring}}{k_{spring}}\right)_R + \left(M_{1spring}\right)_V\left(\frac{M_{1spring}}{k_{\substack{rotational\\spring}}}\right)_R + \left(T_{1spring}\right)_V\left(\frac{T_{1spring}}{k_{\substack{torsional\\spring}}}\right)_R \\
&= \int M_1\frac{M_1}{EI}ds + \int P_1\frac{P_1}{EA}ds \text{ or } P_1\frac{P_1L}{EA} + \int T_1\frac{T_1}{GK_t}ds + \int V_1\frac{V_1Q}{GIb}ds + P_{1sp}\frac{P_{1sp}}{k_{sp}} + M_{1sp}\frac{M_{1sp}}{k_{rot\ sp}} + T_{1sp}\frac{T_{1sp}}{k_{tor\ sp}}
\end{aligned}$$

The shear deformation expression is often simplified from VQ/(GIb) to V/(GA_s).

By definition, EA is the axial rigidity, EI is the flexural rigidity (often denoted D), GA_s is the shear rigidity and GJ or GK_t is the torsional rigidity (often denoted H). For isotropic materials, G is always E/[2(1+ν)].

1.3.2.1 One Dimensional Beam Finite Element

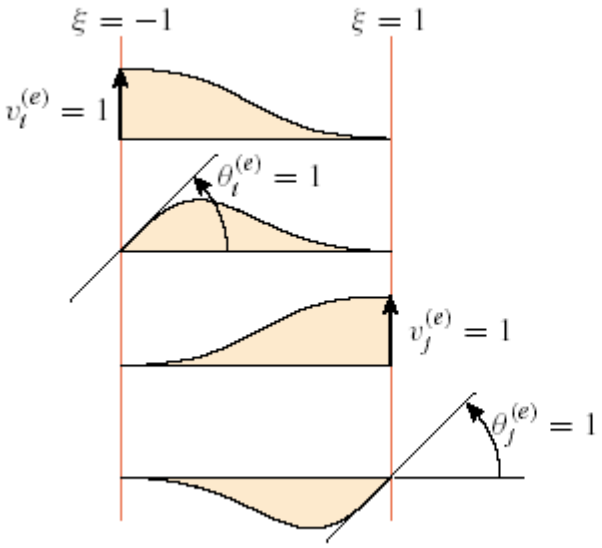
Beam elements such as CBAR should only be used if plane sections remain plane and the neutral axis remains strain free. If an open section clearly warps (plane section no longer plane), then shell elements should be used. Apart from that, small deflection theory must be applicable. Beam theory can be based upon the Bernoulli-Euler beam (also known as the classical or engineering beam) or the Timoshenko beam, the latter of which includes shear deformations.

Beam elements have in general 6 DOFs in their two end nodes. A 2-noded beam element describes in its displacement vector {y}, a **3rd order** complete polynomial

transverse displacement = constant + $x + x^2 + x^3$

variation in one variable x due to the four (two transverse translational and two rotational) nodal DOFs. Writing the transverse displacement in terms of shape functions (with natural coordinates),

$$\text{the transverse displacement} = [\, N_{v_i}^{(e)} \quad N_{\theta_i}^{(e)} \quad N_{v_j}^{(e)} \quad N_{\theta_j}^{(e)} \,] \begin{bmatrix} v_i^{(e)} \\ \theta_i^{(e)} \\ v_j^{(e)} \\ \theta_j^{(e)} \end{bmatrix}$$



$$N_{vi}^{(e)}(\xi) = \tfrac{1}{4}(1-\xi)^2(2+\xi)$$

$$N_{\theta i}^{(e)}(\xi) = \tfrac{1}{8}\ell(1-\xi)^2(1+\xi)$$

$$N_{vj}^{(e)}(\xi) = \tfrac{1}{4}(1+\xi)^2(2-\xi)$$

$$N_{\theta j}^{(e)}(\xi) = -\tfrac{1}{8}\ell(1+\xi)^2(1-\xi)$$

Its strain vector {ε} thus contains a linear variation of curvature (which is defined as the second derivative of the transverse displacement with respect to x along the element). Subsequently, the stress vector {σ} also contains a **linear** variation of bending moment. Note that the strain for a beam finite element is the curvature and the stress is the bending moment. Note that the required continuity between the finite elements is $C^{2-1} = C^1$ since the second derivative of the displacement needs to be defined. The second derivative (which is the curvature) and hence the bending moment will thus be continuous between elements. If the bending moment variation is accurate, so will be the shear force distribution, which is the first derivative of the bending moment. A two-noded beam can model concentrated external forces and moments applied at its nodes exactly because the displacement variation will be cubic, the curvature will be parabolic and the bending moment linear. This is the reason that concentrated loads should never be applied within a beam finite element as the resulting bending moment variation will be piecewise linear, a variation that cannot be modeled by a continuous linear function. Likewise a uniformly distributed load within a beam finite element will require at least a quartic (4^{th} order) displacement variation function and parabolic bending moment variation function, both of which cannot be modeled by a two-noded beam element. Note however that a three-noded beam element will be able to model a uniformly distributed variation in internal load

exactly because the resulting displacement variation will be quintic (5^{th} order) and the resulting bending moment variation will be cubic. If the variation in the displacement function or the stress function is not able to model the actual variation in reality (which is dependent upon the variation of the external loads), there will be a discretization error in the finite element approximation. The approximate order of error in the displacement and the strain and stress of a two-noded beam element are given by

Displacement Error Order = $O(h^{p+1}) = O(h^4)$

Strain (Curvature) / Stress (Bending Moment) Error Order = $O(h^{p-m+1}) = O(h^2)$

where h is the element size, p the order of the *complete* polynomial used in the displacement interpolation and m is the order of the displacement derivative representing the strain. Note that the error is independent of the loading but only occurs if the loading causes a variation which cannot be modeled perfectly by the displacement function. In order to decrease the error, clearly we can employ h-refinement by reducing element size (i.e. increasing the number of elements with the original DOFs still maintained) or employ p-refinement by increasing the order of the *complete* polynomial. Based on the above orders of errors, it is apparent that *finite element analysis is associated with higher accuracy in predicting displacements than in predicting strains and stresses.*

A 2-noded beam element describes in its displacement vector {y}, a **1st order** complete polynomial

axial displacement = constant + x

variation in one variable x due to the two axial translational nodal DOFs. Its strain vector {ε} thus contains a constant variation of axial strain (which is defined as the first derivative of the axial displacement with respect to x along the element). Subsequently, the stress vector {σ} also contains a **constant** variation of axial force. Note that the required displacement continuity between the finite elements is $C^{1-1} = C^0$ since only the first derivative needs to be defined. A two-noded beam can model concentrated external axial forces applied at its nodes exactly (do not apply axial force within element) because the displacement variation will be linear and the axial strain will be constant and the axial stress constant. The approximate order of error in the displacement and the strain and stress of a two-noded bar element are given by

Displacement Error Order = $O(h^{p+1}) = O(h^2)$

Strain (Axial Strain) / Stress (Axial Force) Error Order = $O(h^{p-m+1}) = O(h^1)$

A 2-noded beam element describes in its displacement vector {y}, a **1st order** complete polynomial

torsional rotation = constant + x

variation in one variable x due to the two torsional rotation nodal DOFs. Its strain vector {ε} thus contains a constant variation of torsional strain (which is defined as the first derivative of the torsional rotation with respect to x along the element). Subsequently, the stress vector {σ} also contains a **constant** variation of torsional moment. Note that the required displacement continuity between the finite elements is $C^{1-1} = C^0$ since only the first derivative of the displacement needs to be defined. A two-noded beam can model concentrated external torsional moments applied at its nodes exactly (do not apply torsional moment within element) because the torsional rotation variation will be linear and the torsional strain will be constant and the torsional moment constant.

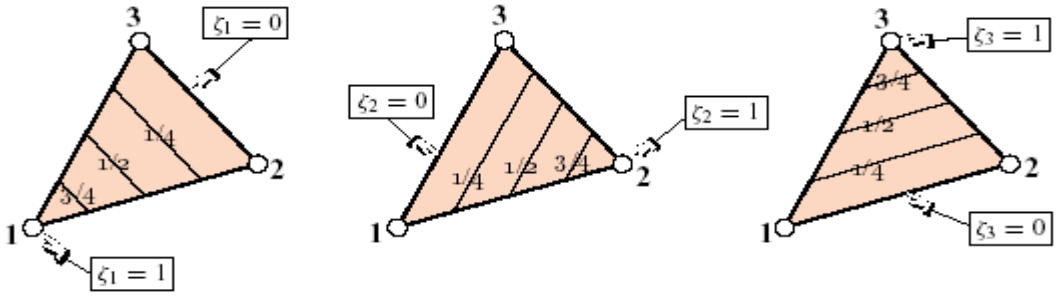
The force-displacement relationship of the Timoshenko beam is shown. For the Euler-Bernoulli beam, β is zero.

$$\begin{Bmatrix} F_1 \\ M_1 \\ F_2 \\ M_2 \end{Bmatrix} = \frac{EI}{L^3(1+\beta)} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & (4+\beta)L^2 & -6L & (2-\beta)L^2 \\ -12 & -6L & 12 & -6L \\ 6L & (2-\beta) & -6L & (4+\beta)L^2 \end{bmatrix} \begin{Bmatrix} \delta_1 \\ \theta_1 \\ \delta_2 \\ \theta_2 \end{Bmatrix} \qquad \beta = \frac{12EI}{GAL^2}$$

1.3.2.2 Two Dimensional Shell Elements (In-Plane Plane Stress or Plane Strain Membrane Stiffness)

Shell elements have in general 5 DOFs in all their nodes. Hence, general shell elements have in-plane stiffnesses (of either plain stress or plane strain) and also transverse bending, shear and torsional stiffnesses. Plate elements by definition have only the transverse bending, shear and torsional stiffnesses. The definition of in-plane stiffnesses is important when the shells map a curved geometry or when the geometry becomes curved as a result of deformations in a nonlinear analysis. In these cases, the transverse loadings are resisted both by transverse bending (and shear) stiffness and also by in-plane stiffnesses. In other words, the membrane and bending actions are coupled in curved shells and when deflections are large in a nonlinear analysis.

Note that plane stress elements refer to thin (with respect to the other dimensions) shells such that transverse stresses due to in-plane forces do not develop because the out-of-plane dimension is not constrained such as in plate girder webs. Plane strain elements refer to thick (with respect to the other dimensions) shells such that transverse strains due to in-plane forces do not develop because of the out-of-plane constraint such as the slice of a retaining wall, a dam or a long pipe subjected to internal pressure. The mid-plane of the shell element lies exactly halfway between the two faces. All in-plane loads act along the mid-plane and are symmetric with respect to the mid-plane. All boundary conditions are also symmetric with respect to the mid-plane.



A 3-noded triangular shell element (known as a constant strain triangle CST) describes in its displacement vector {y}, a **1st order** complete polynomial

in–plane u = constant + x + y

in–plane v = constant + x + y

variation in two variables x, y due to the six (two in-plane displacements at the three nodes) nodal DOFs. Writing the in-plane displacement in terms of shape functions (with natural coordinates) and also including the isoparametric representation of the geometry (i.e. in terms of the shape functions),

$$\begin{bmatrix} 1 \\ x \\ y \\ u_x \\ u_y \end{bmatrix} = \begin{bmatrix} 1 & 1 & 1 \\ x_1 & x_2 & x_3 \\ y_1 & y_2 & y_3 \\ u_{x1} & u_{x2} & u_{x3} \\ u_{y1} & u_{y2} & u_{y3} \end{bmatrix} \begin{bmatrix} N_1^{(e)} \\ N_2^{(e)} \\ N_3^{(e)} \end{bmatrix}, \qquad N_1^{(e)} = \zeta_1, \qquad N_2^{(e)} = \zeta_2, \qquad N_3^{(e)} = \zeta_3.$$

Its strain vector {ε} thus contains a constant variation of two orthogonal normal strains and one shear strain (which are defined as the first derivatives of the in-plane displacement with respect to x, y and xy respectively within the element). Subsequently, the stress vector {σ} also contains a **constant** variation of two orthogonal normal stresses and one shear stress. Because of the constant nature of the stresses, 4-noded or 8-noded quadrilaterals or 6-noded triangles are thus recommended in general. Note that the required displacement continuity between the finite elements is C^0 since only the first derivative of the displacement needs to be defined. Thus the stress, which is the first derivative of the displacement, will not be continuous between elements. A fine mesh will thus be necessary to model large variations in stress. If the variation in the displacement function or the stress function is not able to model the actual variation in reality (which is dependent upon the variation of the external loads), there will be a discretization error in the finite element approximation. The approximate order of error in the displacement and the strain and stress is given by

Displacement Error Order = $O(h^{p+1}) = O(h^2)$

Strain / Stress Error Order = $O(h^{p-m+1}) = O(h^1)$

A 6-noded triangular (serendipity) shell element describes in its displacement vector {y}, a **2nd order** complete polynomial

in–plane u = constant + x + y + x^2 + xy + y^2

in–plane v = constant + x + y + x^2 + xy + y^2

variation in two variables x, y due to the six (two in-plane displacements at the six nodes) nodal DOFs. Writing the in-plane displacement in terms of shape functions (with natural coordinates) and also including the isoparametric representation of the geometry (i.e. in terms of the shape functions),

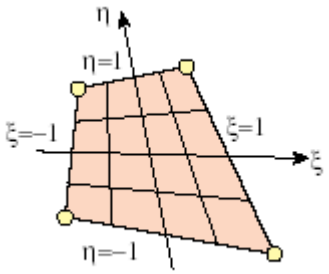
$$\begin{bmatrix} 1 \\ x \\ y \\ u_x \\ u_y \end{bmatrix} = \begin{bmatrix} 1 & 1 & 1 & 1 & 1 & 1 \\ x_1 & x_2 & x_3 & x_4 & x_5 & x_6 \\ y_1 & y_2 & y_3 & y_4 & y_5 & y_6 \\ u_{x1} & u_{x2} & u_{x3} & u_{x4} & u_{x5} & u_{x6} \\ u_{y1} & u_{y2} & u_{y3} & u_{y4} & u_{y5} & u_{y6} \end{bmatrix} \begin{bmatrix} N_1^{(e)} \\ N_2^{(e)} \\ N_3^{(e)} \\ N_4^{(e)} \\ N_5^{(e)} \\ N_6^{(e)} \end{bmatrix}.$$

$$N_1^{(e)} = \zeta_1(2\zeta_1 - 1), \quad N_2^{(e)} = \zeta_2(2\zeta_2 - 1), \quad N_3^{(e)} = \zeta_3(2\zeta_3 - 1),$$
$$N_4^{(e)} = 4\zeta_1\zeta_2, \quad N_5^{(e)} = 4\zeta_2\zeta_3, \quad N_6^{(e)} = 4\zeta_3\zeta_1.$$

Its strain vector {ε} thus contains a linear variation of two orthogonal normal strains and one shear strain (which are defined as the first derivatives of the in-plane displacement with respect to x, y and xy respectively within the element). Subsequently, the stress vector {σ} also contains a **linear** variation of two orthogonal normal stresses and one shear stress. Note that the required displacement continuity between the finite elements is C^0 since only the first derivative of the displacement needs to be defined. Thus the stress, which is the first derivative of the displacement, will not be continuous between elements. A fine mesh will thus be necessary to model large variations in stress. If the variation in the displacement function or the stress function is not able to model the actual variation in reality (which is dependent upon the variation of the external loads), there will be a discretization error in the finite element approximation. The approximate order of error in the displacement and the strain and stress is given by

Displacement Error Order = $O(h^{2+1}) = O(h^3)$

Strain / Stress Error Order = $O(h^{p-m+1}) = O(h^2)$



A 4-noded quadrilateral shell element describes in its displacement vector {y}, a **1st order** complete polynomial

in–plane u = constant + x + y + xy

in–plane v = constant + x + y + xy

variation in two variables x, y due to the eight (two in-plane displacements at the four nodes) nodal DOFs. Writing the in-plane displacement in terms of shape functions (with natural coordinates) and also including the isoparametric representation of the geometry (i.e. in terms of the shape functions),

$$\begin{bmatrix} 1 \\ x \\ y \\ u_x \\ u_y \end{bmatrix} = \begin{bmatrix} 1 & 1 & 1 & 1 \\ x_1 & x_2 & x_3 & x_4 \\ y_1 & y_2 & y_3 & y_4 \\ u_{x1} & u_{x2} & u_{x3} & u_{x4} \\ u_{y1} & u_{y2} & u_{y3} & u_{y4} \end{bmatrix} \begin{bmatrix} N_1^{(e)} \\ N_2^{(e)} \\ N_3^{(e)} \\ N_4^{(e)} \end{bmatrix}.$$

$$N_1^{(e)} = \tfrac{1}{4}(1-\xi)(1-\eta), \qquad N_2^{(e)} = \tfrac{1}{4}(1+\xi)(1-\eta),$$
$$N_3^{(e)} = \tfrac{1}{4}(1+\xi)(1+\eta), \qquad N_4^{(e)} = \tfrac{1}{4}(1-\xi)(1+\eta).$$

Its strain vector $\{\varepsilon\}$ thus contains ε_x constant in x but linear in y, ε_y constant in y but linear in x and γ_{xy} linear in x and y, the strains are defined as the first derivatives of the in-plane displacement with respect to x, y and xy respectively within the element. Subsequently, the stress vector $\{\sigma\}$ also contains σ_x **constant** in x but linear in y, σ_y **constant** in y but linear in x and τ_{xy} **linear** in x and y. Note that the required displacement continuity between the finite elements is C^0 since only the first derivative of the displacement needs to be defined. Thus the stress, which is the first derivative of the displacement, will not be continuous between elements. A fine mesh will thus be necessary to model large variations in stress. The four-noded shell element is however much better then the three noded shell as the variation of stress within the element is linear in some senses, i.e. although the variation of the normal stress in x is constant in x, it is linear in y. If the variation in the displacement function or the stress function is not able to model the actual variation in reality (which is dependent upon the variation of the external loads), there will be a discretization error in the finite element approximation. The approximate order of error in the displacement and the strain and stress is given by

Displacement Error Order = $O(h^{p+1}) = O(h^2)$
Strain / Stress Error Order = $O(h^{p-m+1}) = O(h^1)$

An 8-noded quadrilateral (serendipity) shell element describes in its displacement vector {y}, a **2nd order** complete polynomial

in–plane u = constant + x + y + xy + x^2 + xy + y^2 + x^3y
in–plane v = constant + x + y + xy + x^2 + xy + y^2 + xy^3

variation in two variables x, y due to the eight (two in-plane displacements at the eight nodes) nodal DOFs. Its strain vector $\{\varepsilon\}$ thus contains ε_x linear in x but parabolic in y, ε_y linear in y but parabolic in x and γ_{xy} parabolic in x and y, the strains are defined as the first derivatives of the in-plane displacement with respect to x, y and xy respectively within the element. Subsequently, the stress vector $\{\sigma\}$ also contains σ_x **linear** in x but parabolic in y, σ_y **linear** in y but parabolic in x and τ_{xy} **parabolic** in x and y. Note that the required displacement continuity between the finite elements is C^0 since only the first derivative of the displacement needs to be defined. Thus the stress, which is the first derivative of the displacement, will be continuous between elements. If the variation in the displacement function or the stress function is not able to model the actual variation in reality (which is dependent upon the variation of the external loads), there will be a discretization error in the finite element approximation. The approximate order of error in the displacement and the strain and stress is given by

Displacement Error Order = $O(h^{2+1}) = O(h^3)$
Strain / Stress Error Order = $O(h^{p-m+1}) = O(h^2)$

1.3.2.3 Two Dimensional Shell Elements (Out of Plane Bending, Shear and Torsional Plate Stiffness) [1]

The use of solid elements of model thin plates would be extremely expensive unless one of dimensions of the solid is allowed to be very small in comparison to the other two. Thin solids however would be very badly conditioned. Hence the development of plates and shells. By definition plates resist bending, membranes resist in-plane forces and shells resist both bending and in-plane membrane actions.

Shell elements have in general 5 DOFs in all their nodes. Hence, general shell elements have in-plane stiffnesses (of either plain stress or plane strain) and also transverse bending, shear and torsional stiffnesses. Plate elements by definition have only the transverse bending, shear and torsional stiffnesses. The definition of in-plane stiffnesses is important when the shells map a curved geometry or when the geometry becomes curved as a result of deformations in a nonlinear analysis. In these cases, the transverse loadings (i.e. normal to the surface) are resisted both by transverse bending (and shear) stiffness and also by in-plane stiffnesses, i.e. coupled membrane-bending behavior. To reiterate, flat plates can behave as membranes with no bending strains or as bending elements with no membrane action when the loading is in-plane or out-of-plane. This would be an incorrect representation of reality in the two cases, namely when the deflections are large in a nonlinear analysis and when the initial undeflected geometry is curved to start with. In these cases, the transverse and in-plane loadings are resisted by both bending and membrane action, i.e. their stiffnesses are coupled. Note that membrane actions is far more efficient than bending action. Resisting loading by bending is generally an inefficient method. For out-of-plane loading, it must be remembered that shell models are used to model *thin* flat plate structures subjected to transverse loading in reality. Possible mathematical models include: -

1. *Membrane shell mode*l: for **extremely thin** plates dominated by membrane effects, such as inflatable structures and fabrics (parachutes, sails, etc).
2. *Von-Karman mode*l: for **very thin** bent plates in which membrane and bending effects interact strongly on account of finite lateral deflections. Important model for post-buckling analysis.
3. *Kirchhoff mode*l: for **thin** bent plates with small deflections, negligible shear energy and uncoupled membrane-bending action.
4. *Reissner-Mindlin mode*l: for **thin and moderately thick** bent plates in which first-order transverse shear effects are considered. Particularly important in dynamics as well as honeycomb and composite wall constructions.
5. *High order composite model*s: for detailed (local) analysis of layered composites including inter-lamina shear effects.
6. *Exact model*s: for the analysis of additional effects using three-dimensional elasticity.

The first two models require *geometrically nonlinear* methods of analyses. The last four models are *geometrically linear* in the sense that *all governing equations are set up in the undeformed flat configuratio*n. The last two models are primarily used in detailed or local stress analysis near edges, point loads or openings. All models may incorporate other types of nonlinearities due, for example, to material behavior, composite fracture, cracking or delamination, as well as certain forms of boundary conditions. The Kirchhoff and Reissner-Mindlin plate models are the most commonly used shells models in statics and vibrations, respectively.

The Kirchhoff model requires the thickness dimension *h* to be everywhere *small*, but not too small, compared to a *characteristic length L* of the plate mid-surface. The term "small" is to be interpreted in the engineering sense and not in the mathematical sense. For example, h/L is typically 1/5 to 1/100 for most plate structures. A paradox is that an extremely thin plate, such as the fabric of a parachute or a hot air balloon, ceases to function as a thin plate! The Kirchhoff model also assumes *inextensional bendin*g i.e. the plate does not experience appreciable stretching or contractions of its mid-surface. This is simply known as *plate bending*. The Von-Karman model on the other hand assumes e*xtensional bendin*g, i.e. the mid-surface experiences significant stretching or contraction. This is also called *coupled membrane-bendin*g.

[1] FELIPPA, Carlos A. *Advanced Finite Element Methods.* Center for Aerospace Structures, University of Colorado, 2001.

Note that analyzing plate bending as a series of beams bending in two orthogonal directions ignores the role of twisting moment and also the Poisson contractions.

The Euler-Bernoulli beam and the Kirchhoff plate model both ignore transverse shear deformations (although the shear force is of course still calculated). The assumptions of these models are that the planes normal to the mid-surface remain plane and normal (orthogonal) to the deformed mid-surface. On both the Euler-Bernoulli beam and the Kirchhoff plate, without shear deformation, the required displacement continuity between elements is C^1. The Kirchhoff plate uses one parameter to describe the displacement field, for a QUAD4 with 3 DOFs at each node (two out-of-plane rotational and one transverse displacement),

$$w = \text{constant} + x + y + x^2 + xy + y^2 + x^3 + y^3 + x^2y + xy^2 + x^3y + xy^3$$

which is a complete 3rd order polynomial (incomplete 4th order). Curvature (and stress and moment) would thus be linear like the Euler-Bernoulli beam. But this element is incompatible in normal slope. In fact, Kirchhoff elements generally suffer from the inability to be fully C_1 compatible. However, these incompatibilities reduce to zero with mesh refinement and the elements can performs satisfactorily. Another failing of the Kirchhoff elements is that they cannot be generalized to curved side configurations.

The Discrete Kirchhoff elements on the other hand use independent displacement and rotation interpolation fields w, θ_x and θ_y much like the Mindlin approach which abandons the simple Kirchhoff bending theory altogether.

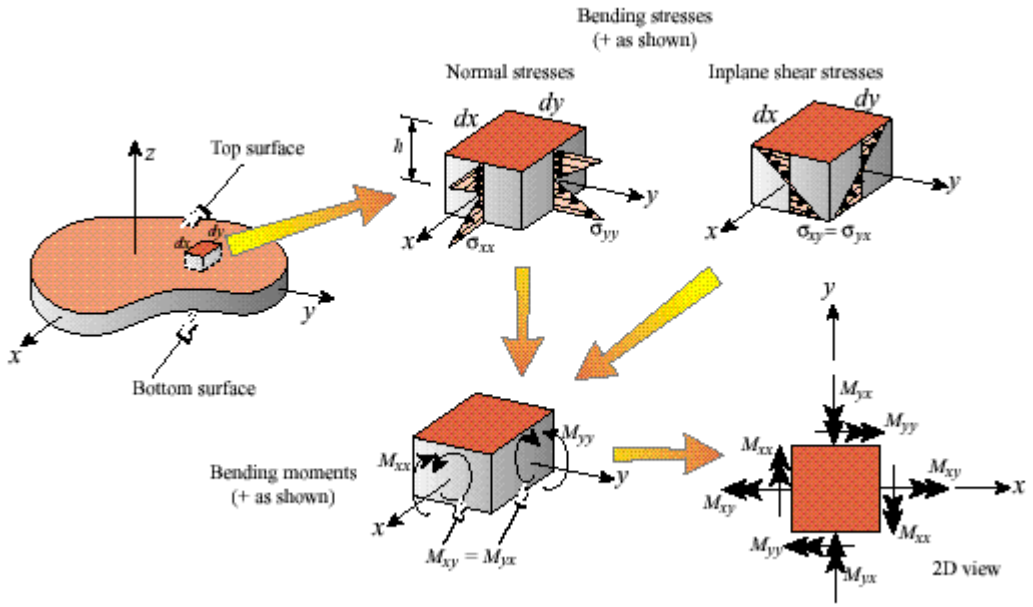
The Timoshenko beam hand includes shear deformation correction terms in its stiffness matrix. In an analogous way, shear correction terms can be included in the Mindlin plate model to represent deeper plates. Here, planes normal to the mid-surface remain plane and but not necessarily normal (orthogonal) to the deformed mid-surface. On both the Timoshenko beam and the Mindlin plate, with shear deformation, the required displacement continuity between elements is C^0. Unlike the Kirchhoff plate element which uses just one parameter to describe the displacement field, the Mindlin model uses uncoupled displacement and rotational fields w, θ_x and θ_y. Each node still has 3 DOFs (two out-of-plane rotational and one transverse displacement). The curvature is no longer the second derivative of the displacement field, instead the first derivative of the rotation field. Because the highest derivative is only the first derivative, the continuity requirement is only C^0. The Mindlin model also automatically results in through-thickness transverse shear strains (unlike the Kirchhoff model which required equilibrium considerations). The Mindlin model is superior to the Kirchhoff model if the plate is thick **as the shear deformations become important when the thickness of the plate is greater than about 1/10 of the plate width in a physical sense, not mathematical sense of the element**. This is because the Mindlin model represents the shear deformation directly instead of having to infer it as the derivative of the bending moment, the latter of which is more inaccurate. Modelling thin plates with Mindlin elements on the other hand lead to the susceptibility of shear locking (a form of ill-conditioning as the shear strains become relatively insignificant). The condition can be avoided by using reduced integration for the shear stiffness terms whilst the bending stiffness terms can still be fully integrated to avoid spurious modes. Hence a QUAD8 would have a 3 x 3 Gauss rule for bending but only a 2 x 2 for shear. The Mindlin model can also be generalized to curved isoparametric formulations.

1.3.2.3.1 Kirchhoff Plate Element

Like beams, Kirchhoff plate elements are C^1 plate elements because that is the continuity order nominally required for the transverse displacement shape functions. The Kirchhoff model is applicable to elastic plates that satisfy the following conditions.

(i) The plate is *thin* in the sense that the thickness *h* is small compared to the characteristic length(s), but not so thin that the lateral deflection become comparable to h.
(ii) The plate thickness is either uniform or varies slowly so that 3-dimensional stress effects are ignored.
(iii) The plate is symmetric in fabrication about the mid-surface.
(iv) Applied transverse loads are distributed over plate surface areas of dimension *h* or greater.6
(v) The support conditions are such that no significant extension of the mid-surface develops.

The kinematics of a Bernoulli-Euler beam is based on the assumption that *plane sections remain plane and normal to the deformed longitudinal axis*. The kinematics of the Kirchhoff plate is based on the extension of this assumption to *biaxial bendin*g i.e. material normals to the original reference surface remain straight and normal to the deformed reference surface. *Material normal*s, also called *material filament*s, are directed along the *normal* to the mid-surface (that is, in the element *z* direction) and extending h/2 above and h/2 below it. The magnitude *h* is the *plate thickness*.



As shown on the diagram, the variation of the two orthogonal direct stresses and the one (bending or in-plane, as opposed to transverse) shear stress over the thickness of the element is linear. These thus give rise to moments when integrated.

$$M_{xx} = -\int_{-h/2}^{h/2} \sigma_{xx}\, z\, dz,$$

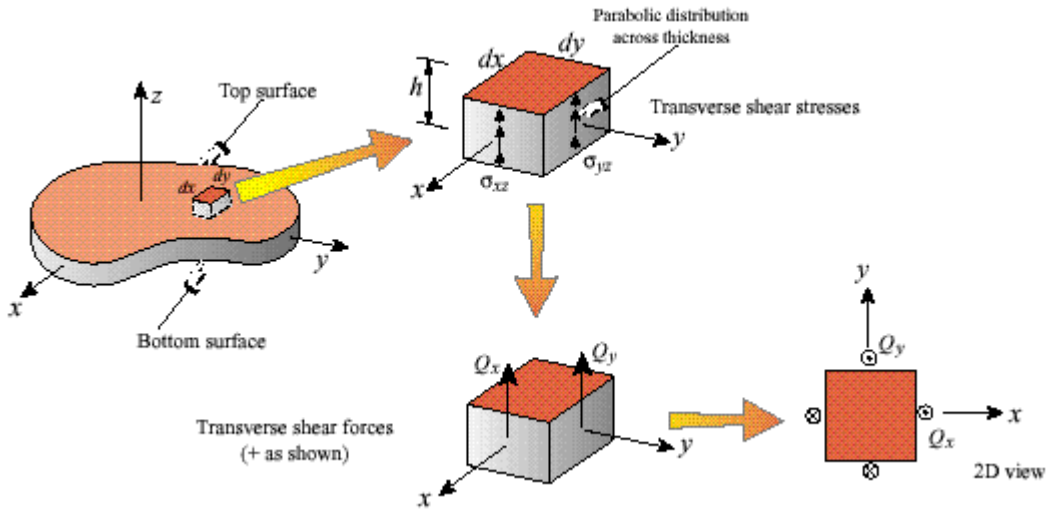
$$M_{yy} = -\int_{-h/2}^{h/2} \sigma_{yy}\, z\, dz,$$

$$M_{xy} = -\int_{-h/2}^{h/2} \sigma_{xy}\, z\, dz,$$

$$M_{yx} = -\int_{-h/2}^{h/2} \sigma_{yx}\, z\, dz.$$

The Kirchhoff model suffers from 2 inconsistencies. Firstly, the transverse shear strains are zero. If the plate is isotropic and follows Hooke's law, this implies $\sigma_{xz} = \sigma_{yz} = 0$ and consequently there are no transverse shear forces. But these forces appear necessarily from the equilibrium equations. Secondly, $\varepsilon_{zz} = 0$ says that the plate is in plane strain whereas for plane stress $\sigma_{zz} = 0$ is a closer approximation to the physics. For a homogeneous isotropic plate, plane strain and plane stress coalesce if and only if Poisson's ratio is zero. Both inconsistencies are similar to those encountered in the Bernoulli-Euler beam model.

As in the case of Bernoulli-Euler beams, predictions of the transverse shear stresses must come entirely from equilibrium analysis. The Kirchhoff plate model ignores the transverse shear energy, and in fact predicts $\sigma_{xz} = \sigma_{yz} = 0$ from the kinematic equations. In practical terms this means that the transverse shear stresses should be significantly smaller than the bending stresses. If they are not, the Kirchhoff model does not apply, nor does the Bernoulli-Euler beam for that matter. From equilibrium conditions, the transverse shear stresses and hence transverse shear forces can be derived.



As indicated on the diagram, the variation of the two transverse shear stresses over the thickness of the element from equilibrium conditions is parabolic, peaking at the neutral axis. These thus give rise to transverse shear forces when integrated.

$$Q_x = \int_{-h/2}^{h/2} \sigma_{xz}\, dz = \tfrac{2}{3}\sigma_{xz}^{max}\, h, \qquad Q_y = \int_{-h/2}^{h/2} \sigma_{yz}\, dz = \tfrac{2}{3}\sigma_{yz}^{max}\, h,$$

The only term not calculated for plates is the σ_{zz}, and hence the F_{zz}.

1.3.3 Finite Element Modelling For Static Analyses

1.3.3.1 Choice of Finite Element to Model the Load Path and Deformation

The choice of the finite element whether beams, shells or solids depend on the stiffness (which in turn depends on the geometry) that we are trying to model in reality. **Bending deformations** govern in beams with span to depth **(S/D) ratios greater than 10** and of course finite element beams with shear deformations can be used to analyze and design. When **S/D less than 10**, **shear deformations** should be included as well, and finite element beams with shear deformations can be used to analyze and design. The inclusion of shear deformations will correctly decrease the stiffness of the element. For **S/D less than 2.5** it becomes a **deep beam** and thus finite element beams may be used to analyze but deep beam design must be employed. When **S/D is less than 1** the section is **not a beam**, i.e. flexural deformations (and shear deformations do not occur and the load path is much more direct; note that flexural stiffness is much smaller than axial stiffness). This occurs in shear walls where the load path is such that the wall arches instead of bending like a beam. Shear walls should thus be modeled using vertically orientated shells or even simply plane stress membranes when there are no out-of-plane forces. Another example of an S/D less than 1 section is the capping beam of contiguous piles, used for foundations. It is inappropriate to model the capping beam as finite element beams between piles with a very small spacing in between such that they are effectively continuous (hence the name contiguous piles). Solid elements are really the only practical option. Alternatively, if explicit modelling is not required, but simply a good representation of the capping beam stiffness, the section can be made rigid as it does not behave like a beam. This is akin to modelling sections that are far more rigid than adjacent sections as rigid for a global representation, and only then performing a local analysis on the section explicitly modeled if required. The design of the capping beam can then be based on a strut and tie model.

Flat thin plates such as flat slabs can be modeled using a mesh of 2D shells (shells encompass both bending plates and in-plane membrane plane stress elements) or simplistically as a grillage of 1D beam elements, which effectively constrains the load paths. Ribbed slabs are best modeled with a combination of shells and beam elements. I-beams, tubes and cylinders can be modeled explicitly using plane stress shell elements. Complicated geometries that do not act like simple beams or thin shells must be modeled with solids.

1.3.3.2 Concepts of Stiffness

Concepts of stiffness are crucial in linear static analyses. It is worth noting that the equivalent stiffness of springs in parallel is the addition of the individual stiffnesses and the equivalent stiffness of springs in series is the inverse of the addition of the inverse of the individual stiffnesses. Stiffness in series occurs when the **load path has to travel both stiffnesses** sequentially. Stiffness in parallel occurs when the **load path is given the option to be more attracted to the stiffer element**. In series, there is no question of whether the load will travel the path or not, for it must, all loads must travel some path, and in series, there is no other option. Hence, elements of different stiffness in series will all experience the same load effects. In a statically indeterminate system, the elements with a greater stiffness in a particular DOF (with other elements attached to the same DOF, hence in parallel) will attract more force, because they are more resistant to deformation. The parallel members with smaller relative stiffness contribution will conversely attract less force. However, all the load that is applied must eventually be balanced by the reactions at the supports, i.e. all the applied load must be resisted in line the principle of conservation of energy. The statically indeterminate system simply distributes this applied loading according to its distribution of stiffness, eventually to its supports. Let us reiterate that in a linear static analysis, all the applied forces must be balanced by the reactions. Deviations of the sum of the applied loads and the reactions from zero are an indication of ill conditioning. These deviations also occur when there is a loss of internal strain energy. This happens when there is artificial rotational stiffness applied onto the rotational DOFs on 2D shells that do not have their own in-plane rotational stiffness in order that the originally singular equations may be solved. There is thus obviously a loss of accuracy as some of the applied loads are not resisted by the structure but lost into the artificial constraints.

1.3.4 Finite Element Modelling For Dynamic Analyses

Modelling for dynamic analyses involves additional considerations on top of those for static analyses.

1.3.4.1 Mass and Stiffness Distribution

A static mesh is usually refined as locations of high stress concentration. A dynamic mesh is often more uniform as the consideration of not only the location of the loading and inherent geometry that cause stress concentrations (as with static analysis), but also the time variation of the loading and the nature of the response of a dynamic analysis where higher modes cause local stress concentrations which are not observed in a static analysis.

As a minimum requirement, the lower modes of the structure must have the correct resonant frequencies and mode shapes. The lowest modes of vibration generally involve mode shapes that **minimize the strain energy** and **maximize the kinetic energy**. Imagine the natural bending modes of a simply-supported beam. The higher the bending mode, the more the strain energy in the system for a maximum amplitude of displacement. Also, the higher the bending mode, the less the kinetic energy in the system as more points within the beam are stationary. Uniform distributions of mass and stiffness require a uniform mesh. Since minimizing the strain energy requires a detailed description of the stiffness distribution, the stiffness matrix must be as accurate as possible. Generally, a dynamic analysis can have a coarser mesh than a static analysis because the response tends to be controlled by the low frequency behavior, usually much less than 10% of the total number of possible modes is necessary. **Areas of low stiffness must be accurately modeled**, as these are the areas where deformation is largest and since the stiffness is low, the strain energy will be minimized. Note also that since deformations are high, the kinetic energy will be high, thus giving the two conditions required to simulate low frequency modes accurately. On the other hand, a coarse mass distribution with lumped masses (instead of the more expensive banded kinematically equivalent coupled mass matrix) is generally sufficient as maximizing the kinetic energy is much easier to achieve. More care must be taken with mass description if the higher modes of vibration are required, where coupled mass formulations may be more accurate. **Large masses must be correctly positioned in the model**. In order to simulate lower modes accurately, to maximize the kinetic energy, the nodes should be positioned where the mass is large.

If only the first few resonance frequencies and mode shapes are required, then a relatively coarse mesh can be used. If higher frequencies are required, then the mesh must be suitably fine to model the more complicated mode shapes. A good parameter may be the modal mass. We know that the modal mass of all the bending modes of a beam is half the beam mass if simply supported and quarter the beam mass if a cantilever. When higher modes deviate from the modal mass value, then we know that the mode shape cannot be captured by the mesh. With more complicated structures, an h- or p-refinement should be performed and the **modal masses of the original and the refined mesh should be compared**; modal masses of higher modes which begin to show discrepancies indicates the extent of the capability of the mesh in capturing these higher modes.

Modal eigenvector displacement and frequencies response does not require a very fine mesh. If a **dynamic stress analysis** is to be performed, then the mesh must be comparably fine to that for a static stress analysis. **Acceleration response** requires a finer mesh than displacement response, as good accuracy of higher modes is required.

1.3.4.2 Nonstructural Mass

The dynamic mass often includes the medium surrounding the structure. For instance, in the analysis of offshore gravity platforms, some degree of the mass of the water surrounding the shafts must be included within the mass. Noting that the mass acts in 3 translational orthogonal directions (likewise inertia acts about 3 orthogonal axes), it is essential that the correct component be modeled. The added mass on the shafts is a horizontal mass component whilst the added mass on the base of the gravity base structure is a vertical mass component. The inclusion of added masses on the dynamic model also means that a uniform global acceleration applied to all parts of the structure to model gravitational force cannot be employed, as the added mass will incorrectly increase the weight of

the structure. Instead, the gravitational force must be modeled using explicit nodal loads or more accurately, gravitational accelerations applied to a set of masses that excludes the added masses.

Another important consideration when modelling non-structural mass is that if rigid links are used to model the center of gravity of the non-structural mass from the structure, then a coupled mass formulation (kinematically equivalent mass matrix) must be used. This is because if lumped mass formulation is used, the rigid links simply will transfer the non-structural mass onto the structure, and hence effectively not incorporating the offset. This will cause significant errors in the lowest frequencies. The offset mass increases the rotational inertia and this has the effect of reducing some of the natural frequencies and hence potentially increasing the response.

1.3.5 Modelling Mechanisms With Inertia Relief – Modelling Displacement Effects Due to Applied Loads On Systems in Conditionally Stable Equilibrium

Inertia relief is used to model mechanisms and unconstrained structures in static solutions. The structure is assumed to be in a steady-state condition, i.e. no transient dynamic effects. The displacements obtained will be relative to a moving frame of reference. The inertia (mass) of the structure is used to resist the applied loadings, that is, an assumption is made that the structure is in a state of static equilibrium (steady-state) even though it is not constrained. Examples include a spacecraft in orbit, an aircraft in flight or an automobile on a test track. The method is used to find out what happens to these structures relative to their moving frame of reference (due to their steady-state condition) when an additional excitation is applied.

There are two ways to invoke the inertia relief. The first method is to specify the SUPORT entry explicitly by including "PARAM, INREL, -1" in the Bulk Data Section. The second method (the recommended method) is to let MSC.NASTRAN select the SUPORT degrees of freedom automatically by including "PARAM, INREL, -2" in the Bulk Data Section. Either way, all 6 possible rigid body motions must be constrained by SUPORT. An optional "PARAM, GRDPNT, x"—where x is a grid point ID—can be specified in the model. If "PARAM, GRDPNT, x" is used, the loads and accelerations will be summed about this point. If "PARAM, GRDPNT, x" is not specified, then the loads and accelerations will be summed about the origin of the basic coordinate system.

When inertia relief is specified, MSC.NASTRAN calculates the forces that result from a rigid body acceleration about the point specified on the "PARAM, GRDPNT, x" in the specified directions. MSC.NASTRAN then calculates the summation of all applied loadings in the same directions. Accelerations are applied to the structure in the appropriate directions to "balance" the applied loadings. The structure is now in a state of static equilibrium, i.e., the summation of all applied loads is 0.0. Since the problem is not constrained, rigid body displacement is still possible. The next step performed by MSC. NASTRAN is to constrain the SUPORT degrees of freedom to a displacement of 0.0 and provide the relative motion of all other grid points with respect to that reference point. Hence, the term "reference" degree of freedom is used to describe the SUPORT degrees of freedom in MSC. NASTRAN. The set of degrees of freedom described on the SUPORT entry belong to the *r*-set or reference set for the solution. The computed solution is the correct one, and it is relative to any rigid body motion that is occurring.

To perform **manual** inertia relief, the following is undertaken.

$ CASE CONTROL SECTION									
PARAM, INREL, -1									
$ BULK DATA									
SUPORT	**GID1**	**C1**	**GID2**	**C2**	**GID3**	**C3**	**GID4**	**C4**	

The inertia relief method requires that a realistic mass distribution exists, and the degrees of freedom listed on the SUPORT entry must be connected elastically to the model in all specified 6 degrees of freedom. (For example, degrees of freedom 4, 5, and 6 on a grid point with only solid elements attached cannot be used since solid elements have no stiffness in these degrees of freedom). It is good practice (but not required by MSC.NASTRAN) to use the same grid point on PARAM, GRDPNT and on the SUPORT entry.

An additional feature allowed in Solution 101 is the solution of a problem under uniform acceleration. This problem is posed using the DMIG, UACCEL Bulk Data entry in addition to the previously mentioned requirements. In this case, uniform accelerations are applied to the model, and the solution is found. Uniform accelerations are useful for situations, such as spacecraft liftoff and landing loadings, which are often specified as static accelerations.

DMIG	UACCEL	"0"	"9"	TIN					
DMIG	UACCEL	L			G1	C1	X1		
	G2	C2	X2		G3	C3	X3		

The OLOAD output will present the applied loads and also 6 sections for the loads necessary to impose unit accelerations in the 6 DOFs about the support point.

Whenever a SUPORT entry is used in static analysis, the epsilon and strain energy printed in the table from UIM 3035 should be checked. The values printed for epsilon and strain energy in the UIM 3035 table should all be numeric zero. The strain energy printed in this table for matrix KLR represents the strain energy in the model when the one SUPORT degree of freedom is moved 1.0 unit, while all other SUPORT degrees of freedom are constrained. If the SUPORT degrees of freedom are properly specified, the model should be capable of rigid body motion (strain-free) with no strain energy. The values printed for the strain energy indicate the ability of the model to move as a rigid body. These values should always be checked. If the structure is not constrained, the values should be numeric zero, but roundoff almost always results in a small nonzero value. Acceptable values are a function of the units, size of the structure, and precision of the hardware.

```
*** USER INFORMATION MESSAGE 3035 (SOLVER)
                              FOR DATA BLOCK   KLR
   SUPPORT PT.NO.              EPSILON              STRAIN    ENERGY
                1          2.3344220E-15          0.0000000E+00
                2          2.3344220E-15          7.2759576E-12
                3          2.3344220E-15          0.0000000E+00
                4          2.3344220E-15          0.0000000E+00
                5          2.3344220E-15          0.0000000E+00
                6          2.3344220E-15          5.3551048E-09
```

The element forces and stresses will be zero since it is motion without strain that we are modelling. Hence the SPC forces corresponding to the SUPORT freedoms will also be zero. The displacements of the structure of course will not be zero except at the SUPORT location, hence the displacements are relative to the SUPORT.

Note that the elements used in inertia relief must have mass and inertia. Hence CBAR elements may be unreliable at times (when they are collinear), thus it is recommended that CBEAMs used instead.

In manual inertia relief, when the r-set is chosen such that structure is not constrained in a statically determinate manner, or if the stiffness attached to these points is inadequate, the rigid body mode shapes may be poor, leading to low accuracy in inertial load calculations. A poor static analysis solution may also result from the poor constraints, independent of the loading problems. Both effects may result in poor overall accuracy. Poor results can occur with little warning when the SUPORT entry is misused. To perform **automatic** inertia relief, the following is undertaken.

```
$ CASE CONTROL SECTION
PARAM, INREL, -2
```

The specification of a SUPORT entry is no longer required. The reference frame is selected automatically, in a manner that poor solutions are unlikely because of the choice of reference frame variables. The DMIG, UACCEL cannot be used with automatic inertia relief.

1.3.6 Submodelling Techniques for Static Analysis [2]

Submodelling refers to refined analysis of a local area after a global analysis is carried out. This clearly eliminates the need for a refined global mesh or a complicated transition to a refined mesh whilst saving computational effort. **Substructuring** on the other hand, refers to a procedure that condenses a group of finite elements into one element called a **superelement**. The computational effort for is greatly reduced if the superelement is used within a structure instead of all the individual elements.

Submodelling can **(and should)** be performed using ***substructuring techniques such as static condensation using superelements (for the remaining parts of the structure)***, but, in practice, other alternative techniques are used for linear static analysis. These alternatives are ***boundary internal forces*** and ***boundary enforced displacements*** techniques. When local analysis is to be performed in a zone of a structure that probably is going to be modified (thickness increments, local reinforcements, etc) from local analysis results (stresses, displacements), the best approach to set appropriate boundary conditions is using the substructuring technique of *static condensation* with MSC.NASTRAN superelements for the remaining parts of the structure. This technique will give **exact results**. *Boundary internal forces* and *enforced displacements* methods will be inappropriate, specially, when large stiffness modifications are foreseen in local model, or when local fine model stiffness is substantially different from the stiffness of local zone modelling used in the complete model.

To reiterate, the substructuring technique using superelements is exact for local modelling. That is to say, the rest of the structure is reduced to superelements, whilst local mesh and design changes are performed on a local part. These local analyses may refer to local stress analysis, local redesign iterations, local buckling analysis and local non-linearity analysis. Submodelling by substructuring using superelements is also **mandatory** when different **components of a whole model strategically and confidentially shared between several companies (i.e. aircrafts, engines and space structures).**

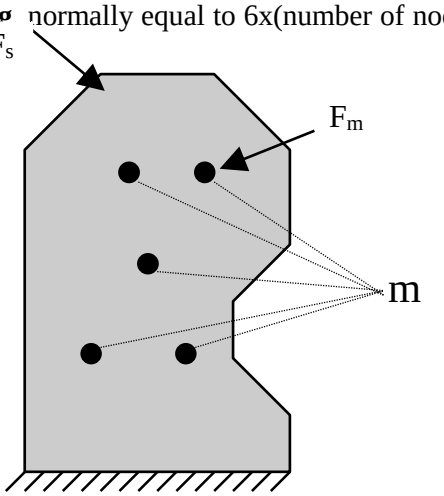
1.3.6.1 Submodelling by Substructuring – Static Condensation (Guyan Reduction) using Superelements

Very often, in a complicated structure, some sort of manual substructuring is required. Different parts of the structure will be analyzed separately. All the applied loads must eventually find their way to the constraints. For instance, in large structural engineering projects where a multiple story building structure is to be analyzed, the higher stories may be analyzed separately from that below. The higher story may incorporate a shear wall as a structural load-resisting element being rested upon say a beam in the story below. It will be incorrect to restrain rigidly all the supports of the higher story, i.e. the base of the shear wall must **not** be all supported on rigid supports (with the intention of applying the reaction forces onto the beam of the story below in the opposite direction). If this was done, the shear wall would not arch, but all the load would simply transfer straight down into the rigid supports. Hence, instead, the relative support stiffness under the shear wall must be incorporated, by say modelling the beam as well. The shear wall will then arch and the load distribution will be far more realistic with more of the load transferring directly to the supports of the shear wall. The reactions on these flexible supports (here the forces on the beam and the ends of the shear wall) can then be transferred onto the model containing the story below. If on the other hand, had all the load-resisting elements been columns supported on equal stiffness foundations, the supports of the upper story could well have been rigid and the reaction applied onto the story below. To model the boundaries of the local model with appropriate stiffness is substructuring. Here we have performed manual substructuring as the stiffness of the remaining part of the structure was approximated manually. To sum up, when manual substructuring is undertaken, the supports of a substructure should be modeled to incorporate their relative stiffness such as to obtain the correct distribution of reactions, which in turn will be applied (of course in the opposite direction – Newton's Third Law) onto the model containing the adjacent substructure. This of course is an approximate procedure as the substructuring is performed manually. Automatic substructuring using superelements to provide the stiffness at the boundaries of the local model is on the other hand exact.

[2] MATINEZ, Angel. *Submodelling Techniques for Static Analysis.* MSC. Software First South European Technology Conference, 7-9 June 2000.

Modelling the stiffness of the adjacent (to the substructure) elements is fundamental to substructure analysis. The static condensation technique does just that by reducing the equations to only involve the DOFs of the substructure and its interface, but in doing so incorporates the stiffness and internal forces of the remaining part of the whole model onto the interface DOFs. The most important characteristic of static condensation is that the boundary conditions at the interface of the substructure depends only on the stiffness and the loading of the remaining part of the whole model and is totally independent of the stiffness and loading of the submodel. Hence any modifications to the loading and stiffness of the submodel are perfectly acceptable and the answers will be exact. Thus, for static analysis, the static condensation technique produces the *exact same results* as that obtained from the full analysis.

Consider a finite element model of a structure that has been properly constrained and loaded. Total number of d.o.f.'s is denoted as **g**, normally equal to 6x(number of nodes). This **g** set can be subdivided in **c** (constrained) and **f** (free).



$$g \begin{cases} c \\ f \begin{cases} m \\ s \end{cases} \end{cases}$$

After applying constraint conditions, static analysis equations become in matrix form as follows:

$$[K_{ff}] \cdot \{\delta_f\} = \{F_f\} \qquad (1)$$

Free (**f**) d.o.f.'s are going to be subdivided in two sets: **m** (masters) and **s** (slaves). Masters d.o.f.'s are the ones to be kept in the reduction process, and slave set will be reduced. This partition gives expression (1) the following form:

$$\begin{bmatrix} K_{mm} & K_{ms} \\ K_{sm} & K_{ss} \end{bmatrix} \cdot \begin{Bmatrix} \delta_m \\ \delta_s \end{Bmatrix} = \begin{Bmatrix} F_m \\ F_s \end{Bmatrix} \qquad (2)$$

that results in these two equations:

$$\begin{aligned} [K_{mm}] \cdot \{\delta_m\} + [K_{ms}] \cdot \{\delta_s\} &= \{F_m\} \\ [K_{sm}] \cdot \{\delta_m\} + [K_{ss}] \cdot \{\delta_s\} &= \{F_s\} \end{aligned} \qquad (3)$$

Solving $\{\delta_s\}$ from second equation of (3) gives:

$$\boxed{\{\delta_s\} = [K_{ss}]^{-1}(\{F_s\} - [K_{sm}] \cdot \{\delta_m\})} \qquad (4)$$

This expression can be used for the recovery calculation of $\{\delta_s\}$, when $\{\delta_m\}$ were known. Considering last expression (4) in first equation of (3), and reordering terms, the following system of equations is obtained:

$$\left([K_{mm}] - [K_{ms}][K_{ss}]^{-1}[K_{sm}]\right) \cdot \{\delta_m\} = \{F_m\} - [K_{ms}][K_{ss}]^{-1}\{F_s\} \qquad (5)$$

that can be expressed in the compact form:

$$\boxed{[K^{*s}_{mm}] \cdot \{\delta_m\} = \{F^{*s}_m\}} \qquad (6)$$

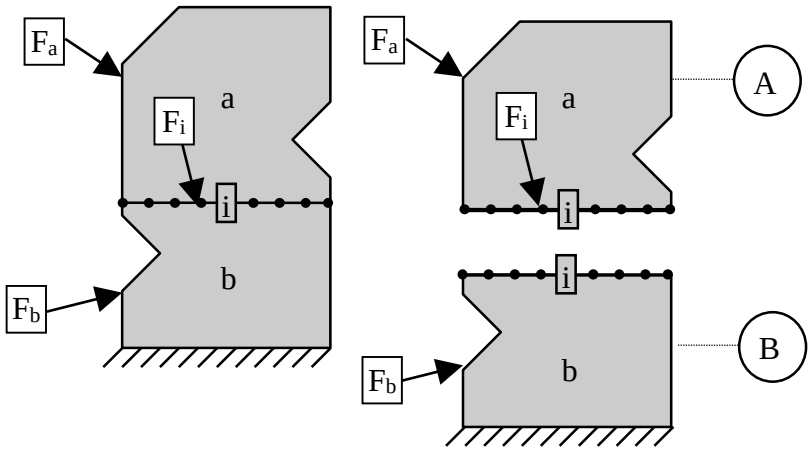
being:

$$\boxed{\begin{aligned}\left[K_{mm}^{*s}\right] &= \left[K_{mm}\right] - \left[K_{ms}\right]\left[K_{ss}\right]^{-1}\left[K_{sm}\right] \\ \left\{F_m^{*s}\right\} &= \left\{F_m\right\} - \left[K_{ms}\right]\left[K_{ss}\right]^{-1}\left\{F_s\right\}\end{aligned}} \qquad (7)$$

$\left[K_{mm}^{*s}\right]$ is named *reduced stiffness matrix* and $\left\{F_m^{*s}\right\}$ the *reduced load vector*. Equation (6) represents the reduced equations system to master d.o.f.'s set (m). Solving this system, displacements $\{\delta_m\}$ at masters are obtained. Expression (4) can be used then, to obtain displacements at slave d.o.f.'s (s).

Static condensation applied to linear static analysis gives an exact solution, equal to the solution of the global equation system (2). Consider a finite element model of a structure that is subdivided in two parts or substructures A and B. Finite element model can be subdivided in the following d.o.f.'s sets:

- a: internal d.o.f.'s of substructure A.
- b: internal d.o.f.'s of substructure B.
- i: interface d.o.f.'s between substructures A and B.



Stiffness matrices of each substructure take the form:

$$K_A = \begin{bmatrix} K_{aa} & K_{ai} \\ K_{ia} & K_{ii}^A \end{bmatrix} \qquad K_B = \begin{bmatrix} K_{ii}^B & K_{ib} \\ K_{bi} & K_{bb} \end{bmatrix} \qquad (8)$$

Assembling the stiffness matrices of each substructure results in the global system of equations for static analysis of the whole structure:

$$\begin{bmatrix} K_{aa} & K_{ai} & 0 \\ K_{ia} & K_{ii} & K_{ib} \\ 0 & K_{bi} & K_{bb} \end{bmatrix} \cdot \begin{Bmatrix} \delta_a \\ \delta_i \\ \delta_b \end{Bmatrix} = \begin{Bmatrix} F_a \\ F_i \\ F_b \end{Bmatrix} \qquad (9)$$

being: $K_{ii} = K_{ii}^A + K_{ii}^B \qquad (10)$

Static reduction is going to be applied to set (**b**) as slave d.o.f.'s and (**a+i**) as masters. Stiffness matrix and load vector are partitioned in the following way:

$$\left[\begin{array}{cc:c} K_{aa} & K_{ai} & 0 \\ K_{ia} & K_{ii} & K_{ib} \\ \hdashline 0 & K_{bi} & K_{bb} \end{array}\right] = \begin{bmatrix} K_{mm} & K_{ms} \\ K_{sm} & K_{ss} \end{bmatrix} \qquad \left\{\begin{array}{c} F_a \\ \hdashline F_i \\ F_b \end{array}\right\} = \begin{Bmatrix} F_m \\ F_s \end{Bmatrix} \qquad (11)$$

Recovering the expressions (7) for the calculation of reduced stiffness matrix and reduced load vector:

$$\begin{aligned}\left[K_{mm}^{*s}\right] &= \left[K_{mm}\right] - \left[K_{ms}\right]\left[K_{ss}\right]^{-1}\left[K_{sm}\right] \\ \left\{F_m^{*s}\right\} &= \left\{F_m\right\} - \left[K_{ms}\right]\left[K_{ss}\right]^{-1}\left\{F_s\right\}\end{aligned}$$

Applying these equations from established partitions in (11):

$$\left[K^{*b}_{(a+i)}\right]=\begin{bmatrix}K_{aa} & K_{ai}\\ K_{ia} & K_{ii}\end{bmatrix}-\begin{bmatrix}0\\ K_{ib}\end{bmatrix}\left[K_{bb}\right]^{-1}\begin{bmatrix}0 & K_{bi}\end{bmatrix}$$
$$\left\{F^{*b}_{(a+i)}\right\}=\begin{Bmatrix}F_a\\ F_i\end{Bmatrix}-\begin{bmatrix}0\\ K_{ib}\end{bmatrix}\left[K_{bb}\right]^{-1}\left\{F_b\right\} \qquad (12)$$

that result in:

$$\left[K^{*b}_{(a+i)}\right]=\begin{bmatrix}K_{aa} & K_{ai}\\ K_{ia} & K_{ii}-K_{ib}\cdot K_{bb}^{-1}\cdot K_{bi}\end{bmatrix}$$
$$\left\{F^{*b}_{(a+i)}\right\}=\begin{Bmatrix}F_a\\ F_i-K_{ib}\cdot K_{bb}^{-1}\cdot F_b\end{Bmatrix} \qquad (13)$$

Decomposing each matrix as sum of two terms:

$$\left[K^{*b}_{(a+i)}\right]=\begin{bmatrix}K_{aa} & K_{ai}\\ K_{ia} & K^{A}_{ii}\end{bmatrix}+\begin{bmatrix}0 & 0\\ 0 & K^{B}_{ii}-K_{ib}\cdot K_{bb}^{-1}\cdot K_{bi}\end{bmatrix}=K_A+\overline{K^{*b}_{ii}}$$
$$\left\{F^{*b}_{(a+i)}\right\}=\begin{Bmatrix}F_a\\ 0\end{Bmatrix}+\begin{Bmatrix}0\\ F_i-K_{ib}\cdot K_{bb}^{-1}\cdot F_b\end{Bmatrix}=\begin{Bmatrix}F_a\\ 0\end{Bmatrix}+\overline{F^{*b}_{i}} \qquad (14)$$

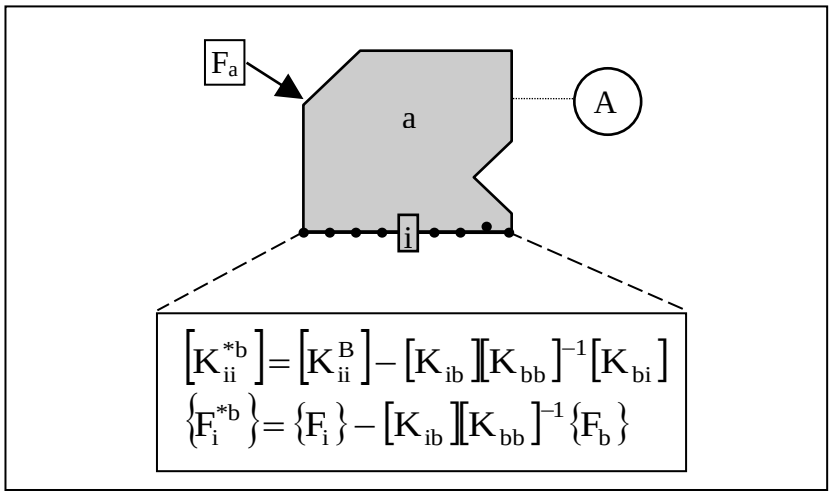
The reduced equations systems to d.o.f.'s set (a+i) has the following form:

$$\left(K_A+\overline{K^{*b}_{ii}}\right)\cdot\begin{Bmatrix}\delta_a\\ \delta_i\end{Bmatrix}=\begin{Bmatrix}F_a\\ 0\end{Bmatrix}+\overline{F^{*b}_{i}} \qquad (15)$$

$$\left[K^{*b}_{(a+i)}\right]\cdot\begin{Bmatrix}\delta_a\\ \delta_i\end{Bmatrix}=\left\{F^{*b}_{(a+i)}\right\} \qquad (16)$$

Reduced stiffness matrix is composed of the stiffness matrix of substructure A and the reduced stiffness matrix of substructure B to the interface d.o.f. (set i) expanded (completed with zeros) to (a+i) size. In the same way, reduced load vector is the applied load to internal d.o.f.'s of substructure A (set a) plus the reduced load vector of substructure B to set i, and expanded, again, to (a+i) size. In other words, to perform a local static analysis of substructure A by static condensation technique, the following items are needed:

- Finite element model of substructure A.
- The effect of substructure B over A at interface d.o.f.'s:
- Reduced stiffness matrix of substructure B to interface d.o.f.'s (set i), and
- Reduced load vector of substructure B to interface d.o.f.'s (set i)



Static Condensation Submodelling Method for Substructure A

The most important characteristic of submodelling analysis by substructuring using static condensation is that the boundary conditions at the interface (reduced model of substructure B) only depends on the stiffness of substructure B, and is totally independent of the stiffness of substructure A. Then, any modification of substructure A will be treated correctly, and the results will be the same than the ones obtained with the complete model.

Let us discuss in terms of a-sets and o-sets. Displacement vector sets in NASTRAN define the equations of motion and are partitioned in the following ways.

(i) g-set is the unconstrained set of structural equations

(ii) the g-set is partitioned into the m-set pertaining to the dependent set of DOFs and the n-set pertaining to the independent set of DOFs

g-set – m-set = n-set

(iii) the n-set is then partitioned into s-set pertaining to the constrained DOFs and the f-set pertaining to the unconstrained DOFs

n-set – s-set = f-set

(iv) the f-set is partitioned into the o-set pertaining to the DOFs eliminated by static condensation and the a-set pertaining to the a-set (analysis set) pertaining to the DOFs not eliminated by static condensation

f-set – o-set = a-set

In the static condensation process you select a set of dynamic DOFs called the a-set; these are the retained DOFs that form the analysis set. The complementary set is called the o-set and is the set of DOFs that are omitted from the dynamic analysis through a reduction process. The process distributes the o-set mass, stiffness, and damping to the a-set DOFs by using a transformation that is based on a partition of the stiffness matrix (hence the term static condensation). This reduction process is **exact only for static analysis**, but leads to **approximations in the dynamic response**. The a-set DOFs are defined by the ASET or ASET1 Bulk Data entries, and the o-set DOFs are defined by the OMIT or OMIT1 Bulk Data entries.

If you start with the stiffness equation in terms of the set of the unconstrained (free) structural coordinates, you have

$$[K_{ff}]\{u_f\} = \{P_f\}$$

Partitioning the free degrees of freedom into two subsets of the *f*-set, you obtain

$$\{u_f\} = \begin{Bmatrix} u_a \\ \hline u_o \end{Bmatrix} \quad \begin{aligned} u_a &= \text{the } a\text{-set variables} \\ u_o &= \text{the omitted set of variables, which are removed by static condensation} \end{aligned}$$

Rewriting

$$[K_{ff}][u_f] = \left[\begin{array}{c:c} \overline{K_{aa}} & K_{ao} \\ \hline K_{oa} & K_{oo} \end{array}\right] \begin{Bmatrix} u_a \\ \hline u_o \end{Bmatrix} = \begin{Bmatrix} \overline{P_a} \\ \hline P_0 \end{Bmatrix}$$

On expansion of top equation

$$\{u_o\} = [G_{oa}]\{u_a\} + \{u_{oo}\}$$

$$[G_{oa}] = -[K_{oo}]^{-1}[K_{oa}]$$

$$\{u_{oo}\} = [K_{oo}]^{-1}[P_o]$$

On expansion of bottom equation

$$\{u_a\} = [K_{aa}]^{-1}\{P_a\}$$

$$[K_{aa}] = [\overline{K_{aa}} + K_{ao}G_{oa}]$$

$$\{P_a\} = [\overline{P_a} + G_{oa}^T P_o]$$

First, the f-set is reduced to the a-set using Guyan reduction. Second, the analysis set $\{u_a\}$ is solved for. Third, the omitted set $\{u_o\}$ is solved for. In **static analysis**, the results using static condensation are numerically **exact**. The partitioned solution merely changes the order of the operations of the unpartitioned solution. Guyan reduction has special applications in dynamic analysis. In **dynamic analysis**, the reduction is **approximate**; the term $\{u_{OO}\}$ is ignored in this case. The reduction is based solely on static transformation **BUT** is **EXACT** provided that **no loads are applied to the *o*-set** degrees of freedom.

To utilize static condensation, you can choose either the ASET/ASET1 or the OMIT/OMIT1 entries. With these entries you should specify only the *a*-set (with ASET/ASET1 entries) or *o*-set (with OMIT/OMIT1 entries) degrees of freedom. The unspecified remaining *f*-set DOFs are automatically placed in the complementary set. However, if you specify both the *a*-set and *o*-set DOFs, then the unspecified remaining *f*-set DOFs are automatically placed in the *o*-set. The same DOF cannot be specified on both the *a*-set and *o*-set; otherwise, the job fails with UFM 2101A. Since the reduction process is performed on an individual degree of freedom, it is possible to have some of the degrees of freedom at a grid point in the *a*-set and other degrees of freedom at a grid point in one of the other mutually exclusive sets. No additional user input is required.

$ BULK DATA									
ASET	**G1**	**C1**	**G2**	**C2**	**G3**	**C3**	**G4**	**C4**	
ASET1	**C**	**G1**	**G2**	**G3**	**G4**	**G5**	**G6**	**G7**	
	G8	**G9**	**G10**	**G11**	**…etc…**				
ASET1	**C**	**G1**	**THRU**	**Gi**					

$ BULK DATA									
OMIT	**G1**	**C1**	**G2**	**C2**	**G3**	**C3**	**G4**	**C4**	
OMIT1	**C**	**G1**	**G2**	**G3**	**G4**	**G5**	**G6**	**G7**	
	G8	**G9**	**G10**	**G11**	**…etc…**				
OMIT1	**C**	**G1**	**THRU**	**Gi**					

The choice of whether to use the ASET/ASET1 or OMIT/OMIT1 is really a matter of convenience.

1.3.6.2 Submodelling – Boundary Internal Force Method

Other way to perform local analysis of substructure A is isolating it and considering internal loads from substructure B to A, as applied loads at interface (set i). Equations system that reflects this approach can be derived reordering terms in expression (15):

$$\begin{bmatrix} K_{aa} & K_{ai} \\ K_{ia} & K_{ii}^{A} \end{bmatrix} \cdot \begin{Bmatrix} \delta_a \\ \delta_i \end{Bmatrix} = \begin{Bmatrix} F_a \\ F_i \end{Bmatrix} + \begin{Bmatrix} 0 \\ -\left(K_{ii}^{B} - K_{ib} \cdot K_{bb}^{-1} \cdot K_{bi}\right) \cdot \delta_i - K_{ib} \cdot K_{bb}^{-1} \cdot F_b \end{Bmatrix}$$
$$\begin{bmatrix} K_{aa} & K_{ai} \\ K_{ia} & K_{ii}^{A} \end{bmatrix} \cdot \begin{Bmatrix} \delta_a \\ \delta_i \end{Bmatrix} = \begin{Bmatrix} F_a \\ F_i \end{Bmatrix} + F_{BA} \qquad (17)$$

F_{BA} being the internal loads from substructure B to substructure A. These **internal loads** are the **sum of two terms**:

- F'_{BA} : Action forces over interface when substructure B is constrained (with zero displacement) at interface d.o.f.'s and loaded with its applied load F_b:

$$\begin{bmatrix} K_{ii}^{B} & K_{ib} \\ K_{bi} & K_{bb} \end{bmatrix} \cdot \begin{Bmatrix} 0 \\ \delta'_b \end{Bmatrix} = \begin{Bmatrix} -F'_{BA} \\ F_b \end{Bmatrix}$$
$$\delta'_b = K_{bb}^{-1} \cdot F_b \qquad (18)$$
$$F'_{BA} = -K_{ib} \cdot K_{bb}^{-1} \cdot F_b$$

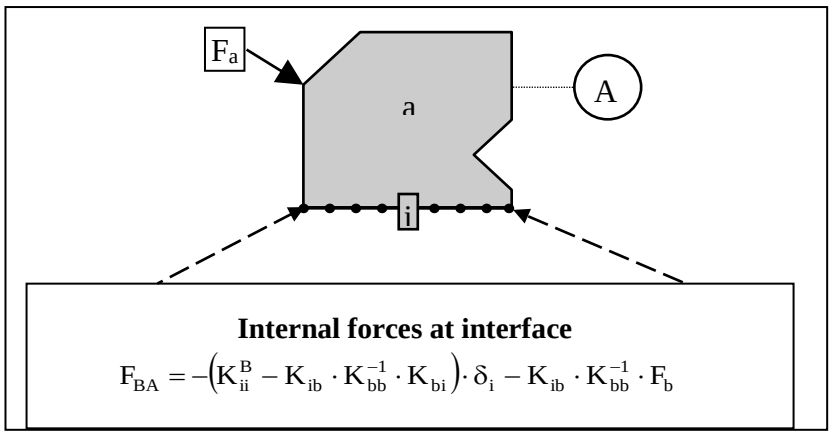
- F''_{BA} : Action forces over interface when substructure B is enforced at interface with prescribed displacements δ_i:

$$\begin{bmatrix} K_{ii}^{B} & K_{ib} \\ K_{bi} & K_{bb} \end{bmatrix} \cdot \begin{Bmatrix} \delta_i \\ \delta''_b \end{Bmatrix} = \begin{Bmatrix} -F''_{BA} \\ 0 \end{Bmatrix}$$
$$\delta''_b = -K_{bb}^{-1} \cdot K_{bi} \cdot \delta_i \qquad (19)$$
$$F''_{BA} = -(K_{ii}^{B} - K_{ib} \cdot K_{bb}^{-1} \cdot K_{bi}) \cdot \delta_i$$

This term is dependent on displacements at interface d.o.f.'s (set i), and, indirectly, dependent on stiffness changes of substructure A. This means that any stiffness modification in substructure A will imply a perturbation of displacements δ_i, and then, changes in internal forces F_{BA}. This lead to an important conclusion: there is an inherent inaccuracy in local analysis of substructure A, using *internal forces method* when stiffness changes are performed in this substructure A.

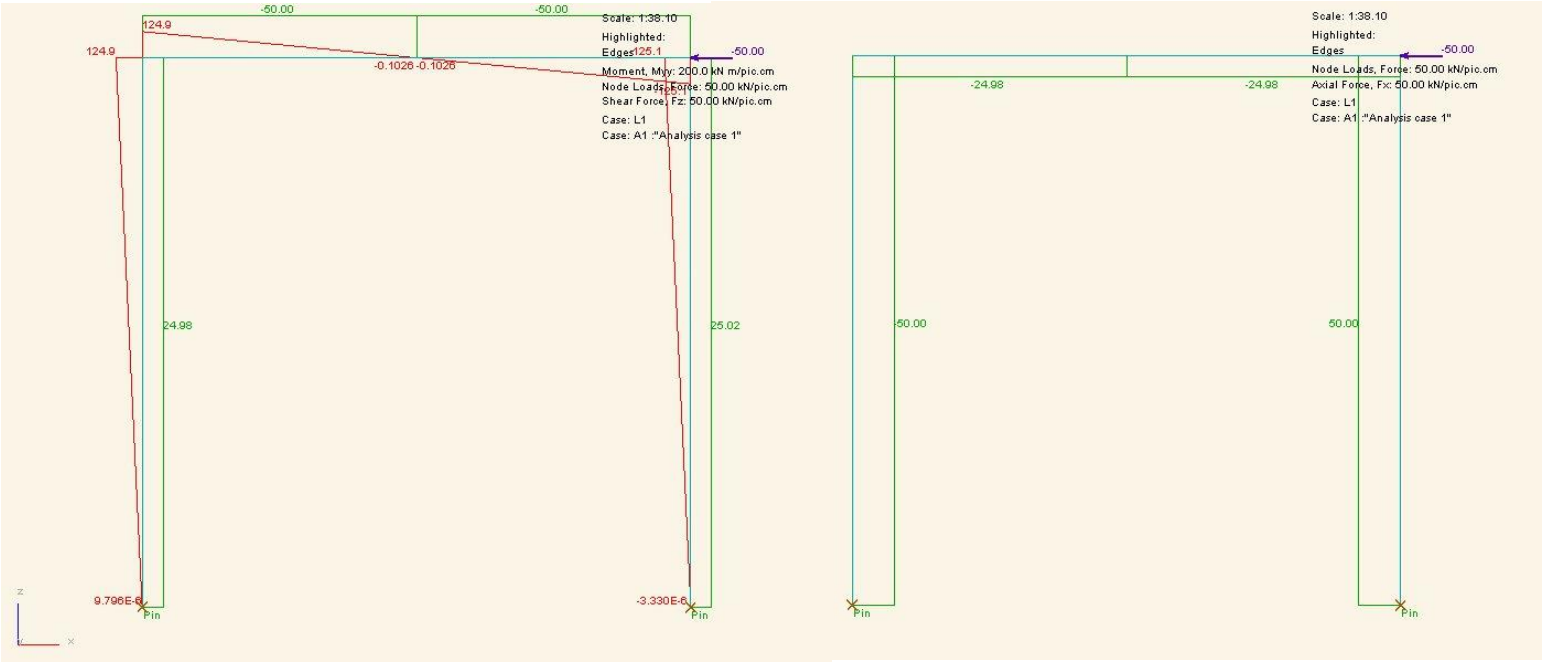
As can be seen, it is demonstrated that $F_{BA} = F'_{BA} + F''_{BA}$ (20)

In this internal forces approach, it is possible that substructure A has not sufficient constraint conditions to avoid rigid body motions or local mechanism problems. In this case, **additional constraints** are needed **to remove these singularities** from stiffness matrix, but their **reactions** should be equal to **zero**. In the case that substructure A has no external constraints, an isostatic (statically determinate) set of constraints should be applied. Reactions at these constraints **will be zero** (or close to zero if the **geometry is accurate** and there are no significant Poisson effects along the constrained boundary in 2D and 3D models) because F_{BA}, F_a and F_i loads should be in static equilibrium.

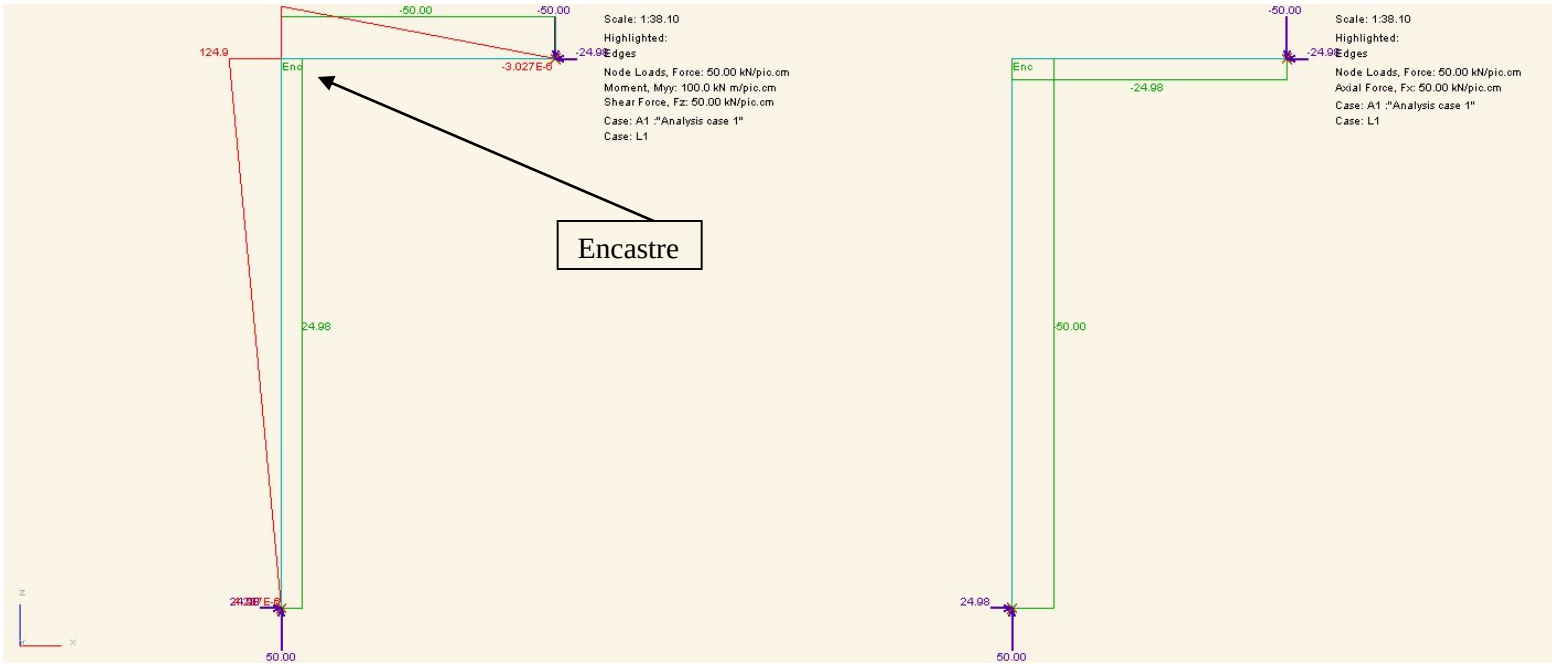


Internal Forces Submodelling Method for Substructure A

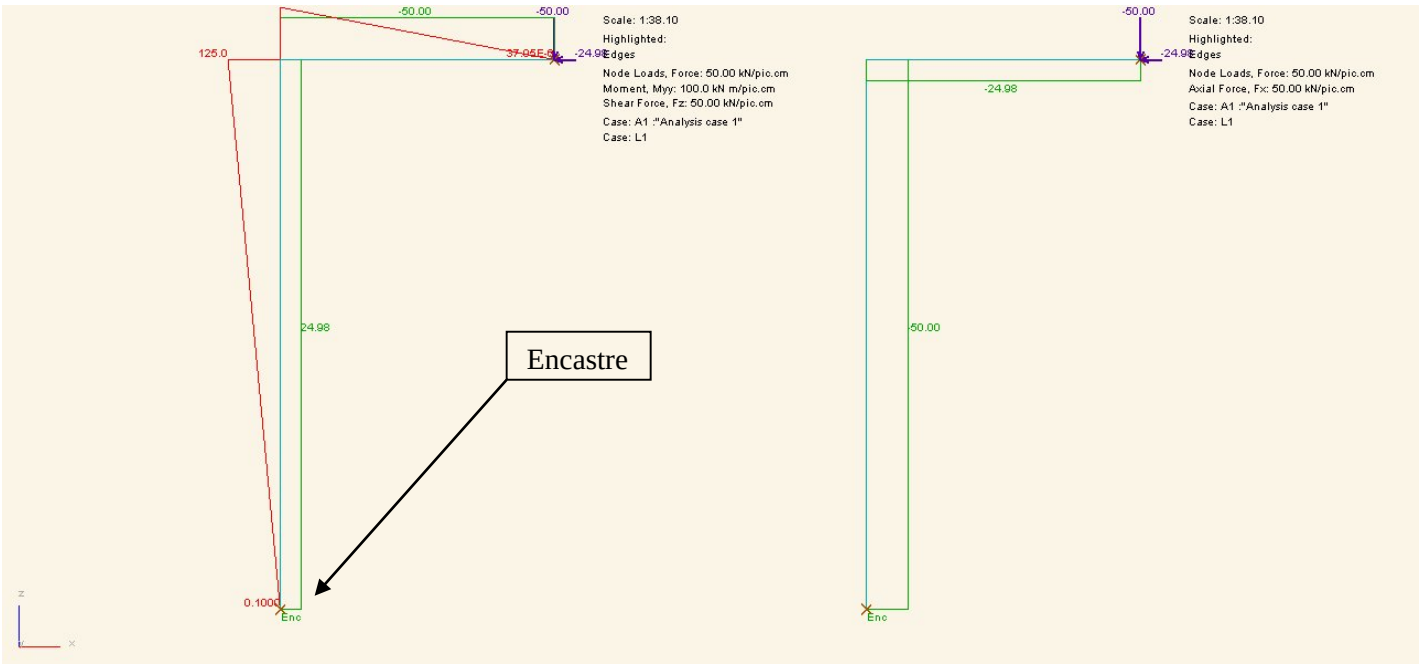
Consider the following pinned base portal frame to submodel. The whole model BMD, SFD and AFD is presented.



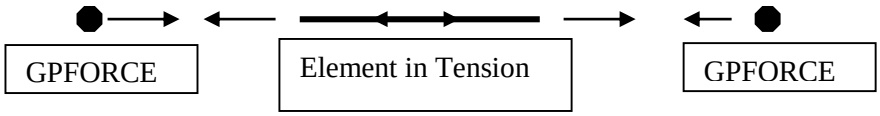
To submodel half the frame and produce the same internal effects, we need to apply the GPFORCE (in opposite direction) at the interfaces and ensure by applying constraints that the submodel is statically determinate.



The reactions at the artificial encastre support will be found to be **ALL ZERO** because the free-body is in static equilibrium from the applied forces. Even if the chosen encastre node been the base i.e. the interface, the correct effects will be obtained. The moment reaction at the base will be found to be **ZERO** as the translational restraints in X and Z equilibrate the applied forces. This method is of course accurate here since the stiffness of the local model has not been altered. If the **local model stiffness had been altered, the boundary force method will be inexact. From a practical viewpoint, this is because the stiffness change causes forces to distribute differently. For instance, if the local model stiffness is reduced dramatically, more of the force will distribute to stiffer parts of the structure and carried into the ground. A special case occurs when the remaining part of the structure is isostatically mounted upon the local model (here if the portal frame was a cantilever), in which case changes in stiffness of the local model will still produce exact results as that of the global model.**



In order to obtain the forces acting upon an element **(be it a beam, shell or solid)**, the (global coordinate) **grid point forces** on the grid for the relevant element is used. Do not confuse grid point stresses with grid point forces. They differ totally in concept. Grid point stresses are the average (global coordinate) stress values at a grid due to element stresses in the **(shell and solid)** elements adjacent to the grid. Hence, direct (say positive tensile) global coordinate stresses in X in adjacent elements will be averaged to yield a positive tensile global stress at the grid point. The grid stress can be thought of in the same way as the element stress, in the sense that that is the element (global coordinate) stress at that location had there been an element there at the grid. Grid point forces on the other hand are not average forces from adjacent elements. The grid point force balance provides insight into how the loads at a grid are distributed amongst its adjoining elements i.e. it provides information of the ***load path***. The specification of ***a grid point forces must always be associated with an element (or applied load or reaction)***. Grid point forces are the global coordinate force acting on a grid due to an adjacent **(beam, shell or solid)** element. It is **NOT THE FORCE ACTING ON THE ELEMENT, BUT ON THE GRID**. If all the elements attached to a grid are considered, then the grid point force should theoretically be zero for equilibrium at the grid connection, hence the *grid point force balance*. Grid point force balance is computed only from linear stiffness elements, the sum of applied loads and thermal loads, and SPC forces. Since GPFORCE is the force acting on the grid (or the structural joint between adjacent elements), it is opposite to the direction of the force acting on the element. Hence, the direction of the GPFORCE from the applied load is the same as the direction of load application, the direction of the GPFORCE from the SPC reaction is the same as the direction of the reaction and the direction of the GPFORCE from an element is opposite the force that acts upon the element.



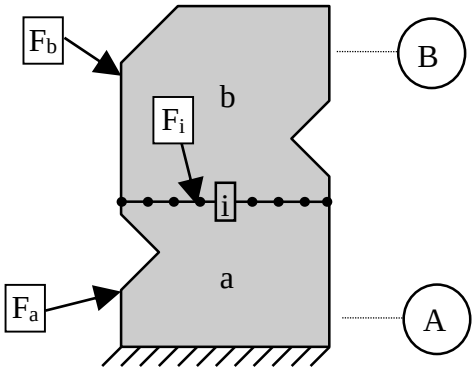
Effects NOT accounted for include those from mass elements in dynamic analysis (inertia loads), rigid elements and MPCs (MPCFORCES), general elements, DMIG entries, and boundary loads from upstream superelements. These effects may lead to an apparent lack of equilibrium at the grid point level. The ***global coordinate*** grid point forces ***for each relevant entity (be they beams, shells, solids, applied loads or SPCs) acting on the grid*** at ***each grid*** is obtained from GPFORCE (PRINT) as follows.

G R I D P O I N T F O R C E B A L A N C E

POINT-ID	ELEMENT-ID	SOURCE	T1	T2	T3	R1	R2	R3
1		F-OF-SPC	1.695155E+03	6.360602E+02	1.951741E+01	8.563783E+01	-2.406043E+01	.0
1	1	QUAD4	-1.695155E+03	-6.360602E+02	-1.951741E+01	-8.563783E+01	2.406043E+01	.0
1		*TOTALS*	.0	.0	3.552714E-15	1.421085E-14	-3.552714E-15	.0
16		APP-LOAD	.0	.0	-5.000000E+01	.0	.0	.0
16	9	QUAD4	2.966625E-13	3.832166E-14	5.000000E+01	-1.397169E-12	1.610404E-12	.0
16		*TOTALS*	2.966625E-13	3.832166E-14	8.171241E-13	-1.397169E-12	1.610404E-12	.0

1.3.6.3 Submodelling – Boundary Internal Force Method of Substructure Isostatically Mounted Over Rest of Structure

Suppose a structure composed of two substructures: A and B; substructure B is **isostatically mounted** over substructure A and **has not any external constraints**. That is, interface set is a congruent set of six d.o.f.'s that removes rigid body motions of substructure B.



Internal Forces Submodelling Method for Substructure A

Interface forces F_{BA} can be obtained directly from static equilibrium equations and do not depend on stiffness of substructure A or B.

In this case, static analysis of substructure B can be performed simply considering constrained to zero its six interface d.o.f.'s:

$$\begin{bmatrix} K_{ii}^{B} & K_{ib} \\ K_{bi} & K_{bb} \end{bmatrix} \cdot \begin{Bmatrix} 0 \\ \delta_b \end{Bmatrix} = \begin{Bmatrix} -F_{BA} \\ F_b \end{Bmatrix} \qquad (23)$$

$$F_{BA} = -K_{ib} \cdot K_{bb}^{-1} \cdot F_b = \varphi_{ib} \cdot F_b \qquad (24)$$

Internal forces (actions over interface) F_{BA} are independent on substructure B stiffness: φ_{ib} matrix is the rigid body modes matrix.

This rigid body matrix can be obtained by means of a set of six enforcing displacements conditions that in matrix form corresponds to a unit matrix I:

$$\begin{bmatrix} K_{ii}^{B} & K_{ib} \\ K_{bi} & K_{bb} \end{bmatrix} \cdot \begin{Bmatrix} I_{ii} \\ \varphi_{ib} \end{Bmatrix} = \begin{Bmatrix} T_{ii} \\ 0 \end{Bmatrix} \qquad (25)$$

From this equation, it is demonstrated that displacements at internal d.o.f.'s of substructure B correspond to rigid body modes matrix: $\varphi_{ib} = -K_{ib} \cdot K_{bb}^{-1}$. (26)

Other important consideration of equations system (25), is that, by definition, elastic forces T_{ii} necessary to perform rigid body motions should be zero. From this expression (25) the calculation of elastic forces is as follows:

$$T_{ii} = K_{ii}^{B} - K_{ib} \cdot K_{bb}^{-1} \cdot K_{bi} = [0] = K_{ii}^{*b} \qquad (27)$$

Last expression states an important conclusion: reduced stiffness matrix of substructure B to an isostatic interface is zero. In order words, just in this case in which interface between substructures A and B is isostatic, local analysis (with stiffness modifications) of substructure A can be exactly performed by static condensation or internal forces methods, because these approaches are now identical.

1.3.6.4 Boundary Enforced Displacement Method

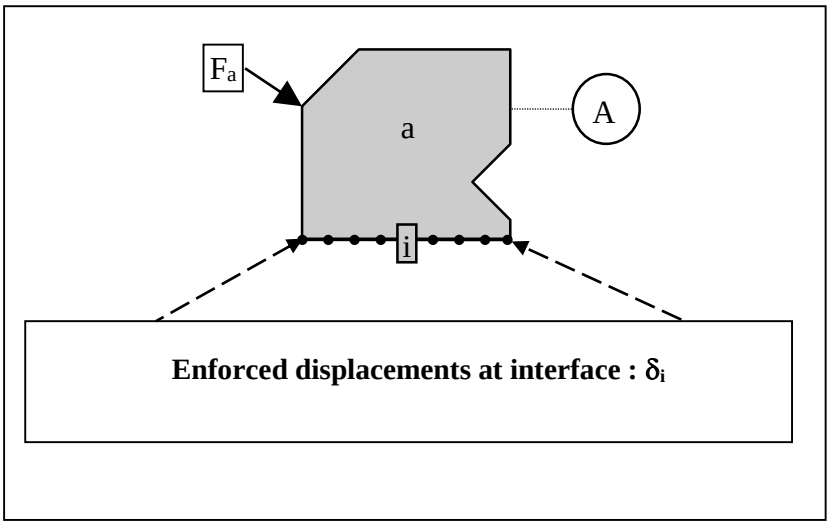
Other submodelling technique to analyse locally substructure A consists in enforce displacements δ_i at interface d.o.f.'s (set i), considering also the applied forces F_a at the internal d.o.f.'s (set a).

$$\begin{bmatrix} K_{aa} & K_{ai} \\ K_{ia} & K_{ii}^{A} \end{bmatrix} \cdot \begin{Bmatrix} \delta_a \\ \delta_i \end{Bmatrix} = \begin{Bmatrix} F_a \\ R_i \end{Bmatrix} \qquad (21)$$

First equations system from above expression gives the static solution for substructure A:

$$K_{aa} \cdot \delta_a = \left(F_a - K_{ai} \cdot \delta_i\right) \qquad (22)$$

External force term in expression (22) is dependent on displacements at interface d.o.f.'s (set i), and, on stiffness changes of substructure A (K_{ai}). This means that any stiffness modification in substructure A will imply a perturbation of this external force term. This lead to the same important conclusion again: there is an inherent inaccuracy in local analysis of substructure A, using *enforced displacements method* when stiffness changes are performed in this substructure A.

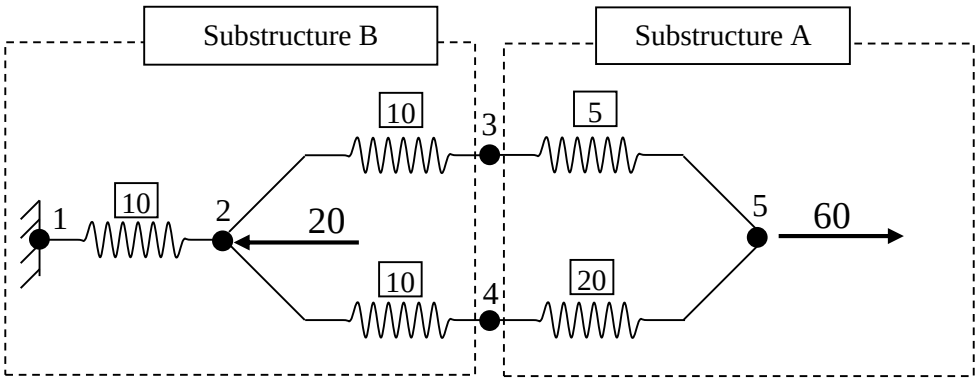


Enforced Displacements Submodelling Method for Substructure A

1.3.6.5 Numerical Example of Submodelling

1.3.6.5.1 Example 1 – No Local (Substructure A) Stiffness Modification

To emphasize the application of submodelling methods, herein it is presented a simple example of structure divided in two substructures. **Substructure to be locally analyzed is A**.



Global structure analysis is performed and solved with the following system of equations:

$$\begin{bmatrix} 30 & -10 & -10 & 0 \\ -10 & 15 & 0 & -5 \\ -10 & 0 & 30 & -20 \\ 0 & -5 & -20 & 25 \end{bmatrix} \cdot \begin{Bmatrix} \delta_2 \\ \delta_3 \\ \delta_4 \\ \delta_5 \end{Bmatrix} = \begin{Bmatrix} -20 \\ 0 \\ 0 \\ 60 \end{Bmatrix} \Rightarrow \begin{Bmatrix} \delta_2 \\ \delta_3 \\ \delta_4 \\ \delta_5 \end{Bmatrix} = \begin{Bmatrix} 4 \\ 6 \\ 8 \\ 10 \end{Bmatrix}$$

Example 1 – No Local (Substructure A) Stiffness Modification: Substructuring Using Static Condensation

Performing static condensation of substructure B:

$$K_B = \begin{bmatrix} K_{bb} & K_{bi} \\ K_{ib} & K_{ii}^B \end{bmatrix} = \left[\begin{array}{cc|c} 30 & -10 & -10 \\ -10 & 10 & 0 \\ \hline -10 & 0 & 10 \end{array}\right] \begin{matrix} \\ (2) \\ (3) \\ (4) \end{matrix}$$

$$\left[K_{ii}^{*b}\right] = \left[K_{ii}^{B}\right] - \left[K_{ib}\right]\left[K_{bb}\right]^{-1}\left[K_{bi}\right]$$

$$\left\{F_i^{*b}\right\} = \left\{F_i\right\} - \left[K_{ib}\right]\left[K_{bb}\right]^{-1}\left\{F_b\right\}$$

$$\left[K_{ii}^{*b}\right] = \begin{bmatrix} 10 & 0 \\ 0 & 10 \end{bmatrix} - \begin{bmatrix} -10 \\ -10 \end{bmatrix} \cdot \frac{1}{30} \cdot \begin{bmatrix} -10 & -10 \end{bmatrix} = 10/3 \begin{bmatrix} 2 & -1 \\ -1 & 2 \end{bmatrix} \begin{matrix} (3) \\ (4) \end{matrix}$$

$$\left\{F_i^{*b}\right\} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix} - \begin{bmatrix} -10 \\ -10 \end{bmatrix} \cdot \frac{1}{30} \cdot \{-20\} = -20/3 \begin{Bmatrix} 1 \\ 1 \end{Bmatrix} \begin{matrix} (3) \\ (4) \end{matrix}$$

Now, local analysis of substructure A, considering static condensation of B, is performed:

$$\left(\begin{bmatrix} K_{ii}^A & K_{ia} \\ K_{ai} & K_{aa} \end{bmatrix} + \begin{bmatrix} K_{ii}^{*b} & 0 \\ 0 & 0 \end{bmatrix} \right) \begin{Bmatrix} \delta_i \\ \delta_a \end{Bmatrix} = \begin{Bmatrix} 0 \\ F_a \end{Bmatrix} + \begin{Bmatrix} F_i^{*b} \\ 0 \end{Bmatrix}$$

Establishing system of equations and solving:

$$\left(\left[\begin{array}{cc|c} 5 & 0 & -5 \\ 0 & 20 & -20 \\ \hline -5 & -20 & 25 \end{array}\right] + \frac{10}{3} \left[\begin{array}{cc|c} 2 & -1 & 0 \\ -1 & 2 & 0 \\ \hline 0 & 0 & 0 \end{array}\right] \right) \begin{Bmatrix} \delta_3 \\ \delta_4 \\ \delta_5 \end{Bmatrix} = \left\{\begin{array}{c} 0 \\ 0 \\ \hline 60 \end{array}\right\} - \frac{20}{3} \left\{\begin{array}{c} 1 \\ 1 \\ \hline 0 \end{array}\right\} \Rightarrow \begin{Bmatrix} \delta_3 \\ \delta_4 \\ \delta_5 \end{Bmatrix} = \begin{Bmatrix} 6 \\ 8 \\ 10 \end{Bmatrix}$$

Displacement results for substructure A are the same than those obtained from global structure as expected.

Example 1 – No Local (Substructure A) Stiffness Modification: Internal Forces Method

$$\begin{bmatrix} K_{ii}^{A} & K_{ia} \\ K_{ai} & K_{aa} \end{bmatrix} \begin{Bmatrix} \delta_i \\ \delta_a \end{Bmatrix} = \begin{Bmatrix} 0 \\ F_a \end{Bmatrix} + \begin{Bmatrix} -K_{ii}^{*b} \cdot \delta_i + F_i^{*b} \\ 0 \end{Bmatrix}$$

$$-K_{ii}^{*b} \cdot \delta_i + F_i^{*b} = -10/3 \begin{bmatrix} 2 & -1 \\ -1 & 2 \end{bmatrix} \begin{Bmatrix} 6 \\ 8 \end{Bmatrix} + \left(-20/3\right) \begin{Bmatrix} 1 \\ 1 \end{Bmatrix} = \begin{Bmatrix} -20 \\ -40 \end{Bmatrix}$$

$$\left[\begin{array}{cc:c} 5 & 0 & -5 \\ 0 & 20 & -20 \\ \hdashline -5 & -20 & 25 \end{array}\right] \begin{Bmatrix} \delta_3 \\ \delta_4 \\ \hdashline \delta_5 \end{Bmatrix} = \begin{Bmatrix} -20 \\ -40 \\ \hdashline 60 \end{Bmatrix}$$

Above system of equations has a singular stiffness matrix and cannot be solved. A restraint condition is needed: considering $\delta_4 = 8$, the rest of displacements can be calculated, resulting in:

$$\begin{Bmatrix} \delta_3 \\ \delta_4 \\ \delta_5 \end{Bmatrix} = \begin{Bmatrix} 6 \\ 8 \\ 10 \end{Bmatrix}$$

that are, again, **the expected displacement results in substructure A**.

Example 1 – No Local (Substructure A) Stiffness Modification: Enforced Displacement Method

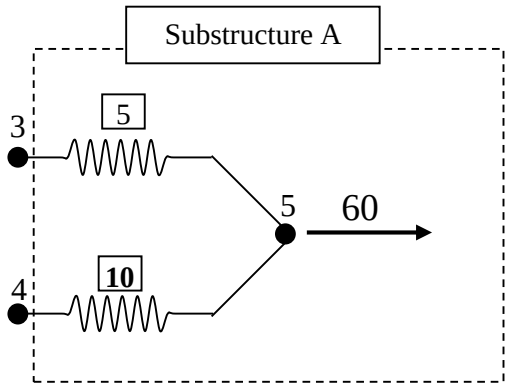
$$K_{aa} \cdot \delta_a = \left(F_a - K_{ai} \cdot \delta_i\right)$$

$$25 \cdot \delta_5 = 60 - \begin{bmatrix} -5 & -20 \end{bmatrix} \begin{Bmatrix} 6 \\ 8 \end{Bmatrix}$$

That gives $\delta_5 = 10$, **the same expected results**.

1.3.6.5.2 Example 2 – Local (Substructure A) Stiffness Modification

Suppose that substructure A is changed to the following configuration:



Global structure analysis is performed and solved with the following system of equations:

$$\begin{bmatrix} 30 & -10 & -10 & 0 \\ -10 & 15 & 0 & -5 \\ -10 & 0 & 20 & -10 \\ 0 & -5 & -10 & 15 \end{bmatrix} \cdot \begin{Bmatrix} \delta_2 \\ \delta_3 \\ \delta_4 \\ \delta_5 \end{Bmatrix} = \begin{Bmatrix} -20 \\ 0 \\ 0 \\ 60 \end{Bmatrix} \Rightarrow \begin{Bmatrix} \delta_2 \\ \delta_3 \\ \delta_4 \\ \delta_5 \end{Bmatrix} = \begin{Bmatrix} 4 \\ 6.4 \\ 7.6 \\ 11.2 \end{Bmatrix}$$

Example 2 – Local (Substructure A) Stiffness Modification: Substructuring Using Static Condensation

Local analysis of substructure A considering static condensation of B

$$\left(\begin{bmatrix} K_{ii}^{A} & K_{ia} \\ K_{ai} & K_{aa} \end{bmatrix} + \begin{bmatrix} K_{ii}^{*b} & 0 \\ 0 & 0 \end{bmatrix} \right) \begin{Bmatrix} \delta_i \\ \delta_a \end{Bmatrix} = \begin{Bmatrix} 0 \\ F_a \end{Bmatrix} + \begin{Bmatrix} F_i^{*b} \\ 0 \end{Bmatrix}$$

Establishing system of equations and solving:

$$\left(\left[\begin{array}{cc:c} 5 & 0 & -5 \\ 0 & 10 & -10 \\ \hdashline -5 & -10 & 15 \end{array} \right] + \frac{10}{3} \left[\begin{array}{cc:c} 2 & -1 & 0 \\ -1 & 2 & 0 \\ \hdashline 0 & 0 & 0 \end{array} \right] \right) \begin{Bmatrix} \delta_3 \\ \delta_4 \\ \delta_5 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ \hdashline 60 \end{Bmatrix} - \frac{20}{3} \begin{Bmatrix} 1 \\ 1 \\ \hdashline 0 \end{Bmatrix} \Rightarrow \begin{Bmatrix} \delta_3 \\ \delta_4 \\ \delta_5 \end{Bmatrix} = \begin{Bmatrix} 6.4 \\ 7.6 \\ 11.2 \end{Bmatrix}$$

Displacement results for substructure A **are exactly the same** than those obtained from global structure as expected.

We shall demonstrate a worked example using static condensation with MSC/NASTRAN superelement capabilities. First, **perform static condensation run of substructure B**, using DMIGOP2 option.

```
SOL 101
TIME 5
DIAG 14
CEND
TITLE = DMIGOP2, RUN 1
PARAM,EXTOUT,DMIGOP2
SPC=1
LOAD=1
BEGIN BULK
ASET1       1     3           4
GRID        1                 0.     0.     0.
GRID        2                 1.     0.     0.
GRID        3                 2.     1.     0.
GRID        4                 2.     0.     0.
```

```
CELAS2      1     10.         1     1     2     1
CELAS2      2     10.         2     1     3     1
CELAS2      3     10.         2     1     4     1
SPC1        1     123456      1
SPC1        1     23456       2
SPC1        1     23456       3
SPC1        1     23456       4
FORCE       1     2                 -20.  1.    0.    0.
ENDDATA
```

From this run a fort.30 file is output with binary output2 format, containing reduced stiffness matrix and reduced load vector at interface d.o.f.'s declared in ASET1 card. Next perform **local analysis of substructure A.**

```
ASSIGN INPUTT2='fort.30',UNIT=30
SOL 101
TIME 5
CEND
TITLE = DMIGOP2, RUN 2
SPC=1
SUBCASE 1
 SUPER=10,1
SUBCASE 2
 DISP=ALL
 SUPER=20,1
 LOAD=1
SUBCASE 3
 SUPER=0,1
 LOAD=1
BEGIN BULK
PARAM EXTUNIT     30
SEBULK      10    EXTERNAL
BEGIN SUPER=10
$ Substructure B (external superelement)
EXTRN       3     1     4     1
GRID        3           2.    1.    0.
GRID        4           2.    0.    0.
BEGIN SUPER=20
$ Substructure A
GRID        3           2.    1.    0.
GRID        4           2.    0.    0.
GRID        5           3.    0.    0.
CELAS2      4     5.    3     1     5     1
CELAS2      5     10.   4     1     5     1
SPC1        1     23456 3
SPC1        1     23456 4
SPC1        1     23456 5
FORCE       1     5           60.   1.    0.    0.
ENDDATA
```

Internal d.o.f.'s displacements of substructure A and interface d.o.f.'s displacements are output in F06 file:

```
DATA RECOVERY FOR SUPERELEMENT       20 IS NOW INITIATED.
DMIGOP2, RUN 2                            MAY   5, 2000  MSC.NASTRAN 10/11/99   PAGE    24
                                                                 SUPERELEMENT 20  ,   1
                                                                 SUBCASE 2

                              D I S P L A C E M E N T   V E C T O R

 POINT ID.   TYPE         T1          T2          T3          R1          R2          R3
        3      G    6.400000E+00    .0          .0          .0          .0          .0
        4      G    7.600000E+00    .0          .0          .0          .0          .0
        5      G    1.120000E+01    .0          .0          .0          .0          .0
```

Example 2 – Local (Substructure A) Stiffness Modification: Internal Forces Method

$$\left[\begin{array}{cc:c} 5 & 0 & -5 \\ 0 & 10 & -10 \\ \hdashline -5 & -10 & 15 \end{array}\right]\left\{\begin{array}{c}\delta_3\\ \delta_4\\ \hdashline \delta_5\end{array}\right\}=\left\{\begin{array}{c}-20\\ -40\\ \hdashline 60\end{array}\right\}$$

Above system of equations has a singular stiffness matrix and cannot be solved. A restraint condition is needed: considering $\delta_4 = 8$, the rest of displacements can be calculated, resulting in:

$$\left\{\begin{array}{c}\delta_3\\ \delta_4\\ \delta_5\end{array}\right\}=\left\{\begin{array}{c}8\\ 8\\ 12\end{array}\right\}$$

that **are incorrect and clearly different** than expected.

Example 2 – Local (Substructure A) Stiffness Modification: Enforced Displacement Method

$$K_{aa}\cdot\delta_a=\left(F_a-K_{ai}\cdot\delta_i\right)$$

$$15\cdot\delta_5=60-\left[-5\quad -10\right]\left\{\begin{array}{c}6\\ 8\end{array}\right\}$$

That gives δ_5= 11.333, that is, **again, an incorrect** result.

1.3.6.6 Static Submodelling Methods Summary

Three static submodelling methods are suggested, namely internal force method, enforced displacement method and static condensation substructuring using superelements. The **internal force method** applies cut boundary forces at *all* the cut boundaries of the submodel, constraining the submodel anywhere really such that it is ***just*** statically determinate. The reactions will be found to be zero (or close to zero if the geometry is accurate and there are no significant Poisson effects along the constrained boundary in 2D and 3D models) because the submodel is subject to a self-equilibrating set of forces. The limitation is only that changes in stiffness of the submodel (due to design geometry or material changes or simply due to the fact that the stiffness will change from the global model to the local model) will produce inaccuracies in the solution. Physically, this is so because if you change the local stiffness of the submodel, the force will either be more attracted or less attracted depending on whether you increase the stiffness or decrease it. A special case occurs when the rest of the structure is 'isostatically mounted upon the submodel' i.e. for instance a tree where the submodel is the trunk and the rest of the structure is the branches. In this special case, the internal force method will be exact.

The **enforced displacement method** applies cut boundary displacements at *all* the cut boundaries of the submodel. No further constraints are required as the submodel will probably no longer be singular due to the application of the enforced displacements, failing which it is perfectly acceptable to constrain the mechanisms such that the submodel is statically determinate. Again the reactions will be found to be zero because the submodel is self-equilibrating. St. Venant's principle states *"if an actual distribution of forces is replaced by a statically equivalent system, the distribution of stress and strain is altered only near the regions of load application. This implies that stress concentration effects are localized around the concentration; therefore, if the boundaries of the submodel are far enough away from the stress concentration, reasonably accurate results can be calculated in the submodel".* Because the submodel will usually be of a finer mesh density, it becomes necessary to interpolate displacements from those at known points along the cut boundaries to *all* the nodes along the cut boundaries. The limitations of the enforced displacement method is the same as that as for the internal force method, being that inaccuracies are introduced when stiffness changes are made to the submodel.

The **substructuring method using the static condensation capability of superelements** reproduces the exact state of stress and displacements in the local model compared to that of the global model. Apart from refined stress analysis on the detailed local model, stiffness modifications can be made to the local model for local redesign iterations, investigation into local buckling analysis and the modelling of localized non-linearity (to the submodel) can be made. The exact solution is obtained so long as the rest of the structure is modeled with superelements. The stiffness (and internal forces) of the rest of the structure is contained within the superelements, hence it is like modelling the entire structure, only that computationally it is far more efficient.

1.3.7 Submodelling Techniques for Dynamic Analysis

1.3.7.1 Submodelling by Substructuring - Static Condensation (Guyan Reduction) using Superelements

For linear dynamic analysis, the static condensation technique is only an **approximation** (unlike linear static analysis where it is exact) of the whole structure behaviour, and methods with highest efficiency are the ones based on modal synthesis. However, static condensation **becomes exact** for dynamic analysis **if there are no loads applied onto the o-set DOFs**, i.e. the set of freedoms omitted by static condensation.

1.3.7.2 Submodelling by Substructuring - Generalized Dynamic Reduction (GDR) using Superelements

Generalized dynamic reduction (GDR) offers a more accurate solution by including an approximation of the dynamic effects when forming the transformation. Bulk Data entries for GDR are DYNRED, QSET, and QSET1, and SPOINT. In addition, the Case Control command DYNRED is also required.

1.3.7.3 Submodelling by Substructuring - Component Mode Synthesis (CMS) using Superelements

Component mode synthesis (CMS) is a form of superelement dynamic reduction wherein matrices are defined in terms of modal coordinates (corresponding to the superelement modes) and physical coordinates (corresponding to the grid points on the superelement boundaries). CMS is advantageous because there are fewer modal coordinates than physical coordinates—perhaps only one percent as many. In addition, CMS can utilize modal test data, thereby increasing the accuracy of the overall analysis.

1.3.8 Modelling Flat Plates Using 2D Shells or 1D Grillages

Grillage analyses may sometimes be used to model flat plates such as flat slabs. A grillage consists of 1D beams in two orthogonal directions to replicate the two-way spanning action that flat shells or plates perform perfectly. Two types of models may be used in both 1D grillages and 2D shells, i.e. with torsional stiffness or without. If torsional stiffness is specified, then it must be designed for. For the shell, the correct shear modulus G according to linear elastic theory $G = E/(2(1+\nu))$ should model the torsional stiffness and thus must be designed for. To not have to design for the torsion, the shear modulus G should be manually set very low, although this makes the assumption that there is no shear flexibility modeled. This is because the shear rigidity is given approximately by GA_s i.e. a function of the shear modulus as well.

Now for grillages to replicate shells or plates, firstly since beams are used in two orthogonal directions, the **density of the beams must be halved** in order not to double up; Secondly, the bending rigidity D for a plate in theory should be **$EI/(1-\nu^2)$ instead of EI** as for a rectangular beam. Thirdly, if torsion is to be designed for, the **torsional rigidity GJ should be such that it is equal to the bending rigidity EI** so that there is equal bending stiffness of the plate about any horizontal axis. Since $G = E/(2(1+\nu))$, J should be set such that $J = EI/G = 2I(1+\nu)$. One of the advantages of using grillages instead of shells is the ability to design out torsional effects, i.e. by putting J = 0, shear deformations are still accounted for since G is still defined.

1.3.9 Modelling Down-Stand Beams on Structural Floors, Pile Caps and Changes in Thickness of Plates

Floor slabs with down-stand beams, pile caps or changes in thickness of the plate **should** be modeled with **offsets to the neutral axis of the local section.** Down-stand beams should be modeled with offsets from the shells elements to the centroid of the down-stand beam, which is modeled as beam elements, NOT shells. Pile caps or local changes in thickness of a plate can be modeled with local shells offset (to their neutral axis) from the general plate elevation.

Connections between elements should be at their neutral axis. Connections of slabs with offset beams are no different, the elastic neutral axis should be found and both the beams and the slabs should be offset from their common neutral axis to connect into column (beam) elements. This is important especially so in linear dynamic problems where an incorrect connection point will result in incorrect mode shapes.

To model the offset in MSC.NASTRAN, three methods can be used. RBE2 or RBAR, both with ALL 6 DOFs of the single dependent grid constrained to ALL 6 DOFs of the single independent grid, repeated over each and every offset grid pair can be used. Alternatively this can be automated easily using ZOFFS on the shell element connection card, this being the recommended approach. Lastly, we could also use MID4, but this is not recommended.

Note that offsets should be modeled because their would considerably increase the bending stiffness due to the Ar^2 stiffness terms (by parallel axes theorem). Clearly both the bending stiffness and the membrane stiffness of the shell is required if offsets are modeled.

1.3.10 Modelling Stiffened Plates

Modelling stiffeners using plate elements usually results in a too stiff representation. Stiffeners require cubic displacement functions to model the bending action, but shells in membrane action are capable of only linear or parabolic displacement interpolation. Hence, a single or two-shell element stiffener representation is almost always too stiff and peak stresses will be severely overestimated. Of course, with refinement, the model approaches the correct stiffness. The recommended approach is instead to use offset CBAR beam elements to model the stiffeners.

1.3.11 Modelling Connections

Rigid joints do not allow relative rotation between connecting members whereas pinned joints do not transfer bending moments. Within the analysis code, a pinned ended beam element is akin to having a zero-length rigid link between the connecting members in all the DOFs except the rotational DOF that corresponds to the bending moment. This also means that we could theoretically model a partial fixity with a zero-length rotational spring between the rotational DOFs of the connecting elements. Modelling steel frames with pinned connections may sometimes be appropriate if we desire that all lateral loads are taken by the bracing members instead of through the moment connections. However, the rotations at pinned connection will be much greater, and the design must account for this. However, trusses are usually triangulated so that the lateral forces are accounted for by the inclined elements. Monolithic concrete frames are modeled with fixed connections. Connections in timber are usually pinned.

RBE2 or RBAR elements can be used to define hinges or sliding joints. To model a hinge connection with an RBAR, use coincident grid points at the center of rotation and define an RBAR between the two grid points. This RBAR has zero length, which is acceptable for the RBAR. Make **ALL 6** of the components associated with one grid point independent. Make only a select number of components of the other grid point dependent, leaving independent the components representing the hinge. With RBE2, again connect the two coincident nodes with an RBE2 with all DOFs of the dependent grid except that representing the hinge **automatically dependent on ALL 6 DOFs** of the independent grid. Both connections are then exactly the same. RBE2 is characteristic in the sense that **ALL 6** DOFs of the independent grid is automatically selected, the user then has the option of selecting the DOFs of the dependent grid, but of **more than one grid** if intended. RBAR on the other hand, allows the option of selecting the DOFs of both the independent and dependent grid, but of **only one** pair.

Modelling pinned connections in 2D and 3D elements require different considerations. Consider the modelling of a **thin 2D lever with a pinned connection**. The lever is allowed to pivot about a pin through its mounting hole. The interface between the lever and the pin is of consideration. Of course, to restrain all the nodes along the curve of the hole in translational and leaving the rotational DOFs free with respect to the basic coordinate system, is blatantly incorrect. To connect a 'spider' i.e. an RBE2 to all the dependent nodes on the curve of the hole and pin the independent node defined in the middle of the mounting hole may be acceptable, but this does (incorrectly) add stiffness to the model or rather an infinite rigidity around the hole (as if there was firstly, infinite friction between the curve lining the mounting hole and the pin, and secondly no allowance for the hole to expand radially or squash into an oval shape) as the constrained nodes cannot move in translation relative to each other, and hence the stress distribution around the hole will be totally incorrect. Another method to model this pinned connection would be to define independent SPCs around the outline curve of the hole, but ensuring that a local coordinate systems **in cylindrical coordinates (a spherical coordinate system will be used in 3D problems requiring a pinned connection)** is defined for all the constrained nodes. The nodes are of course constrained only in the radial direction to achieve the desired pivoting effect. Now each point on the curve of the hole is allowed to move tangentially and hence no stiffness or rigidity (or friction between curve lining the mounting hole and the pin) is added to the model and hence will produce the correct effect as the lever is allowed to pivot. But the stress distribution around the hole will still be inaccurate because the SPC would incorrectly resist the pull of the nodes away from the pin (hence not allowing the hole to expand) on top of correctly resisting the push. Hence, a manual iterative method must be undertaken to remove the SPCs that are resisting the pull of the lever away from the pin by inspecting the sign of the reaction on the SPCs. Gradually all the offending SPCs will be removed and the correct solution will be achieved. This similar effect can be attained automatically by modelling the pin explicitly and defining **contact elements between the lever and the pin.** This is probably the best method to model the pinned connection (be it in 2D or 3D). These contact elements can clearly resist the push but will not resist the separation between the lever and the pin. Of course, the orientation of these contact elements must be normal to the interface curves.

1.3.12 Modelling Spot Welds

Spot welds are best modeled with **CBUSH** elements with large stiffness in all 6 DOFs in MSC.NASTRAN. Alternatively, **RBAR or RBE2 (both with ALL 6 DOFs of the single dependent grid constrained to ALL 6DOFs of the single independent grid, repeated over each and every spot weld grid pair)** elements may be used effectively. CELAS elements can also be used but is not recommended as apart from being extremely difficult to set up, problems with the generation of internal constraints may arise.

The CWELD and PWELD entries are also ideal. Connections can now be established with ease between points, elements, patches, or any of their combinations. In short, the new CWELD connector element is general in purpose, easy to generate, less error-prone than any of the preceding methods, and always satisfies the condition of rigid body invariance. Test and analysis correlation has also been shown to be excellent. Connections can either be established conventionally, from point-to-point, or in a more advanced fashion between elements and/or patches of grid points. In the case of elements and patches, actual weld attachment points will usually occur within element domains or patches and will be computed automatically, with corresponding automatic creation of necessary grid points and degrees-of-freedom. Element- and patch-based connections, moreover, eliminate the need for congruent meshes. Reference grids that determine spot weld spacing, for example, can be defined beforehand which, when projected (using the CWELD entry) through the surfaces to be attached, uniquely determine the weld elements' location and geometry.

1.3.13 Modelling Bolts

CBAR elements can be used to model bolt element within a 2D or 3D mesh. Rigid elements can be used to model the finite dimension of the diameter of the bolt head or washer. Now if there is no tolerance between the bolt and its hole, then the bolt can be modeled with CROD elements to transfer only axial (and torsional) forces.

1.3.14 Modelling Applied Loads

1.3.14.1 Applied Concentrated Loads on One Dimensional Elements

CBAR elements have cubic polynomial displacement functions, thus models linear variation in bending moment. Hence, point loads and moments should be only applied at the end nodes of the beam. A single structural beam member should be discretized by a number of CBAR elements to model the internal point loads. An internal point load results in a piecewise cubic variation of the transverse displacement, a mode of deformation that is not captured by the continuous cubic variation of the two-noded beam finite element. A continuous uniformly distributed load within a beam finite element results in a continuous quartic variation of the transverse displacement, again a mode of deformation that is not subsumed by the continuous cubic variation of the finite element. Applying internal loads onto beam elements will cause the equivalent nodal loads that the code computes based on the assumed interpolation function to be inaccurate. Hence the nodal displacements will in turn be inaccurate. On the other hand, if the loads are only applied at the nodes, we can be sure that the correct behavior is captured and thus there is no need for mesh refinement.

1.3.14.2 Applied Face and Line Loads on Two and Three Dimensional Elements

Unlike beam elements, loads applied to shell elements should not be point loads or moments applied at the nodes. In theory, the stress under a point load is infinite and as the mesh is refined, the finite element method attempts to replicate the infinite stress. Consider a point load F at a single node with a characteristic element length 'a'. As the model is refined, 'a' becomes smaller, and so the stress F/a becomes larger and tends toward infinity in the limit. This causes singularity. Now consider a pressure or distributed load P over n nodes. As the model is refined, the area of the pressure application stays the same, because it is related to the physical problem and not the mesh. While the number of nodes sharing in the pressure increases, the applied force at each node will decrease. The stress is equal to the pressure independently of the number of nodes, so there is no tendency towards infinity, and therefore no singularity. Singularities thus exist when **a point load** or likewise **an SPC** is applied to a shell or solid mesh simply due to the nature of a node having no area in the FEM. Even a needle tip, when observed under a microscope shows a finite tip radius. A point load in FEA on the other hand is applied on a node with no area. Hence the stress within the adjacent element increases as the mesh is refined. The results will never converge as the mesh is refined, as the solution become **mesh-dependent**. This is bad enough in h-element analysis, more so in p-element model, where the code will attempt to converge in a futile attempt. Hence, it is best to distribute the point load over a line or area although the interpolation function cannot model the resulting deformations exactly. This is perfectly acceptable, as unlike 1D beams, meshes of 2D shells have to be refined. Note that face loads (applied as a stress) are dependent upon the geometry of the face of the shell and line edge loads (applied as a stress) are dependent upon the thickness and the edge dimension of the shell. The following table lists the load and boundary conditions that will and will not cause a singularity.

Load or Constraint on	Beam	Shell	Solid
Point	No	Yes	Yes
Curve or edge	No	No	Yes
Surface or face	N/A	No	No

Face or line loads should never be applied manually as individual equivalent concentrated loads. That is to say, a face pressure load should not be decomposed by the user by multiplying by the area and dividing by the number of grids on the face and applying these average concentrated loads all the grids on the face. This is because the user's equivalent loading **WOULD NOT** be comparable to the true consistent loading. The equivalent loads at the grid points computed from the element loads are known as consistent loads, and they are calculated by applying the principle of virtual work.

$$\{f\} = -\left\{\int_{\Omega} [N]^T \{b\}\, d\Omega\right\}$$

The same shape function that is used in deriving the element stiffness is used for arriving at this load-and hence the word consistent load. They are a function of the element types and the applied loads.

1.3.14.3 Applied Concentrated Loads on Two and Three Dimensional Elements

Rigid elements such as **RBE2 (with ALL 6 DOFs of the many dependent grids constrained automatically to ALL 6 DOFs of the single independent grid where the load is applied)** are usually used to distribute specified force or moment to be applied onto a 2D or 3D mesh. Clearly, the **independent grid** point must **correspond in space** to the point of application of the external load, i.e. the elastic neutral axis of the section.

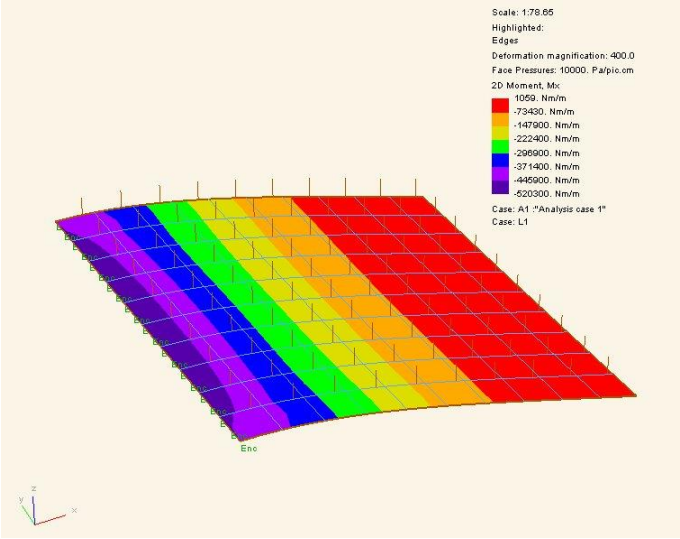
Alternatively the **RBE3** element may be used. The RBE3 element is a powerful tool for distributing applied loads and mass in a model. Unlike the RBAR and RBE2, the RBE3 **does not add additional stiffness** to your structure (a 2D shell mesh of a hollow tube being applied with a force or moment using an RBE2 *"spider"* to constrain all the grids on the section to the independent grid at the neutral axis center adds stiffness to the hollow tube section). Forces and moments applied to reference points are distributed to a set of independent degrees of freedom based on the RBE3 geometry and local weight factors. Basically, the **external force** is applied on the reference grid **REFGRID** in the **component DOF** direction **REFC**. The default action of this element is to place the REFC degrees of freedom in the *m*-set denoted by the grids GMi and components CMi after the field "UM". **C1** is then **ALL 3** translational component (rotational components only used in special circumstances) numbers of the first group of **independent** grids **G1,1, G1,2, G1,3,G1,4** etc … where the load is to be transferred to.

$ BULK DATA									
RBE3	**EID**		**REFGRID**	**REFC**	**WT1**	**C1**	**G1,1**	**G1,2**	
	G1,3	**WT2**	**C2**	**G2,1**	**G2,2**	**Etc…**	**WT3**	**C3**	
	G3,1	**G3,2**	**Etc…**	**WT4**	**C4**	**G4,1**	**G4,2**	**Etc…**	
	"UM"	**GM1**	**CM1**	**GM2**	**CM2**	**GM3**	**CM3**		
		GM4	**CM4**	**GM5**	**CM5**	**Etc…**			

To define a *"spider"* to transfer the beam type forces onto a 2D shell or 3D solid mesh, the REFGRID should correspond to a grid defined at the elastic neutral axis (centroid of section), the REFC should correspond to the component direction where the forces act i.e. in general 123456, WT1 denotes the weighting factor of the first group i.e. 1.0 for equal weighting here, C1 as 123 denotes the all three translational components of the independent grids of the first group G1,i which are the grids around the section. Note that C1 should be 123 even if there is force in only one or two translational directions. This is a software requirement as with all R-type elements to avoid singularity, not an engineering requirement. C1 should generally NEVER have any rotational components 456. Here only one group has been defined. Different groups are only required for different weighting factor to distribute the force. This occurs in certain groups of grids are deemed to be of different stiffness to the other grids. If a group of grids is deemed to be less stiff and hence attracting less force than that to the other grids, it will be justifiable to use a lower weighting factor to distribute less force to those group of grids.

1.3.15 Modelling Support Conditions

When supporting 2D shells, it is best to apply the support conditions on all the nodes along the edge including the mid-side nodes in order to obtain a smooth stress variation along the edge. There will be a smooth variation of the bending stresses along and perpendicular to the edge and the through-thickness shear stress variation along and perpendicular to the edge. No stress singularity is introduced if the support condition is applied along the line.



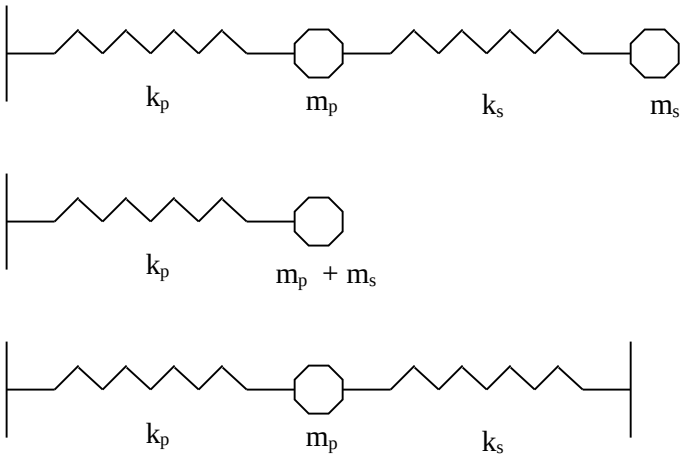
Of course, analogous to an applied point load, **A SINGLE** SPC applied on a 2D or 3D mesh will result in a singularity because a finite force value is applied on a node which has no area, hence the stress which is the force/area is infinite. The following table lists the load and boundary conditions that will and will not cause a singularity.

Load or Constraint on	Beam	Shell	Solid
Point	No	Yes	Yes
Curve or edge	No	No	Yes
Surface or face	N/A	No	No

However, if stress recovery at the regions of the stress singularity is not required, then St. Venant's Principle, which states that stress and deflection far from an applied load or constraint can be represented by a statically equivalent loading scenario, can be invoked. One of the fortunate characteristics of the FEM is that local effect will most likely have little effect on the global behaviour.

Support stiffness is sometimes incorporated in the form of a ground spring, this may for instance model piles or foundations of different relative stiffness. This is **only done to model the differential settlement**. If the relative stiffness of **ALL** the supports is the same, then there is no reason to model the support stiffnesses. A ground spring is simply a spring eventually totally constrained to a rigid support. Reducing the stiffness of a support spring will probably reduce the force going into that support constraint, but the load must be accounted for elsewhere, i.e. all the load must be eventually accounted for. The total reactions must balance the total applied loading (assuming no ill-conditioning effects).

1.3.16 Primary and Secondary Component in Dynamic Analysis



Secondary components may have to be included into the dynamic model of the primary component if they have frequencies comparable to those of the primary component. It is not the mass or the stiffness of the secondary component in isolation that matters in deciding whether they are dynamically prominent, but their natural frequencies. If the resonant frequencies of the secondary component are greater ($\omega_s/\omega_p > 2.0$) than those of the primary model, then they can be modeled as added mass on the primary system. If the resonant frequencies of the secondary component are lower ($\omega_s/\omega_p < 0.5$) than those of the primary model, then they can be modeled as stiffness (which are grounded, hence no mass DOF) on the primary system. Otherwise, the secondary system must be included within the dynamic model of the primary system.

1.3.17 Modelling Rigid Body Motion

An excellent method of evaluating whether the intended rigid body motion is correctly represented within the finite element model is to perform a SOL 103. Note that AUTOSPC may have to be set to the non default value of NO to avoid grid point singularities from being constrained. The eigenvalue analysis also cannot model local mechanisms (identified by a critical MAXRATIO). But the eigenvalue analysis can model global mechanisms, i.e. the rigid body motion. The correct stiffness properties, boundary conditions and rigid element DOF dependency can be adjusted until the correct rigid body motions (at zero or close to zero frequencies) are obtained from visual inspection of the eigenvectors following an eigenvalue SOL 103 analysis.

1.3.18 Meshing Strategies

With mesh refinement, the results will converge to the true solution for displacements and stresses. Stress analysis requires a much finer mesh density than static displacement response. This is because of the rate of convergence of displacement response being much greater than that of the stress. It would thus help to allow for fine discretization in regions of high stress gradient and stress concentrations such as: -

(a) near entrant corners, or sharply curved edges – using **chordal deviation** automatic meshing techniques say
(b) in the vicinity of concentrated loads, concentrated reactions, cracks and cutouts
(c) in the interior of structures with abrupt changes in thickness, material properties or cross sectional areas

For 1D meshing, **linear1D (1D elements from one 2D plane to another) or the linemesh automesher (line mesh along a line)** can be used.

For 2D meshing, uniform well shaped quadrilateral meshing is possible using **spline** and **ruled** within defined boundaries. It is also possible to **extrude (along an arbitrary line) or to revolve a line** to generate a 2D mesh. In the limit, the **2D automesher** can be used to mesh irregularly any arbitrary planar surfaces with a combination of quads and triads.

For 3D meshing, only volumes that can foreseeably be generated from **extruding (in general along an arbitrary line) or revolving a planar 2D mesh** can be meshed uniformly with the more accurate hexagonal elements. Meshing an enclosed volume of arbitrary shape often results in having to utilize the **automesher** (using either a **fixed nominal element size** or using **chordal deviation techniques** that refines the mesh size as the radius of curvature decreases), which employs tetrahedral meshes, hence a.k.a. **tetramesher**. CTETRA4 are constant strain elements and hence should be minimized since many more elements are required for convergence. The arbitrary volume can be better mapped with CTETRA10 elements utilizing the mid side node to model the surface curvature. Second order elements (QUAD8, TRIA6, TETRA10 and HEXA20) are useful in better approximating the curvature of the actual geometry due to their mid-side nodes. Hence it may be better to generate the mesh with second order elements straight away than to mesh with first order elements and only then change to second order elements. Automeshers are generally tetrameshers. Hexagonal automeshers are rare if not nonexistent, although a fair estimate would be that one would need 5 tetrahedrons for every brick element.

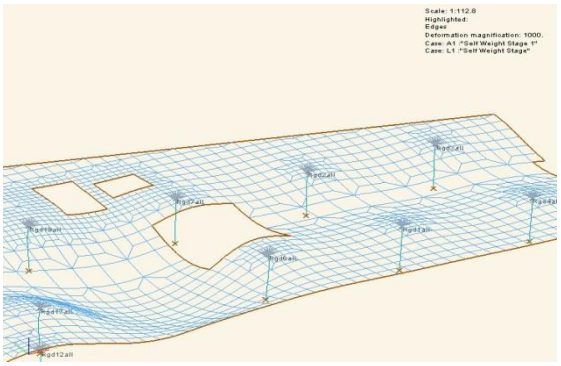
Employing the automesher requires a **clean geometry**. For transferring geometry, kernel based geometry formats such as ACIS and Parasolids are recommended. Standard based geometry formats such as IGES, DXF and VDA may lead to translation errors. Inheriting a CAD geometry model often requires a cleanup operation to yield geometry suitable for the automesher. This process is certainly worth embarking upon before employing the automesher. One aspect of the geometry that must be cleaned up is **short edges and sliver surfaces**. **Short edges** occur in geometry when lines or surfaces do not meet exactly at the same point due to **misalignment of features**. **Sliver surfaces (i.e. geometry surfaces with high aspect ratio)** occur in geometry due to **misalignment of features** or due to the **close proximity of fillet edges to other edges**. Short edges and sliver surfaces with a dimension short compared to either the model size, or more importantly the nominal mesh size, will introduce errors into the model by forcing one edge of a triangle or quadrilateral to the length of that short dimension. This occurs because the automesher will try to space nodes on the geometry edges first and then within the geometry surfaces at the defined nominal element size. Since most automeshers are constrained to use every node and edge of the model, when short edges or short dimensions are encountered, a side of an element will be placed on the short edge whilst the other sides of the element will be of the user defined nominal element size. This clearly causes highly distorted elements. Limiting the size of the short edges to no less than 1/3 of the nominal element size is good practice. In short, the geometry should be checked for both short edges and sliver surfaces. The former is identified by checking for any **short edge** that may exist in the geometry **due to the misalignment of lines or surfaces edges that should meet at a common point or common line**, respectively. The latter is identified by firstly, checking for surfaces with high aspect ratio due to short edges which in turn is **due to the misalignment of lines or surfaces that should meet at a common point or common line**, respectively and secondly, by identifying **lines or surface edges of close proximity** belonging to different features.

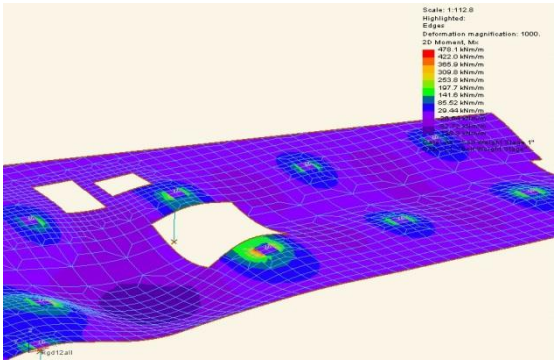
1.3.19 Mesh Transition Techniques

1.3.19.1 Connecting 1D Elements to 2D Elements

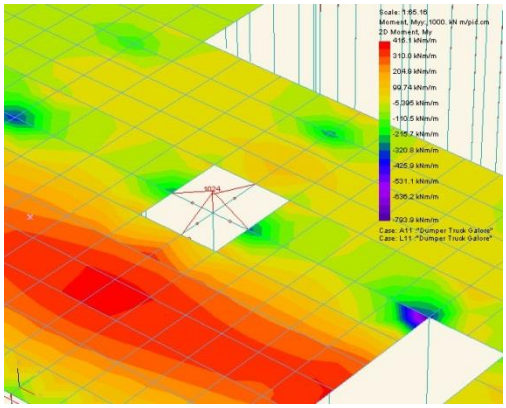
When connecting 2D shells to beams in the same plane, it is best to connect the beam to the mid-side nodes of the shells as well to ensure compatibility between the elements. Offsets should be modeled as this affects the stiffness greatly. No stress singularity is introduced as the beams are connected to all nodes lining an edge of shell elements within or on the edge of the shell mesh.

When connecting one dimensional beam elements to shells orthogonally, special considerations must be observed. 2D shells are often used to model irregular flat slabs with 1D beams modelling the vertical columns. Very often we incorrectly model vertical columns fixed into a horizontal mesh of 2D shells at a particular node and perform stress recovery at the intersection. Obviously, as a 2D mesh is refined, theoretically, the stress (be it the bending or the shear) at the point of intersection goes to infinity (stress singularity), and the 2D mesh attempts to replicate that. However, since stress recovery (bending moment) at the pile tops are usually of concern in design for hogging, to read those values would be very much dependent upon the mesh size (**mesh-dependent**) and would simply be meaningless. Infinite stresses will result irrespective of the stress recovery procedure used in the shells, whether BILINEAR, CUBIC etc. Incidentally, if the mesh is not so fine, one can obtain pretty believable (but incorrect) answers, as the regions of stress concentration have not been reached. But it is when you refine the mesh that you suddenly start to get huge variations of stress (hence derived moment) and great discontinuities. To think that people normally associate 2D elements to give you more accurate results. One method is to use rigid **RBE2s** connecting the beam to a few nodes on the shell mesh in order to distribute the point load. The area over which the point load is distributed should replicate reality. The rigid connection should connect to all the nodes on the 2D element mesh within the relevant vicinity, i.e. including also the mid-side nodes. There will be no stress singularity.





Another method is to mesh the pile tops of slab with beams with the same stiffness in line with methods of grillage analysis. Alternatively, a rigid may be incorporated with a dimension of the pile diameter to model the pile top, which is much stiffer and does not in reality act like a beam since it is very much constrained by the pile. These methods would be akin to moving the mesh transition region (here between two different elements in 1D beams and 2D shells) from the vicinity of interest (for stress recovery). But stress singularity still exists at the transition.

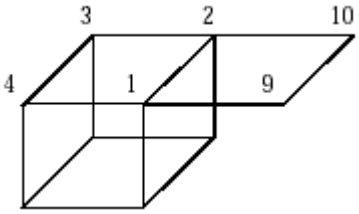


Similar considerations apply when stress recovery is intended on the slabs over the edges supported monolithically by edge piles, which transfer moments. Again the edges of the slab can be meshed with beams in line with the methods of grillage analysis. But stress singularity still exists at the transition. Alternatively, since the slab is in series with the piles, the moment experienced at the edges of the slab must be equivalent to the moment in the piles. Hence the moment in the piles can be used to design for the moment in the edges of the slab.

1.3.19.2 Connecting 2D Elements to 3D Elements

Solid elements have stiffness only in the translational DOFs at the attachment grid points; they have no stiffness for rotational DOFs. To connect a shell into a 3D mesh requires special modelling to connect the rotational DOFs of the shell into the 3D solid mesh. One method would be to add an additional layer of shells that continue into the solid mesh. Another method would be to use **RBE2s (each RBE2 with ALL 6 DOFs of the many dependent grids on the solid mesh constrained automatically to ALL 6 DOFs of the single independent grid on the shell mesh)** to rigidly connect the translational and rotational stiffnesses of the shell mesh to the translational stiffnesses of the solid mesh, although this will locally increase the model stiffness.

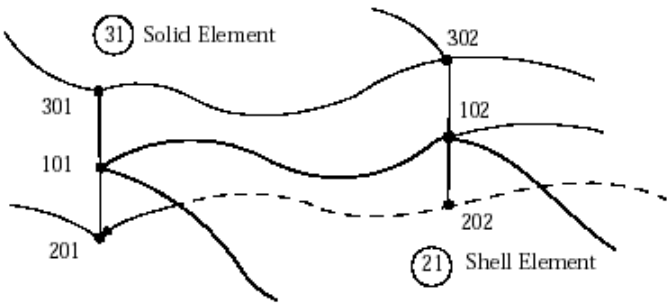
RBE3s can be used to model the connection between shells and solids. By using RBE3s, the rotations of the attached grid points are simply slaved to the translations of the adjacent grid points. In the following example, two RBE3 cards are required. These RBE3 elements transmit the loads to the independent DOFs. If RBE2 elements are used, then the connection is "rigid."



$ BULK DATA									
$ RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
RBE3	1		1	456	1.0	123	2	3	
	4								
RBE3	2		2	456	1.0	123	1	3	
	4								

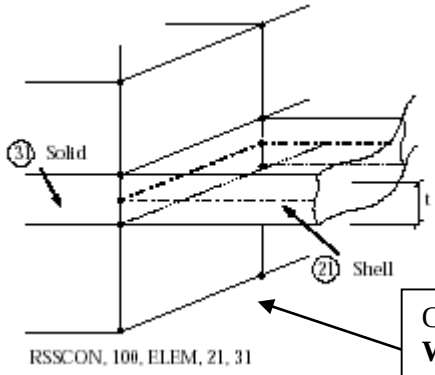
An alternate method is to use the **RSSCON** element. When using the RSSCON capability, the shell element mesh must line up with the solid element mesh so that there is an exact element-to-element correspondence. RSSCON generates a multipoint constraint, which puts the shell degrees of freedom in the dependent set (m-set). The three translational degrees of freedom and the two rotational degrees of freedom of the shell edge are connected to the three translational degrees of freedom of the upper and lower solid edge. Poisson's ratio effects and temperature loads are modeled correctly. There are two options for making this connection using the RSSCON entry, namely

1. The ELEM option, which allows you to specify the element ID of the shell and the element ID of the solid to which the shell is to be connected.
2. The GRID option, which allows you to specify the grid point ID of the shell and the upper and lower grid point IDs of the solid. Two triplets of grid point IDs may be specified on one RSSCON entry.

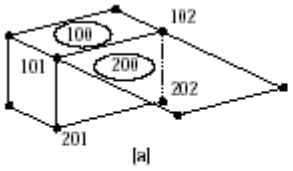


RSSCON, 100, ELEM, 21, 31
or
RSSCON, 200, GRID, 101, 201, 301, 102, 202, 302

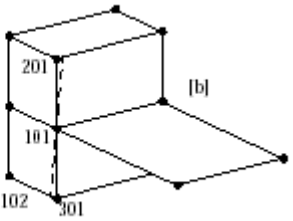
The best modelling practice is when the height of the connected solid element is chosen equal to the thickness of the shell. If the height of the connected solid element is much larger than the thickness of the shell element, then the connection modeled with RSSCON will be stiffer than the continuum model. In a mesh where shell grid points are identical or coincide with solid grid points, the RSSCON Bulk Data entry may model a connection that is too stiff



RSSCON, 100, ELEM, 21, 31



RSSCON, 200, GRID, 101, 201, 101, 102, 202, 102
or
RSSCON, 200, ELEM 200, 100

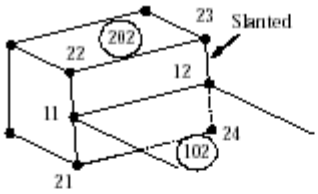


RSSCON, 100, GRID, 101, 201, 301

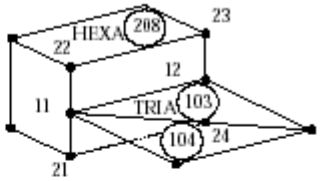
Connecting a shell into solid by extending the shell one layer into the solid **WILL** introduce **stress concentrations**. The best method would thus be to use rigids or RSSCON to connect transverse to plane of shell.

Best Modelling **Too Stiff Connections**

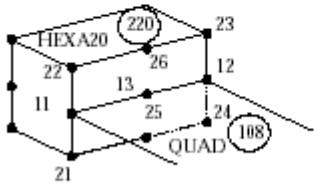
Other versatile connections are presented.



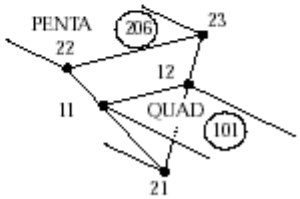
RSSCON, 100, ELEM, 102, 202
or
RSSCON, 100, GRID, 11, 21, 22, 12, 24, 23



RSSCON, 200, ELEM, 103, 208
RSSCON, 201, ELEM, 104, 208 (not necessary)
or
RSSCON, 200, GRID, 11, 21, 22, 12, 24, 23



RSSCON, 200, ELEM, 108, 220
or
RSSCON, 200, GRID, 11, 21, 22, 12, 24, 23
RSSCON, 201, GRID, 13, 25, 26



RSSCON, 200, ELEM, 101, 206
or
RSSCON, 200, GRID, 11, 21, 22, 12, 21, 23

The RSSCON connector produces excellent results if the user follows good modelling practices. The geometry of the RSSCON connector is checked, and fatal messages are issued for invalid connections. Additional recommendations are given below.

1. Midside noded elements should *only* be connected to other midside noded elements. For example, a CQUAD8 element connected to a CHEXA20 element is acceptable; however, a CQUAD4 element connected to a PENTA15 element is not allowed. Elements with missing midside grid points are not allowed.

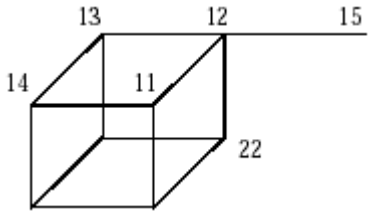
2. For midside noded elements, avoid using the RSSCON where shells connect to triangular solid faces.

3. Do not mix coordinate systems when using midside nodes. In other words, when using midside nodes, do not define a local coordinate system containing all the shell grid points and the basic coordinate system to define the solid grid points. Otherwise, incorrect answers will be generated.

4. Only one edge of a single shell element may connect to a solid surface.
5. Do *not* connect more than one shell element to one solid element when using the ELEM option.
6. Do not attempt solid-shell connections to severely warped (greater than 20%) mid-side noded solid elements.
7. The plane of the shell element should not be parallel to the plane of the connecting face of the solid element.
8. The connecting edge of the shell element must lie in the face of the solid element. In addition, the vertex grid points of the shell edge connecting to a quadrilateral solid face must line up with its opposite edges; for triangular solid faces, the connections may be made between any of the edges.

1.3.19.3 Connecting 1D Elements to 3D Elements

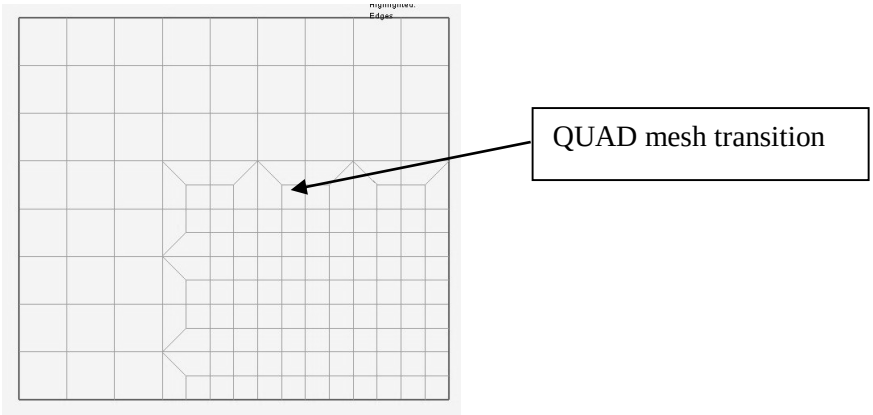
A method would be to use **RBE2s** to rigidly connect the translational and rotational stiffnesses of the beam to the translational stiffnesses of the solid mesh, although this will locally increase the model stiffness. Alternatively, one **RBE3** element can make the connection. These RBE3 elements transmit the loads to the independent DOFs. If RBE2 elements are used, then the connection is "rigid."



$ BULK DATA									
$ RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
RBE3	1		12	456	1.0	123	11	13	
	22								

1.3.19.4 Fine to Coarse Mesh Transitions in Shell Meshes

Compatibility would be satisfied across inter-element boundaries barring poor user meshing. For compatibility, the displacement function must be continuous across element boundaries, i.e. at least C^0 continuity at these boundaries. Inter-element C^0 continuity is satisfied for standard TRIA3, TRIA6, QUAD4, QUAD8, HEX8 and HEX20 elements. Compatibility is not satisfied when a QUAD4 is connected to the mid-side nodes of a QUAD8 or a TRIA3 to the mid-side nodes of a QUAD8 and other poorly formed connections or mesh transitions. Note however that QUAD8 connected to a TRIA6 will ensure compatibility. A good mesh transition for refinement is simply to use QUADs (instead of TRIAs) throughout the transition.



1.3.20 Stress Singularity (Artificial Stress Concentration) and St. Venant's Principle in Linear and Nonlinear (Material) Analysis

Stress singularities exist when a single point load or a single SPC is applied on a shell or solid mesh. They also exist when a line of point loads or SPCs are employed on a solid mesh. The following table summarizes these conditions.

Load or Constraint on	Beam	Shell	Solid
Point	No	Yes	Yes
Curve or edge	No	No	Yes
Surface or face	N/A	No	No

In perfect analogy, stress singularities also exist when **a single beam element connects to a shell mesh**. In all these cases, the stress singularity causes meaningless results as the mesh is refined as the ultimate solution becomes mesh dependent. Hence, they also cause p-element solution algorithms to fail as the code attempts, in vain, to converge to a finite solution.

There are no stress singularity problems when shells are connected in plane, except at **an internal corner in a planer shell mesh with zero radius subject to a membrane force** where there will be an infinite stress if made from a perfectly elastic material. Generally, a radius must be created in the corner to achieve a measurable stress. The confusing issue with modelling this in finite elements is that sensible stresses can be predicted in the corner. This is just coincidence however; refinement of the mesh would see the stresses in the corner increase without limit. Internal corners should always be considered in any analysis as a potential failure area and these must have a realistic fillet radius inserted for convergent stress values to be predicted. Remember that the stress value reported in a non-filleted internal corner is only dependant on the element size and has nothing to do with any real value that might occur there.

There is no stress singularity problems when shells are connected out of plane, except when **a plane shell mesh subject to a membrane force is stiffened by another shell mesh oriented perpendicular to it in the direction of the membrane force.**

All these stress singularities can be avoided using proper modelling techniques. Single point loads should not be applied to a shell mesh, instead only edge loads or pressure loads. A single point load or a line of point loads should not be applied on a solid mesh, instead only pressure loads should be used. Note that equivalent user defined multiple point loads to represent edge loads (for shells) or pressure loads (for shells or solids) should not be attempted as the equivalent nodal loads generated depend on the shape functions of the particular element.

$$\{f\} = -\left\{\int_{\Omega} [N]^T \{b\}\, d\Omega\right\}$$

Likewise, a single SPC should not be applied to a shell mesh, instead only a line or a surface patch of SPCs, and in both cases, SPCs should be applied also on the mid-side nodes of the elements if they exist. A single SPC or a line of SPCs should not be applied on a solid mesh, instead only a patch of SPCs, but also applying SPCs on the mid-side nodes of the elements if they exist.

In perfect analogy, when **1D elements connect to 2D or 3D elements**, multiple 1D elements should be used to avoid the stress singularity. For instance multiple gap elements to model the contact interface between two shell or solid meshes are perfectly acceptable. No stress singularity results. Likewise, no stress singularity exists when a beam mesh lines the edge of a group of shell elements within or at the edge of a shell mesh. The connection should however be compatible. Connecting the mid-side nodes if exist is recommended.

Likewise, connecting a **2D shell mesh into a 3D solid mesh** by extending the shell one layer into the solid WILL introduce stress concentrations. The best method would thus be to use rigids or RSSCON to connect transverse to

plane of shell so that the membrane force is distributed, whilst still coupling the rotational DOF of the shell into the solid through their translational DOFs.

However, if stress recovery at the regions of the stress singularity is not required, then **St. Venant's Principle**, which states that stress and deflection far from an applied load or constraint can be represented by a statically equivalent loading scenario, can be invoked. One of the fortunate characteristics of the FEM is that local effect will most likely have little effect on the global behaviour. The above St. Venant principle is acceptable for linear analysis (on h-element analysis bur not linear adaptive p-element techniques). However, nonlinear material analysis will suffer from a similar problem as the adaptive p-element linear solution. High local stresses result from high local strains. A nonlinear material analysis will try to resolve the stiffness at a singularity by adjusting the material modulus as defined by its stress-strain relationship. Fictitiously high strains may cause additional nonlinear iterations that should not have been required by the true strain state of the model. Hence stress singularities must be avoided at all costs in nonlinear analyses.

1.3.21 Modelling Preload (Prestress)

Preloads or prestresses are applied to elements such as cables and bolted joints to incorporate the differential stiffness within the calculations. Of course, the preload is also considered when performing stress recovery. A minimum of 2 runs is required to be able to simulate the correct preload in a bolted joint, but only if there is to be external loads applied to the system. If no external loads are to be applied, in say the simulation of bolt tightening, then the analysis can be executed in one run, even for a materially nonlinear response, as long as you accept the small displacement (geometrically linear) limitations of the method used. The preload applied to a bolted joint will cause the bolt to stretch and the clamped component (called the abutment) to be compressed. How much the bolt stretches and the abutment compresses depends on the relative stiffness of these two components. There are 3 methods for configuring the preload in MSC.NASTRAN, all with accompanying restrictions. These are the temperature load method, the MPC constraint method and the GAP element method.

1.3.21.1 Temperature Load Method

This is the most general method and the only one that allows correct solution of a geometric nonlinear response. In this method, a negative temperature "freezes" the bolt so it contracts. Obtaining the correct preload in linear analysis requires 2 runs because you don't know the relative stiffnesses of the bolt and abutment. So you run once with any negative value of temperature and look at how much the bolt stretched and how much preload it induced. Now simply factor the temperature that freezes the bolt and re-run to obtain the correct preload. If the analysis is nonlinear, and the bolt or abutment will change stiffness due to deformation, this method becomes tiresome because you will need to iterate to find the correct preload as you try to follow the nonlinear force-displacement curve of the bolted joint. In this case, it is better to use the MPC method described below to obtain the approximate displacement of the bolt due to the preload. Of course, this will be limited by small displacement theory. Then you can apply the temperature to obtain the displacement from this run. This method also presents problems when real thermal loads are applied. Care must be taken to calculate a modified temperature for the bolt, which is now heated or frozen for 2 reasons. If external loads are to be applied, apply them as usual.

1.3.21.2 MPC Constraint Method

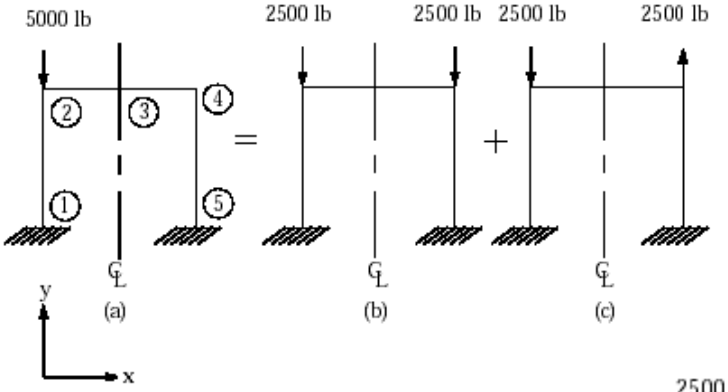
MPC constraints in MSC.NASTRAN are subject to small displacement theory because the constraint direction is not updated due to deformation of the structure. Consequently, if the analysis is geometrically nonlinear, then the results may be inaccurate if the bolt direction changes significantly during analysis. Using beam elements with this method requires that the bolt is split somewhere along its length and the two parts of the bolt connected together with an RBE2 element in all directions except the axial direction. The axial direction of the two parts of the bolt is connected with an MPC and a third point (an SPOINT) is used to apply the preload (using an SLOAD entry). The displacement in the SPOINT from this run is recovered and used in a second run as the SPCD value to apply to the MPC (the SLOAD is removed for this run). Alternatively, you can now use this displacement to calculate the temperature to apply in a run similar to the one above. If external loads are to be applied, apply them as usual.

1.3.21.3 GAP Element Method

The GAP element is subject to the same small displacement theory as MPCs, except the axial direction may have large displacement; large rotations are not supported. Again using beam elements with this method requires that the bolt is split somewhere along its length and the two parts of the bolt connected together with an RBE2 element in all directions except the axial direction. The axial direction of the two parts of the bolt is connected with a GAP element and the F0 field of the PGAP entry is used to specify the preload. Once again, the displacement (now in the GAP) from this run is recovered and used in a second run as the U0 value on the PGAP entry (F0 is left blank for this run). If external loads are to be applied, apply them as usual. In the MPC and GAP cases, if you use SLOAD or F0 respectively to load the bolted joint, this will give the correct behaviour as long as external loads are not applied. Applying external loads without using SPCD or U0 causes the load in the bolt not to change when the external load is applied; so 2 runs are needed to yield correct behaviour, but only when external loads are applied.

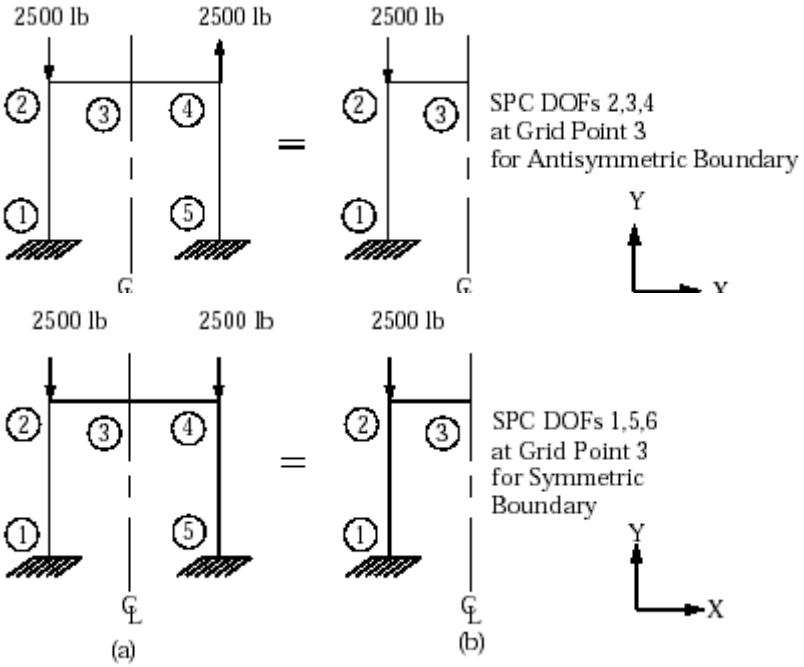
1.3.22 Modelling Symmetry

A structure that has one or two planes of symmetry (irrespective of loading) can be half or quarter modeled, respectively. If the loads applied to the structure are symmetric relative to the plane of symmetry, then the full model can be replaced with half the model by applying a symmetric boundary condition. On the other hand, if the loads are antisymmetric, the same simplification can be achieved by applying the antisymmetric boundary condition. A symmetric boundary condition implies that the displacements normal to the plane of symmetry and rotations about the axes in the plane of symmetry are zero at the plane of symmetry. An antisymmetric boundary condition implies that the displacements in the plane of symmetry and rotations normal to the plane of symmetry are zero at the plane of symmetry. Even when the loading is not symmetric or antisymmetric about the structure's plane of symmetry, advantage can be taken of symmetry. This is because any general loading pattern can be broken up into a combination of symmetric and antisymmetric loads relative to the plane of symmetry.




```
SUBCASE 1
LABEL = SYMMETRIC CONSTRAINTS-Y LOAD
SPC = 1
LOAD = 2
SUBCASE 2
LABEL = ANTISYMMETRIC CONSTRAINTS-Y LOAD
SPC = 2
LOAD = 2
SUBCOM 3
LABEL = LEFT SIDE OF MODEL-Y LOAD
SUBSEQ 1.0, 1.0
DISP=ALL
SUBCOM 4
LABEL = RIGHT SIDE OF MODEL-Y LOAD
```

The structure above can be analyzed with the half model but with 4 subcases. The first and second subcases model the symmetric and antisymmetric boundary conditions respectively, performing the analysis on the left half of the model. The third subcase combines the first and second subcases for the total results of the left half of the model. Finally, the fourth subcase computes the results of the right half of the structure.



Symmetry modelling techniques are **not recommended** in modal and subsequent dynamic response analysis. Whilst a combination of symmetry and anti-symmetry can be used to find all the mode shapes, this combination is not feasible for most dynamic studies. Also avoid symmetry in buckling analysis. Symmetry is however a great idea in nonlinear static analyses.

To apply **symmetry conditions on a plane in a 3D model**, all that is required is a set of translations SPCs in a direction normal to the cut plane. This would avoid Poisson effects at the boundary since contraction is not constrained in the plane of the cut, a scenario which will exist if all the nodes on the symmetry plane were connected by a rigid element (RBE2 with all 6 DOFs constrained) to an independent node on the neutral axis on which the 1D beam-like symmetry boundary conditions are applied. Symmetry conditions is different in elements that use mathematical representation (or DOFs) for their missing spatial dimensions, such as shell and beam models, where on top of the translation normal to the cut plane, the rotation about the two axes in the plane of the cut must be constrained.

1.3.23 Multi-Point Constraints (MPC) [3]

Single-point constraints refer to zero and non-zero essential boundary conditions being applied to a particular node onto one or more of its DOFs. Single-point constraints with zero prescribed displacements are implemented by reducing the global stiffness matrix to a reduced stiffness matrix that does not include the equations pertaining to the constrained DOFs. In other words, within the static equilibrium equation [K]{U}={P}, the rows and columns (hence also the associated DOFs in {U}) of [K] and the rows of {P} associated with the constrained DOFs are eliminated. For non-zero prescribed displacements essential boundary conditions, the rows of [K] and {P} associated with the non-zero constrained DOFs are eliminated and the prescribed values of the constrained DOFs are included within the {U} vector. Of course, in a computer implementation, the global assembly procedure and the reduction process will be intertwined for efficiency.

Multi-point constraints refer to constraints on multiple nodes. Three general methods of implementing multi-point constraints are: -

> **A. Master-Slave Elimination Using Rigid Elements with Case Control Command RIGID = LINEAR**. The degrees of freedom involved in each MFC are separated into master and slave freedoms. The slave freedoms are then explicitly eliminated. The modified equations do not contain the slave freedoms.
>
> **B. Penalty Augmentation Using CGAP Elements**. Also called the *penalty function method*. Each MFC is viewed as the presence of a fictitious elastic structural element called *penalty element* that enforces it approximately. This element is parametrized by a numerical *weight, w* on the stiffness. The exact constraint is recovered if the *weight, w* goes to infinity. In other words, we are simply adding an appropriately stiff element between the constrained DOFs in order to simulate a constrained connection. The MFCs are imposed by augmenting the finite element model with the penalty elements. The theory behind using this method to simulate general non-zero constraints is described within *Courant penalty functions*.
>
> **C. Lagrange Multiplier Adjunction Using Rigid Elements with Case Control Command RIGID = LAGR (also RIGID = LGELIM)**. For each MFC an additional unknown is adjoined to the master stiffness equations. Physically this set of unknowns represent *constraint forces* that would enforce the constraints exactly should there be applied to the unconstrained system.

The master-slave elimination method is clearly the exact method of implementing multi-point constraints. However, the algorithm requires that dependent (slave) DOFs not be dependent upon more than one master DOF. The method also requires rearranging the equilibrium equations and hence is more computationally intensive. Nonlinear constraints are even more difficult to implement.

The penalty function method on the other hand, adds an appropriately stiff element between the DOFs in order to simulate the correct prescribed displacement between them. For a zero displacement constraint, such as a rigid link, we would want the stiffness of the element to be as high as numerically possible. But unfortunately, the higher the ratio of the stiffness of the *rigid link* to the structure elements the more ill-conditioned the stiffness matrix will be. Obviously we have two effects at odds with each other. Making w larger reduces the constraint violation error but increases the solution error. The best w is that which makes both errors roughly equal in absolute value. This tradeoff value is difficult to find aside of systematically running numerical experiments. In practice the heuristic *square root rule* is often followed. This rule can be stated as follows. Suppose that the largest stiffness coefficient, before adding penalty elements, is of the order of 10^k and that the working machine precision is p digits. Then choose penalty weights to be of order $10^{k+p/2}$ with the proviso that such a choice would not cause arithmetic overflow. The name "square root" arises because the recommended w is in fact $10^k(10^p)^{1/2}$. Thus it is seen that the choice of penalty weight by this rule involves knowledge of both stiffness magnitudes and floating-point hardware

[3] FELIPPA, Carlos A. *Introduction to Finite Element Methods.* University of Colorado, 2001.

properties of the computer used. The penalty function method is less computationally intensive. There is no master-slave interdependence and hence there is no restrictions on dependent DOFs unlike the master-slave elimination method. The penalty function method can also be readily used to implemented nonlinear constraints. The main disadvantage, however, is a serious one; the choice of weight values that balance solution accuracy with the violation of constraint conditions. For simple cases the square root rule mentioned previously often works, although effective use calls for knowledge of the magnitude of stiffness coefficients. Such knowledge may be difficult to extract from a general-purpose "black box" program. For complex cases selection of appropriate weights may require extensive numerical experimentation, wasting the user time with numerical games that have no bearing on the real objective, which is getting a solution. The deterioration of the condition number of the penalty-augmented stiffness matrix can have an even more serious side effect in some solution procedures such as eigenvalue extraction or iterative solvers. Finally, even if optimal weights are selected, the combined solution error cannot be lowered beyond a threshold value.

The Lagrange Multiplier Adjunction method enforces the multi-point constraints by specifying appropriate forces at the associated DOFs. Since there forces or Lagrange multipliers λ are unknown before the solution is made, they are appended to the vector of original unknowns {U} by the process called adjuntion. The modified stiffness matrix is thus called the (multiplier-augmented) bordered stiffness matrix. This is solved for all the unknowns including the multiplier. In contrast to the penalty method, the method of Lagrange multipliers has the advantage of being exact (aside from computation errors). It provides directly the constraint forces, which are of interest in many applications. It does not require any guesses as regards weights. As the penalty method, it can be extended without difficulty to nonlinear constraints. It is not free of disadvantages. It introduces additional unknowns, requiring expansion of the original stiffness method. It renders the augmented stiffness matrix indefinite (i.e. no longer positive definite), a property that may cause grief with some solution methods such as Cholesky factorization or conjugate gradients. Finally, as the master-slave method, it is sensitive to the degree of linear independence of the constraints i.e. a DOF cannot be dependent upon more than one independent DOF. On the whole the Lagrangian multiplier method appear to be the most elegant for a general-purpose finite element program that is supposed to work as a "black box" by minimizing guesses and choices from its users. Its implementation, however, is not simple. Special care must be exercised to detect singularities due to constraint dependency and to account for the effect of loss of positive definiteness of the bordered stiffness on equation solvers.

	Master-Slave Elimination	**Penalty Function**	**Lagrange Multiplier**
Generality	Fair	Excellent	Excellent
Ease of implementation	Poor to fair	Good	Fair
Sensitivity to user decisions	High	High	Slight
Accuracy	Variable	Mediocre	Excellent
Sensitivity as regards to constraint dependence	High	None	High
Retains positive definiteness	Yes	Yes	No
Modification of DOF vector required	Yes	No	Yes

1.3.24 Model Checkout

1.3.24.1 Grid Point Singularities (AUTOSPC, EPZERO, K6ROT, SNORM)

1.3.24.1.1 AUTOSPC and EPZERO

Grid point singularity is identified by considering the stiffness at each single node (contrast with mechanism type singularity which considers stiffness at more than one grid point). Since NASTRAN is primarily an implicit code, there is a need to remove DOFs with low or zero stiffness to avoid the stiffness matrix from becoming singular. The stiffness (and mass for that matter) in every DOF on a grid is provided by elements. CELAS elements provide axial stiffness only. CROD elements provide axial and torsional stiffness, but not torsional inertia. CBAR elements provide axial, torsional, bending, and shear stiffnesses but do not provide torsional inertia. Shells provide stiffness in all DOFs except for in the in-plane rotational DOF. Solids provide stiffness in only the 3 translational DOFs. If a DOF of a grid is unconstrained or if there is no contribution of stiffness from any element, then it has no stiffness and hence is singular, in which case an SPC must be automatically added using AUTOSPC.

AUTOSPC scans the stiffness of the 6 DOFs at each grid. The stiffness ratio is the ratio of stiffness in the weakest direction to that in the strongest direction, considering all possible directions rather than just the coordinate direction. Translational and rotational stiffnesses are considered separately. NASTRAN then compares the STIFFNESS RATIOS with a failure criterion, to decide whether the DOF is singular and an SPC should be applied. At each grid, a 3 x 3 partition of the stiffness matrix for each of the three translational and three rotational DOFs is solved as an eigenvalue problem to determine the principal stiffnesses. Each stiffness term is compared to the principal stiffness using the formula

$$\varepsilon = \frac{K_{ii}}{K_{max}}$$

where *Kii* is the term in the *i*-th row and *i*-th column of the matrix and *Kmax* is the principal stiffness. If ε is less than the value of PARAM, EPZERO, the global direction nearest *i* is considered singular. The default value for EPZERO is 10^{-8}, but this can be adjusted using EPZERO, although this should be changed with great care.

G R I D P O I N T S I N G U L A R I T Y T A B L E

POINT ID	TYPE	FAILED DIRECTION	STIFFNESS RATIO	OLD USET EXCLUSIVE	OLD USET UNION	NEW USET EXCLUSIVE	NEW USET UNION	
53	G	4	0.00E+00	B	F	SB	S	*
53	G	5	0.00E+00	B	F	SB	S	*

The default AUTOSPC is YES is all solution schemes except for SOL 106 and SOL 129.

```
PARAM, AUTOSPC, YES
PARAM, EPZERO, < Value 1.0E-8 by default >
```

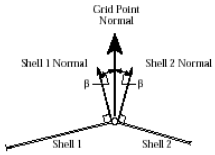
In a CBAR assemblage with all the stiffness (axial, torsion, bending in 2 directions and optionally shear in 2 directions) properties specified, no AUTOSPC is required as there are 6 stiffness contributions to every DOF. In CBAR assemblage with missing stiffness properties or likewise in CROD assemblages, certain DOFs may have to be suppressed as there may be no stiffness contribution. In the limit, if a single DOF spring solitarily connects a node, all other DOFs of the node will be constrained except for the DOF in which the spring acts. Likewise, rotational DOFs of solids and in-plane rotational DOFs of shells are constrained. Solids have no rotational DOFs and shells do not have in plane rotational DOFs. This is because of the nature of shells providing in plane rotational stiffness through the in plane translational stiffnesses of many elements within the surface mesh and solids providing rotational stiffness about the three axes through the translational stiffnesses of many elements within the volume mesh.

The DOFs constrained by the program must be checked, since over-constraining can cause incorrect results. The potentiality of over-constraining is usually associated with 1D elements. **Constraining the in-plane rotational DOFs of shells and all 3 rotational DOFs of solids do not generally over-constrain the model as the rotational stiffnesses (and associated deformations) in 2D and 3D elements, by their very nature, are provided by the stiffness of the many translational DOFs.**

The potentiality of over-constraining a finite element model consisting of 1D elements shall be investigated thoroughly. To this end, it is crucial to know the nature of the load path in the structure being modeled. All applied loads must be balanced ultimately by the reactions through certain types of effects (which appear as internal forces and stresses) within the structure. There are 4 ways of carrying load, namely in axial force, in torsion, in bending moment and in shear force. These lead to deformations. An axial force causes axial deformation, torsion causes torsional deformation, bending moments cause bending deformations and shear forces cause shear deformations. **If it is envisaged that a particular load path does not involve significant bending moment and shear force, that load path can be modeled with a CROD element.** (Note that CBAR elements become CROD elements if both ends are pinned in bending rotations (not torsional) or if a very small value of bending stiffness is inputted such that the bending moment becomes very small. A pinned ended beam element is akin to having a zero-length rigid link between the connecting members in all the DOFs except the rotational DOF that corresponds to the bending moment). The main nature of the load path on outrigger columns on a sky-scrapper is in axial force. CROD elements may be sufficient in this case. The same argument applies for using CROD elements to model a truss where the load path is primarily axial in nature. However, CROD elements or pin-ended CBAR elements provide no rotational stiffness to the DOFs. To avoid a singular matrix, these DOFs are artificially constrained using AUTOSPC. The user must then ensure that these lost loads into these artificial constraints are small enough to be ignored. The applied loads are no longer totally balanced by the actual support reactions, but some energy is lost into the artificial constraints. If the load path is primarily axial, the loads lost into these artificial constraints will be small. **But if a significant force is lost into the artificial constraint, the model is deemed to be over-constrained.** The load path is not primarily axial in nature. CROD elements should hence not be used to model a load path where there are significant bending and shear actions as there will be no bending stiffness to transfer the loads. This will cause a mechanism (singular matrix) to occur unless an automatic constraint is placed by AUTOSPC. If however, CBAR elements are used with bending stiffness contribution in the outrigger and the truss examples, the potentiality of there being some bending action is accounted for, just in case. It will of course be observed that the bending moments and shear forces in the outriggers and the truss CBAR elements to be small if the load path is primarily axial in nature. No singularities exist when CBAR elements (with axial, torsion, bending and optionally shear stiffnesses) are used. Akin to the outrigger and truss examples, applying a concentrated moment on a cable in reality will result in a mechanism. That is to say, applying a concentrated moment about the common grid adjoining two collinear CROD elements will be futile, as the DOF will be constrained by AUTOSPC, as it has no stiffness to resist the moment. The moment load is lost into the constraint, again because the anticipated load path was not modeled by the element. If however, a CBAR element of very low stiffness is added to the grid, such that AUTOSPC does not constrain the DOF, the application of the concentrated moment will cause the displacements to be very large, as the resistance is very low. Likewise, torsional stiffness in CBAR elements can often be ignored if there is deemed to be little torsional forces i.e. that torsion is an insignificant load path. If a DOF has no torsional stiffness component, then the DOF component will be automatically constrained by the code. But if a torsion load path is intended, the torsion will be lost into the artificial constraint, making the structure incorrectly too stiff. If on the other hand, the stiffness in that DOF component is not too small in comparison with the other DOFs in that particular grid such that AUTOSPC is not invoked, the torsional forces will cause large torsional deformations within the element. **To conclude, if a DOF is without stiffness and is not constrained by the user, then a grid point singularity will exist unless the code enforces an artificial restraint by AUTOSPC. If the DOF is part of a critical load path, then loads will be lost into the artificial constraint instead of being carried by the action of the structural elements and ultimately being transferred into the real supports. If instead the DOF has a low stiffness, but not low enough to be constrained by AUTOSPC, then the matrix will not be singular (although likely to be quite ill-conditioned depending on the precision of the machine), but the applied loads will cause a large deflection as the resistance is low.**

1.3.24.1.2 K6ROT and SNORM

For linear solutions (i.e. all except SOL 106 and SOL 129), the default K6ROT is 0.0, whilst for nonlinear analysis the default is 100.0 (this is opposite to the defaults for AUTOSPC). K6ROT is intended primarily for geometric nonlinear analysis. For highly nonlinear solutions a value of 1.E5 has been used successfully. NASTRAN internally inspects an appropriate rotational stiffness value to be applied, this value is then multiplied by K6ROT. For most instances, the default is appropriate. **There may be instances where K6ROT is beneficial in linear analysis.** Shell elements do not have stiffness in rotation about an axis normal to the shell. In models consisting of thousands of shell without reinforcement (e.g. car models) this can produce a very long list of singularities, making checking tedious. This may be solved with K6ROT, which removes the singularity (and prevents AUTOSPC) by specifying a small value of stiffness in the rotational degree of freedom. In other words, a **grounded** in-plane rotational spring is inserted to remove the singularity. Secondly, for linear analysis, if a surface is flat, then there is no reason why we cannot allow AUTOSPC to automatically constraint the in-plane rotational freedoms. If however, the surface is only slightly curved (say 179° between adjacent shell elements), then letting AUTOSPC constrain the freedoms may be unacceptable, as the model may not pass the rigid body checks (GROUNDCHECK). In this case, K6ROT with a value of 1.0 may be deemed appropriate.




```
PARAM, K6ROT, < 0.0 to 100.0 >
```

In linear analysis, instead of using K6ROT, it is better to use PARAM, SNORM. By default, the direction of the normal rotation vector for flat plate elements is assumed to be perpendicular to the plane of each element. If the model is curved, the shell bending and twist moments must change direction at the element intersection. If transverse shear flexibility is present, the deformations may be too large. (Because elements using low-order formulations ignore the edge effect, this rarely causes any problems-a large value of the parameter K6ROT partially cures the problem). With the unique normal (SNORM) option, the rotational degrees of freedom at each corner of an element are measured relative to the specified normal vector direction. Thus, all elements connected to a grid point will use a consistent direction for defining shell bending and twisting moments. Shell normals are available for CQUAD4, CQUADR, CTRIA3, and CTRIAR elements. Normals are activated if the actual angle between the local element normal and the unique grid point normal is less than 20.0°, the default value. The default can be changed by setting SNORM to the desired real value up to 89°. The unique grid point normal is the average of all local shell element normals at a specific grid point. Generated grid point normals may be overwritten by user-defined normals. A shell normal defines a unique direction for the rotational degrees of freedom of all adjacent elements. A shell normal vector is created by averaging the normal vectors of the attached elements. If the actual angle is greater than the value defined on PARAM, SNORM, (default = 20.0) the edge is assumed to be a corner, and the old method is used. **Shell normals improve the accuracy of the results in curved shells where in-plane shear and twisting moments act together**. The type of structure that exhibits the most change in results is a thick curved shell with large in-plane shear forces and twisting moments. Most other problems, such as flat plates and curved shells with pressure loads, show changes in results of less than 1%. More degrees of freedom may be constrained using this improved formulation. This formulation results in zero in-plane rotational stiffness values. The CQUAD8 and CTRIA6 elements are not included in the shell normal processing. If they are modeled correctly, they do not require shell normal processing. Connecting these elements to the lower-order flat elements is not recommended. Transverse shear flexibility (MID3 on the PSHELL property entry) should be left on when normals are used.

```
PARAM, SNORM, 20.0 $ Default value is 20.0
```

In linear solutions, PARAM, K6ROT, 0.0 and PARAM, SNORM, 20.0 are recommended. Transverse shear flexibility (MID3 of PSHELL) should be included. AUTOSPC should also be switched on (as is the default in linear analysis). K6ROT can be used in linear analysis to reduce the output in the Grid Point Singularity Table due to AUTOSPC, simply for ease of checking the automatic constraint DOFs normally more critical for 1D elements. It can also be used with a value of 1.0 in linear analysis when the shell surface is only slightly curved. However, K6ROT with values of around 100.0 is intended primarily for nonlinear analysis.

1.3.24.2 Mechanism Singularity of Stiffness Matrix (MAXRATIO, BAILOUT)

A mechanism type singularity requires the consideration of stiffness at more than one grid point (contrast with grid point singularity which considers stiffness at each grid). MAXRATIO checks for

I. **global mechanism singularity,** and for
II. **local mechanisms singularity**

within the system. AUTOSPC may have accounted for all the local *grid point* singularities (of beam grids without certain stiffness properties, in plane rotational freedom of shell grids and all rotational freedoms of solid grids), but it remains to be seen if the system is restrained **globally from rigid body motion** or if there are **local mechanisms**.

Mathematically, a mechanism type singularity is detected during decomposition, based on the maximum ratio of the matrix diagonal to the factor diagonal

$$\text{MAXRATIO} = \frac{K_{ii}}{D_{ii}}$$

Log_{10} MAXRATIO indicates how many significant digits may have been lost during the decomposition.

where K_{ii} is the i-th diagonal term of the original stiffness matrix and D_{ii} is the i-th diagonal term of the factor diagonal matrix. Note that for a symmetric matrix K it can be represented as

$$[K] = [L][D][L^T]$$

$[L]$ = lower triangular factor
$[D]$ = factor diagonal matrix

If for any row i, this ratio is greater then MAXRATIO, then the matrix will be considered nearly singular (having mechanisms). If any diagonal terms of the factor are negative, the stiffness matrix will be implausible (non-positive definite). **The exact grid point failing the criteria is not significant for the global rigid-body mechanisms and may change with small changes in the model; The exact grid point failing the criteria is significant for the local mechanism.** NASTRAN then takes appropriate action depending on the BAILOUT parameter. If negative, NASTRAN is directed to continue processing. PARAM, BAILOUT should really therefore **NOT BE USED** (it should be specified 0 to terminate the analysis if near singularities are detected), instead of failing the MAXRATIO, the modelling should be corrected.

```
$ BULK DATA
PARAM, MAXRATIO, < Value 1.0E5 by default >
PARAM, BAILOUT, < 0 or -1 >
```

```
 THE FOLLOWING DEGREES OF FREEDOM HAVE FACTOR DIAGONAL RATIOS GREATER THAN 1.00000E+05 OR HAVE
NEGATIVE TERMS ON THE FACTOR DIAGONAL.
      GRID POINT ID       DEGREE OF FREEDOM   MATRIX/FACTOR DIAGONAL RATIO          MATRIX DIAGONAL

           6714                 T1                -7.19297E+15                      6.02908E+07

 *** DMAP FATAL MESSAGE 9050 (SEKRRS) -  RUN TERMINATED DUE TO EXCESSIVE PIVOT RATIOS
     IN MATRIX KLL. USER PARAMETER BAILOUT MAY BE USED   TO CONTINUE THE RUN.
```

Should be positive and less than 1.0E5

First, consider the **global mechanism**. For a **fully coupled structure**, there can be 6 global rigid body motions, which must all be constrained. **Non fully coupled portions of the structure** exhibiting rigid body motion can also be regarded as a global mechanism. Translational rigid body motion can be constrained by the user applying the 3 DOF translational constraints at a particular grid or at multiple grids (of course not all 3 DOF constraints need to be applied at the very same grid, so long as the structure cannot exhibit rigid body translation on the whole). As for rotational rigid body motion, the manner in which constraints need to be applied to restrain the rigid body modes depend on the type of model, whether beams, shells or solids. **Beam models** can be completely restrained against rotational rigid body motion by applying the 3 rotational DOF constraints at **any one grid**. **Shell models** can be restrained against the 2 out-of-plane rotational rigid body motion by applying the 2 rotational DOF constraints at any one grid. However, the in plane rotational rigid body motion **CANNOT BE** constrained by applying a in-plane rotation restraint at a grid or multiple grids (as AUTOSPC does) because individual shell elements do not have any

in-plane rotational stiffness in general. But of course, the shell mesh as a whole does have in plane rotational stiffness. Thus, the in plane rotational rigid body motion can only be suppressed by a set of translational DOFs in the two directions in the plane of the shells applied at different grids. Alternatively, a set of grids can be connected to a grid with an RBE2, which is then suppressed in its rotational freedoms. Likewise, with solid models, all 3 rotational rigid body motions **CANNOT BE** suppressed by applying rotational DOF constraints at a grid or multiple grids (as AUTOSPC does) because individual solid elements have not any rotational stiffness. But of course, the solid mesh as a whole does have rotational stiffnesses about the 3 axes. These are suppressed by a set of translational DOFs in the three directions applied at different grids. Alternatively, a set of grids can be connected to a grid with an RBE2, which is then suppressed in its rotational freedoms. On all three type of models, be it beam, shell or solid models, a set of translational DOF constraints in the 3 directions applied at different grids will act to constrain the rotational rigid body motion. It must be remembered of course, that these application of constraints will act to alter the behaviour of the model and hence should be representative of reality.

Second, consider **local mechanisms**. Local mechanisms refer to **individual finite elements**. Local mechanisms occur when an element if not properly connected to the main structure. That is to say, its stiffnesses are not fully coupled into the main structure. It is best to illustrate with examples. A local mechanism will exist if a CBAR or CBEAM has no torsional stiffness whilst its adjacent *collinear* beams have torsional stiffness. The local beam may exhibit rigid body motion, as the torsional stiffness (and hence the ability to prevent the spinning about the axis) of the element is not attached onto the main structure. This would not be prevented by AUTOSPC because there is a stiffness contribution to the torsional DOF from the adjacent elements. A local mechanism also occurs when a CBAR or CBEAM is pinned in all three rotational DOFs on both ends, irrespective of whether it is collinear of not with adjacent beams. Note that to model trusses, only the two bending rotational DOFs are released, not the torsional DOF. This is perfectly acceptable. Likewise, pinned ended (in the two bending rotation DOFs) beams on multi-storey buildings can be used so long as the torsional DOF is not also released. But if the torsional stiffness was forgotten on either the truss or the multi-storey building, then the beam can spin about its axis. This also occurs when all 3 rotation DOFs of the two ends of the beam are released. The about-axis rotation motion of the beam is not connected onto the main structure and hence is a local mechanism. This would not be prevented by AUTOSPC because there are contributions to the rotational stiffness from adjacent elements, which are not pinned ended. Next, a local mechanism can occur if shells elements without bending properties (MID2 on PSHELL) are used. This however may be prevented by AUTOSPC. A local mechanism can also occur if a CBAR or CBEAM element connects into shells elements without special modelling techniques to transfer the in-plane (of the shell) moment. Hence if a 1D element connects to the surface of the shell mesh, there is no coupling of the in-plane rotational stiffness, and hence the 1D element is a mechanism as it can spin about the axis normal to the shell surface. This would not be prevented by AUTOSPC as the beam provides the rotational stiffness. An RBE2 or RBE3 connection should instead be employed. A local mechanism also occurs when a CBAR or CBEAM connects onto one grid of a solid mesh, even if the other end of the CBAR or CBEAM is also connected onto the solid mesh. Again there is nothing to stop the beam from spinning about its axis as the rotational motion is not coupled into the solid mesh. This would not be prevented by AUTOSPC as the beam provides the 3 rotational stiffnesses. An RBE2 or RBE3 connection should be employed. Next, a local mechanism occurs also when a shell mesh is connected into a solid mesh without proper modelling techniques to connect the out-of-plane (of the shell) rotational stiffnesses. This would not be prevented by AUTOSPC as the shell mesh provides the out-of-plane (of the shell) rotational stiffnesses.

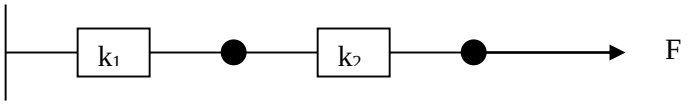
Global mechanisms are best pin-pointed by running a SOL 103 (with AUTOSPC possibly set to the non-default value of NO) and visually inspecting the modes of vibration that have zero (or close to zero) frequency. This will highlight the global rigid body motion of the entire structure and also the non fully coupled portions of the structure as mechanisms. Local mechanisms on the other hand will cause a SOL 103 to crash and the DOFs with a critical MAXRATIO will be presented in the .f06 file for inspection and subsequent attention.

1.3.24.3 Grid Point Singularities (AUTOSPC, K6ROT) and Local and Global Mechanism Singularity (MAXRATIO, BAILOUT) for Linear, Nonlinear, Static and Dynamic Solutions

Solution	Grid Point Singularity	Local and Global Mechanism Singularity
Linear static SOL 101 Linear buckling SOL 105	AUTOSPC, YES K6ROT, 0.0.	Local mechanisms (identified with critical MAXRATIOs) may cause the solution scheme to crash, failing which the results would be suspect. Global mechanisms cannot be modeled, the structure must be at least statically determinate by applying sufficient zero or prescribed displacement boundary conditions. Local and global mechanisms show up as critical MAXRATIOs.
Nonlinear static SOL 106	AUTOSPC, NO K6ROT, 100.0	Local mechanisms (identified with critical MAXRATIOs) may cause the solution scheme to diverge, failing which the results would be suspect. Global mechanisms (identified by a SOL 103 with AUTOSPC possibly NO) cannot be modeled because convergence to a static equilibrium state can never be achieved, hence all parts of the structure must be at least statically determinate by applying sufficient zero or prescribed displacement boundary conditions.
Linear modal SOL 103	AUTOSPC, YES K6ROT, 0.0.	Local mechanisms (identified with critical MAXRATIOs) may cause the solution scheme to crash, failing which the results would be suspect. Global mechanisms can be modeled (possibly with having to set AUTOSPC to the non-default value of NO), showing up as zero frequency modes. Intentional or unintentional global mechanisms can be readily visible, inspected and attended to.
Linear dynamic SOL 109, SOL 112, SOL 108, SOL 111	AUTOSPC, YES K6ROT, 0.0.	Local mechanisms (identified with critical MAXRATIOs) may cause the solution scheme to crash, failing which the results would be suspect. Global mechanisms can be modeled as long as the unrestrained degree of freedom has defined mass terms. Note that since the analysis is linear, the stiffness and damping is based on the initial undeflected geometry. Must **eliminate unintentional global mechanisms**, otherwise the mechanism will continue vibrating, possibly inducing incorrect results.
Nonlinear dynamic (implicit) SOL 129	AUTOSPC, NO K6ROT, 100.0.	Local mechanisms (identified with critical MAXRATIOs) may cause the solution scheme to diverge, failing which the results would be suspect. Global mechanisms can be modeled as long as the unrestrained degree of freedom has defined mass terms. Must **eliminate unintentional global mechanisms**, otherwise the mechanism will continue vibrating, possibly inducing incorrect results.
Nonlinear dynamic (explicit) LS-DYNA	None	Local mechanisms will introduce errors. Global mechanisms can be modeled as long as the unrestrained degree of freedom has defined mass terms. Must **eliminate unintentional global mechanisms**, otherwise the mechanism will continue vibrating, possibly inducing incorrect results.

1.3.24.4 Ill-Conditioning of Stiffness Matrix (EPSILON, Load and Reaction Discrepancy PRTRESLT)

Singularities occur when there is no stiffness (or very low stiffness) in a DOF component *relative to other DOF components* at a particular node. The contribution of stiffness may come from any element. The relative contribution of the adjacent elements is immaterial. On the other hand, ill-conditioning occurs when the relative stiffness contribution in a *particular DOF component* at a node from adjacent elements is large. For illustration, take an example of two springs in series of stiffness k_1 and k_2. Assume $k_2 >> k_1$.



Two equations can be written for this 2-DOF system, namely

$$(k_1 + k_2)\, u_1 + (-k_2)\, u_2 = 0$$

$$(-k_2)\, u_1 + (k_2)\, u_2 = F$$

which can be solved simultaneous for the two displacement unknowns. Getting rid of u_2 from the two equations without performing any arithmetic, the displacement for u_1 is obtained as

$$u_1 = F / (k_1 + k_2 - k_2)$$

Now if $k_2 >> k_1$ then there will be error $e = k_2 - k_2$ (not zero due to numerical inaccuracy) introduced in the calculation of u_1. And because the value of the second k_2 is large compared to k_1, e would be comparable in magnitude to k_1. Subsequently, the force in the first element will be calculated as

$$f_1 = k_1 u_1 = F\, k_1 / (k_1 + e)$$

when in reality it should be

$$f_1 = k_1 u_1 = F$$

Hence, ill-conditioning occurs when there is large differences in stiffness between adjacent DOF components at a particular grid point. A definite indication of ill-conditioning is when the sum of loading at a particular load due to explicit nodal loads and also equivalent element loads is not equal the sum of the element end forces for all the elements connected to that particular node. Codes generally perform an out-of-balance check to detect ill-conditioned matrices. In MSC.NASRAN, the condition of overall stiffness matrix is ascertained using EPSILON in the .f06 file. If all the stiffness terms in the matrix are of the same order of magnitude then the numerical calculations carried out NASTRAN on the matrix are easy. However, if there are very large differences between the largest and smallest stiffness terms then the accuracy of the analysis will be impaired due to limitations of computer numerical accuracy. The matrix is then considered to ill-conditioned. EPSILON is based on a strain energy error ratio. The EPSILON value should theoretically be equal to zero, but due to computer accuracy limitations it usually in the order of 10^{-7} to 10^{-8}. Large values indicate that the stiffness matrix is ill-conditioned. A system of linear equations is said to be ill-conditioned if small perturbations in the system lead to large changes in the solution. NASTRAN checks for evidence of ill-conditioning in the system of equations representing the structural model. An EPSILON value of 1 would indicate that parts of the model are disconnected, or that there is a mechanism in the structure. Hence an EPSILON value of 1 would be obtained if the model fails the MAXRATIO check, i.e. the global singularity (rigid body motion) or local mechanism check.

The upper limit for EPSILON is a matter of judgment. Usually, analyses producing **EPSILON values below 10^{-9} are considered acceptable**. A large value of epsilon (greater than about 10^{-3}) is an indication of numerical ill-conditioning and requires further investigation. It does not matter whether epsilon is positive or negative, as long as it is small. In addition, a value of, say, 10^{-12} is not a "better" value than 10^{-10}; both are "small enough".

```
*** USER INFORMATION MESSAGE 5293 FOR DATA BLOCK KLL

LOAD SEQ. NO.     EPSILON            EXTERNAL WORK   EPSILONS LARGER THAN .001 ARE FLAGGED WITH ASTERISKS
        1       -4.5961966E-14      5.6341639E+00
        2       -9.2748066E-15      9.4523830E+00
```

After decomposition, a singularity may lead to an incorrect solution. In static analysis, MSC.NASTRAN solves

$$Ku = P$$

to obtain *u* (displacements). Using these displacements, a “residual” loading vector is calculated as follows:

$$Ku - P = \delta P$$

This residual vector should theoretically be null but may not be null due to numeric roundoff. To obtain a normalized value of the residual loading, an error measure EPSILON is calculated by

$$\varepsilon = \frac{u^T \cdot \delta P}{u^T \cdot P}$$

Alternatively, ill-conditioning can be ascertained by inspecting the discrepancy between the applied forces and reactions. The following **PRTRESLT** parameter prints the resultant of the applied loads (OLOAD RESULTANTS), the resultant of the restraints (SPCFORCE RESULTANTS and MPCFORCE RESULTANTS) all about the point specified by PARAM, GRDPRT, N, the default being the origin of the basic coordinate system.

$ BULK DATA
PARAM, PRTRESLT, YES

OLOAD RESULTANT

SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
1	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
	FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
	FZ	----	----	1.000000E+02	0.000000E+00	-5.000000E+02	----
	MX	----	----	----	0.000000E+00	----	----
	MY	----	----	----	----	0.000000E+00	----
	MZ	----	----	----	----	----	0.000000E+00
	TOTALS	0.000000E+00	0.000000E+00	1.000000E+02	0.000000E+00	-5.000000E+02	0.000000E+00
2	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
	FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
	FZ	----	----	3.307500E+02	0.000000E+00	-8.268750E+02	----
	MX	----	----	----	0.000000E+00	----	----
	MY	----	----	----	----	0.000000E+00	----
	MZ	----	----	----	----	----	0.000000E+00
	TOTALS	0.000000E+00	0.000000E+00	3.307500E+02	0.000000E+00	-8.268750E+02	0.000000E+00

SPCFORCE RESULTANT

SUBCASE/ DAREA ID	LOAD TYPE	T1	T2	T3	R1	R2	R3
1	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
	FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
	FZ	----	----	-1.000000E+02	0.000000E+00	0.000000E+00	----
	MX	----	----	----	0.000000E+00	----	----
	MY	----	----	----	----	5.000000E+02	----
	MZ	----	----	----	----	----	0.000000E+00
	TOTALS	0.000000E+00	0.000000E+00	-1.000000E+02	0.000000E+00	5.000000E+02	0.000000E+00
2	FX	0.000000E+00	----	----	----	0.000000E+00	0.000000E+00
	FY	----	0.000000E+00	----	0.000000E+00	----	0.000000E+00
	FZ	----	----	-3.307500E+02	0.000000E+00	0.000000E+00	----
	MX	----	----	----	0.000000E+00	----	----
	MY	----	----	----	----	8.268750E+02	----
	MZ	----	----	----	----	----	0.000000E+00
	TOTALS	0.000000E+00	0.000000E+00	-3.307500E+02	0.000000E+00	8.268750E+02	0.000000E+00

The EQUILIBRIUM Case Control Command can also be used to perform the equilibrium balance check.

To conclude, ill-conditioning indicated by high values of EPSILON is caused by a high difference in stiffness between adjacent elements in the model (the solution of which would obviously be to use rigid elements such as RBE2 connecting the corresponding stiffness to model the much stiffer element) and by large mass in dynamic analysis whilst ill-conditioning indicated by an EPSILON value of 1.0 is caused when the model fails the MAXRATIO check, i.e. the global singularity (rigid body motion) or local mechanism check.

1.3.24.5 Grid Point Weight Generator

The total mass and inertia of the model can be checked by specifying the GRDPNT, N parameter with respect to the grid point N, a value of N = 0 denoting the origin of the model coordinate frame being used.

```
$ BULK DATA
PARAM, GRDPNT, < 0 for about origin or specify a Grid ID >
```

```
      O U T P U T   F R O M   G R I D   P O I N T   W E I G H T   G E N E R A T O R
                                REFERENCE POINT =        0
                                          M O
 *  1.300000E+01  0.000000E+00  0.000000E+00  0.000000E+00  5.000000E+00 -3.000000E+00 *
 *  0.000000E+00  1.300000E+01  0.000000E+00 -5.000000E+00  0.000000E+00  7.000000E+00 *
 *  0.000000E+00  0.000000E+00  1.300000E+01  3.000000E+00 -7.000000E+00  0.000000E+00 *
 *  0.000000E+00 -5.000000E+00  3.000000E+00  8.000000E+00 -1.500000E+00 -2.500000E+00 *
 *  5.000000E+00  0.000000E+00 -7.000000E+00 -1.500000E+00  1.000000E+01  0.000000E+00 *
 * -3.000000E+00  7.000000E+00  0.000000E+00 -2.500000E+00  0.000000E+00  8.000000E+00 *
                                          S
                    *  1.000000E+00  0.000000E+00  0.000000E+00 *
                    *  0.000000E+00  1.000000E+00  0.000000E+00 *
                    *  0.000000E+00  0.000000E+00  1.000000E+00 *
        DIRECTION
    MASS AXIS SYSTEM (S)     MASS              X-C.G.        Y-C.G.        Z-C.G.
           X            1.300000E+01      0.000000E+00  2.307692E-01  3.846154E-01
           Y            1.300000E+01      5.384616E-01  0.000000E+00  3.846154E-01
           Z            1.300000E+01      5.384616E-01  2.307692E-01  0.000000E+00
                                          I(S)
                    *  5.384615E+00 -1.153847E-01 -1.923079E-01 *
                    * -1.153847E-01  4.307692E+00 -1.153846E+00 *
                    * -1.923079E-01 -1.153846E+00  3.538461E+00 *
                                          I(Q)
                    *  5.503882E+00                            *
                    *                5.023013E+00               *
                    *                             2.703873E+00 *
                                           Q
                    *  8.702303E-01  4.915230E-01  3.323378E-02 *
                    *  3.829170E-01 -7.173043E-01  5.821075E-01 *
                    *  3.099580E-01 -4.938418E-01 -8.124324E-01 *
```

1.3.24.6 Displacement Compatibility Check

Displacement compatibility refers the fact that the displacement shape functions are continuous between adjacent elements. For instance, the corner and mod-side nodes of adjacent elements must be connected and so a mesh transition is required for a mesh of linear elements to mesh into parabolic elements. To ensure displacement compatibility between adjacent elements, a preprocessor should be used to perform a visual inspection on a ***shrink plot*** of all the elements from their nodes. Missing elements may be detected visually.

More effectively, the pre-processor should be used to plot the ***free ends (1d mesh), free edges (2d mesh) and free faces (for 3D mesh)*** of the model. Unintentional holes and incompatibility within the model will clearly show up in the display. A real modal analysis (assuming mass correctly incorporated) can also be performed to ensure that all parts of the model are properly connected. Or, a 1g i.e. gravity load static analysis in the 3 directions (in 3 separate subcases) can be performed (assuming mass correctly incorporated) to check for displacement incompatibilities.

To *zip up* the unconnected grids that are coincident (to a tolerance applicable to model), the ***equivalence*** function of the pre-processor may be used. However, ensure that intentional coincident grids that model scalar stiffness, scalar damping or hinges are not *equivalenced*.

1.3.24.7 Element Quality Checks

It has been mentioned that the benefits of isoparametric element formulations is that curved geometries can be mapped more accurately because the geometry is also described by the shape functions. However, the geometry must not be too distorted, whether in its initial undeflected configuration or at a later stage of a nonlinear analysis. The parameter that suffers from badly distorted configurations is the Jacobian | J |. The Jacobian maps the natural coordinates to the global coordinates in the stiffness expression of the elements. Element distortions generally stem from excessive variations in the Jacobian between different nodes. Codes tend to (based on experience) place bounds on the permissible variation of the Jacobian over a particular element between different nodes. Re-entrant angles (where the corner node of a QUAD8 is squeezed into the element beyond the mid-point must be avoided as the will cause the Jacobian to change sign. As the corner node approaches the mid-point, | J | → 0, causing infinite strains. Even in an undistorted QUAD8, if the mid-side node were beyond the middle third, this would cause the Jacobian to vanish at a certain node and hence again produce infinite strains. There are four possible forms of element distortion, namely: -

(i) Aspect ratio distortion
(ii) Angular distortion, i.e. skew and taper
(iii) Volumetric distortion
(iv) Mid node position distortion

All these distortions can be quantified as some terms in the Jacobian going to zero, causing a numerical singularity arising solely from the mapping function. It can lead to large errors in the stress results but has a lesser effect on the computed displacements. These numerical singularities have been used in elements modelling cracks. The default element quality checks in MSC.NASTRAN are presented as follows. If any of these are violated, then the analysis will crash. The following are the default MSC.NASTRAN element quality values which should be ensured within the pre-processor. A few of these default values can be changed using the GEOMCHECK Executive command.

Name	Value Type	Default	Comment
Q4_SKEW	Real≥0.0	30.0	Skew angle in degrees
Q4_TAPER	Real≥0.0	0.50	Taper ratio
Q4_WARP	Real≥0.0	0.05	Surface warping factor
Q4_IAMIN	Real≥0.0	30.0	Minimum Interior Angle in degrees
Q4_IAMAX	Real≥0.0	150.0	Maximum Interior Angle in degrees
T3_SKEW	Real≥0.0	10.0	Skew angle in degrees
T3_IAMAX	Real≥0.0	160.0	Maximum Interior Angle in degrees
TET_AR	Real≥0.0	100.0	Longest edge to shortest edge aspect ratio
TET_EPLR	Real≥0.0	0.50	Edge point length ratio
TET_DETJ	Real	0.0	| J | minimum value
TET_DETG	Real	0.0	| J | minimum value at vertex point
HEX_AR	Real≥0.0	100.0	Longest edge to shortest edge aspect ratio
HEX_EPLR	Real≥0.0	0.50	Edge point length ratio
HEX_DETJ	Real	0.0	| J | minimum value
HEX_WARP	Real≥0.0	0.707	Face warp coefficient
PEN_AR	Real≥0.0	100.0	Longest edge to shortest edge aspect ratio
PEN_EPLR	Real≥0.0	0.50	Edge point length ratio
PEN_DETJ	Real	0.0	| J | minimum value
PEN_WARP	Real≥0.0	0.707	Quadrilateral face warp coefficient
BEAM_OFF	Real≥0.0	0.15	CBEAM element offset length ratio
BAR_OFF	Real≥0.0	0.15	CBAR element offset length ratio

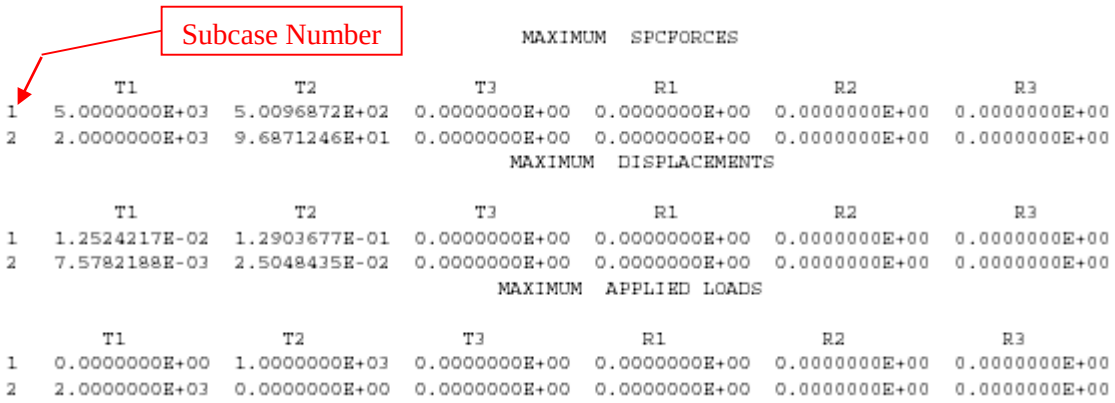
Evaluates location of mid-side nodes

The results are printed under USER INFORMATION MESSAGE 7555.

1.3.24.8 Maximum Values

The summary of maximum displacements, loads and SPCs and MPCs all in the basic coordinate system can be requested using the following parameter.

```
$ BULK DATA
PARAM, PRTMAXIM, YES
```



MAXIMUM SPCFORCES

	T1	T2	T3	R1	R2	R3
1	5.0000000E+03	5.0096872E+02	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00
2	2.0000000E+03	9.6871246E+01	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00

MAXIMUM DISPLACEMENTS

	T1	T2	T3	R1	R2	R3
1	1.2524217E-02	1.2903677E-01	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00
2	7.5782188E-03	2.5048435E-02	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00

MAXIMUM APPLIED LOADS

	T1	T2	T3	R1	R2	R3
1	0.0000000E+00	1.0000000E+03	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00
2	2.0000000E+03	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00

1.3.24.9 Element Orientation Check

The element **X-, Y- and Z- orientation for beams** must be consistent as stress recovery is presented in the element coordinate system, the **Z-normal for shells** must be consistent but the element X- and Y- orientation can be random as long as the stress recovery is presented in the global coordinate system, and for solids a random X-, Y- and Z- element orientation is fine as long as the stress recovery is presented in the global coordinate system.

1.3.24.10 Duplicate Grid and Element Check

A pre-processor should be equipped with the ability to detect duplicate grids or elements, be it beams, shells or solids, as visual inspection is impossible. Unintentional duplicate grids occur when an equivalence operation was forgotten. Duplicate grids may however be intentional for the modelling of scalar elements CELAS, CBUSH or rigid elements RBE2. Duplicate elements may occur if a line, surface or volume was accidentally meshed more than once.

1.3.24.11 Element Summary Table

The ELSUM Case Control command provides a quick summary table of properties for the elements in the model. These properties include element-id, material-id, length (or thickness), area, volume, structural mass, non-structural mass, total mass, and weight (weight mass * total mass).

0

E L E M E N T P R O P E R T Y S U M M A R Y

0ELEMENT TYPE = ROD

ID	MID	LENGTH	AREA	VOLUME	SM	NSM	TM	WEIGHT
401	202	1.000000E+01	6.000000E+00	6.000000E+01	4.800000E+02	5.000000E+00	4.850000E+02	4.850000E+02
402	201	1.000000E+01	4.000000E+00	4.000000E+01	1.600000E+02	8.000000E+00	1.680000E+02	1.680000E+02
403	202	1.000000E+01	6.000000E+00	6.000000E+01	4.800000E+02	5.000000E+00	4.850000E+02	4.850000E+02
SUBTOTAL MASS =					1.120000E+03	1.800000E+01	1.138000E+03	1.138000E+03
TOTAL MASS =					1.120000E+03	1.800000E+01	1.138000E+03	1.138000E+03

1.3.24.12 Unconstrained Equilibrium Check (Grounding Check)

1.3.24.12.1 Enforced Displacement Method

This is a superb method of evaluating some modelling errors that cause unintentional grounding of the structure. If a structure is truly unconstrained, then movement at one point of this structure should cause the whole structure to move as a rigid body. The following procedure can be used to perform this check.

1. Remove all the constraints
2. Apply a unit enforced displacement in the x-direction at one selected grid point while constraining the other five components at this selected grid point to zero. This grid point should be close to the center of gravity of the structure, although not a requirement. The x-displacements at all the other grid points should also be equal to unity if the structure is truly unconstrained. If the displacement at a certain grid point is not equal to unity in the x-direction, it is very likely that it overstrained. The likely causes for this can include **incorrect modelling of rigid elements**, **offset beams**, **non-collinear CELASi**.
3. Repeat the same procedure for the other two orthogonal directions.

At the conclusion of this check, you should remember to put your original constraints back into your model.

1.3.24.12.2 Automated Rigid Body Method

In addition enforced displacement method, you may also request a more robust and more automatic grounding check of the stiffness matrix. This grounding check is also referred to as rigid body check because the stiffness matrix is multiplied by the rigid body transformation matrix. The grounding check identifies unintentional constraints and ill conditioning in the stiffness matrix. Automatic grounding check is requested using the GROUNDCHECK Case Control command.

$$\text{GROUNDCHECK}\left[\left(\left[\begin{matrix}\text{PRINT}\\ \text{NOPRINT}\end{matrix}\right], \text{PUNCH}, \text{SET} = \left\{\begin{matrix}\text{G, N, N + AUTOSPC, F, A}\\ \text{ALL}\end{matrix}\right\}\right)\right.$$
$$\left.\text{GRID} = \text{gid}, \text{THRESH} = \text{e}, \text{DATAREC} = \left[\begin{matrix}\text{YES}\\ \text{NO}\end{matrix}\right], (\text{RTHRESH} = \text{r})\right] = \left\{\begin{matrix}\text{YES}\\ \text{NO}\end{matrix}\right\}$$

For example, the following command GROUNDCHECK = YES will request a grounding check of the g-set. The following command GROUNDCHECK(SET=ALL) = YES will request a grounding check of all DOF sets. The check may be performed at any or all stages of the stiffness reduction based on the specification of the SET keyword.

SET Keyword	DOF Set	Description
G	g-set	before single point, multipoint constraints, and rigid elements are applied
N	n-set	after multipoint constraints and rigid elements are applied
N+AUTO	n-set with AUTOSPC	same as the n-set with the rows/columns in the stiffness matrix corresponding to degrees-of-freedom constrained by the PARAM,AUTOSPC operation zeroed out
F	f-set	after single point, multipoint constraints, and rigid elements are applied
A	a-set	after static condensation

User Information Message 7570 is issued by GROUNDCHECK for each DOF set requested. The strain energy is computed in each direction of the rigid body motion and by default, if the strain energy exceeds the tolerance, then “FAIL” is printed out for that directory. The tolerance is set by dividing the largest stiffness term by 1.E10.

The THRESH keyword specifies the maximum strain energy that passes the grounding check. DATAREC=YES requests data recovery of grounding forces. The RTHRESH=r keyword prints the grounding forces larger than r percent of the largest grounding force if DATAREC is set to YES.

Here is an example for the g-set that indicates the stiffness matrix passes the grounding check in all six rigid body directions.

```
RESULTS OF RIGID BODY CHECKS OF MATRIX KGG      (G-SET)  FOLLOW:
     PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF   2.725275E-04
           DIRECTION        STRAIN ENERGY        PASS/FAIL
           ---------        -------------        ---------
             1               1.484295E-09          PASS
             2               2.182787E-10          PASS
             3               1.637090E-11          PASS
             4               1.619810E-10          PASS
             5               2.727802E-10          PASS
             6               1.054841E-07          PASS
     SOME POSSIBLE REASONS MAY LEAD TO THE FAILURE:
       1. CELASI ELEMENTS CONNECTING TO ONLY ONE GRID POINT;
       2. CELASI ELEMENTS CONNECTING TO NON-COINCIDENT POINTS;
       3. CELASI ELEMENTS CONNECTING TO NON-COLINEAR DOF;
       4. IMPROPERLY DEFINED DMIG MATRICES;
```

Here is an example for the f-set that indicates the stiffness matrix fails the grounding check in all six rigid body directions.

```
*** USER INFORMATION MESSAGE 7570 (GPWG1D)
    RESULTS OF RIGID BODY CHECKS OF MATRIX KFF      (F-SET)  FOLLOW:
    PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF   1.464858E-04
          DIRECTION        STRAIN ENERGY        PASS/FAIL
          ---------        -------------        ---------
            1               2.564102E+05          FAIL
            2               7.326008E+05          FAIL
            3               2.237437E+03          FAIL
            4               6.057062E+02          FAIL
            5               4.015165E+03          FAIL
            6               8.747863E+04          FAIL
    SOME POSSIBLE REASONS MAY LEAD TO THE FAILURE:
      1. CONSTRAINTS WHICH PREVENT RIGID-BODY MOTION.
```

If the DATAREC keyword is specified and the rigid body check fails then data recovery will be performed to compute and print the grounding forces to make it easy to locate the source of the failure. By default, only those grounding forces larger than 10 percent of the largest grounding force will be printed (see RTHRESH keyword). Here is an example of grounding forces created by moving the model in rigid body direction 1 associated with the f-set failure shown above.

```
 DIRECTION        1
                          G R O U N D   C H E C K   F O R C E S  ( F - S E T )

      POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
              3      G      2.564102E+05  -2.884615E+05    .0             .0             .0             .0
              4      G      2.564102E+05   2.884615E+05    .0             .0             .0             .0
```

1.3.24.12.3 Thermal Equilibrium Method

Check against grounding is especially necessary if the structure is subjected to thermal loading. The thermal equilibrium method can actually be used to check against grounding as follows.

1. Remove the actual boundary conditions and apply a set of statically determinate constraints. Typically this procedure is done by constraining all six DOFs at a single grid point. If a single grid point is used, make sure this single grid point contains six degrees of freedom. A single grid point of a model consisting of all solid elements, for example, cannot satisfy this requirement since each grid point of a solid element contains only three degrees of freedom.
2. Change all the thermal coefficients of expansion to a single value.
3. Apply a uniform ΔT to the structure.

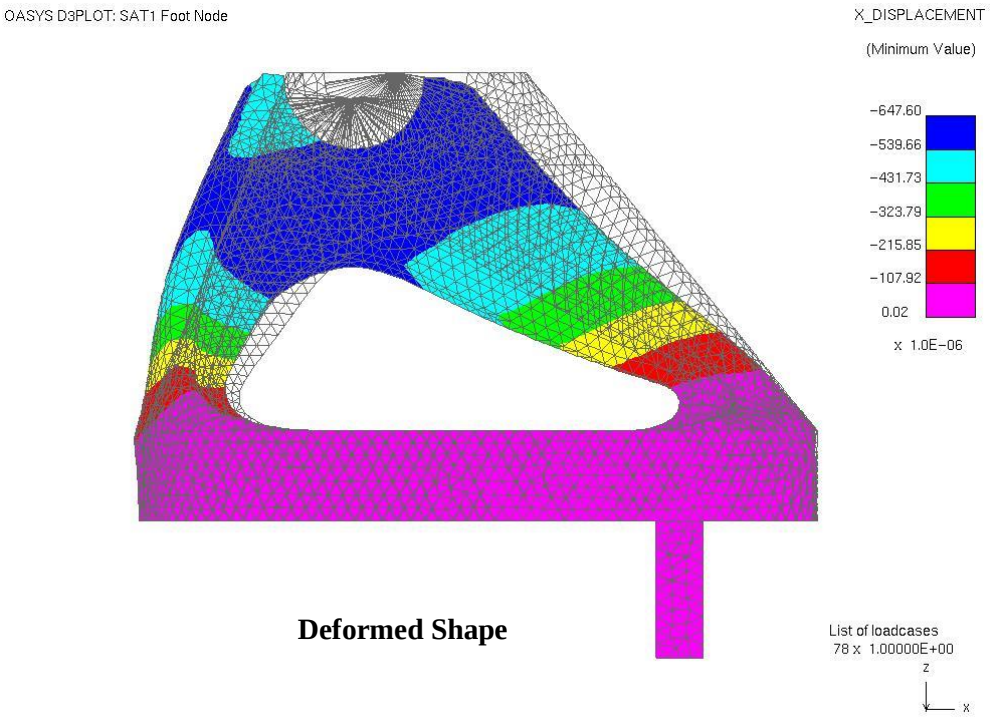
If the model is "clean," then the structure should be strain free; in other words, there should be no reaction loads, element forces, or stresses. If this is not the case, then you may want to investigate around the vicinity where the element forces or stresses are nonzero. Incorrect modelling of rigid elements or offsets is a common cause of these types of errors. Once you are satisfied with your model, remember to change the boundary condition, thermal coefficients of expansion, and ΔT back to their original values.

1.3.24.12.4 Load OLOAD and Reaction SPCFORCE Discrepancy using PARAM, PRTRESLT

Internal constraints (due to non-colinear CELASi) can also be detected when there is a discrepancy between the OLOAD resultants and SPCFORCE resultants.

1.3.24.13 Deformed Shape

It is very illustrative to inspect the deformed shape DISP (PLOT) in order to ensure that the model correctly simulates the intentions.



Deformed Shape

ASCII inspection is also recommended and is requested by DISP (PRINT).

D I S P L A C E M E N T V E C T O R

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	.0	.0	.0	.0	.0	.0
2	G	.0	.0	.0	.0	.0	.0
3	G	7.578219E-03	-2.504843E-02	.0	.0	.0	.0
4	G	-2.421781E-03	-2.495156E-02	.0	.0	.0	.0

1.3.24.14 Massless Mechanisms

A massless mechanism mode is defined as a shape that causes very small strain energy when the structure is deflected in this shape. This shape also causes very little kinetic energy when the structure moves in this shape. The ratio of strain energy over kinetic energy is proportional to the eigenvalue of this mode. This result is an indeterminate quantity produced by two terms approaching zero. If applied loads excite this shape small loads can cause very large deflections. If a real mode analysis can be forced to produce a solution in the presence of this condition the mode produced with this shape can have any eigenvalue from positive infinity to negative infinity. These frequencies may change drastically for small changes in the model, or when run on a different computer. Eigensolution failures due to Sturm number inconsistencies are often due to the presence of massless mechanisms. The presence of such modes reduces the reliability and repeatability of eigensolutions and dynamic analysis solutions.

A method is now provided to automatically constrain massless mechanisms for eigensolutions, controlled by the user parameter PARAM,MECHFIX,AUTO, which is turned on by default in MSC.Nastran. When this method is used it is unlikely that the eigensolution will fail because of three failed shifts, and the solution produced will be repeatable for small changes in the model, loading condition, or computer type used in the solution.

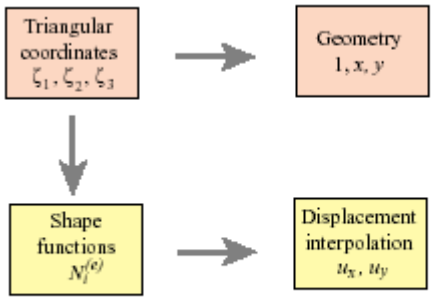
1.3.25 Isoparametric Elements and Numerical Integration [4]

Most finite elements used in commercial codes are either isoparametric, super-parametric or sub-parametric elements. These elements can map highly distorted curvilinear geometries. An isoparametric element maps the geometry with the same shape functions defining the displacement function. A super-parametric element maps the geometry with a higher order shape function than that which defines the displacement function. Finally, a sub-parametric element maps the geometry with a lower order shape function than that which defines the displacement function. Super-parametric and isoparametric representation is illustrated for the 3-noded triangle. The super-parametric representation is presented as follows.

$$\begin{bmatrix} 1 \\ x \\ y \end{bmatrix} = \begin{bmatrix} 1 & 1 & 1 \\ x_1 & x_2 & x_3 \\ y_1 & y_2 & y_3 \end{bmatrix} \begin{bmatrix} \zeta_1 \\ \zeta_2 \\ \zeta_3 \end{bmatrix},$$

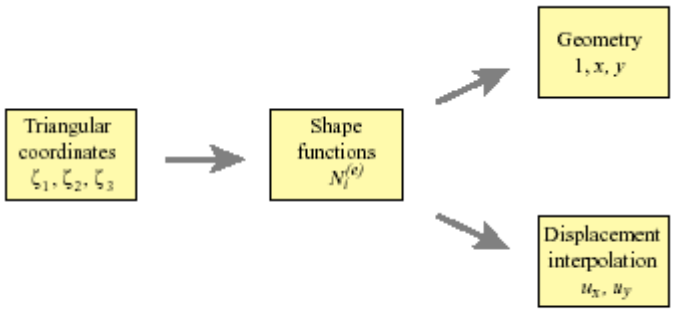
$$u_x = u_{x1} N_1^{(e)} + u_{x2} N_2^{(e)} + u_{x3} N_3^{(e)} = u_{x1}\zeta_1 + u_{x2}\zeta_2 + u_{x3}\zeta_3$$
$$u_y = u_{y1} N_1^{(e)} + u_{y2} N_2^{(e)} + u_{y3} N_3^{(e)} = u_{y1}\zeta_1 + u_{y2}\zeta_2 + u_{y3}\zeta_3$$

In the super-parametric representation, the triangular coordinates define the element geometry through the first set of relationships. The second set of relationships show the displacement expansion defined by the shape functions, which are in turn expressed in terms of the triangular coordinates. Evidently the element geometry and element displacements are not treated equally. If we proceed to higher order triangles with straight sides, only the displacement expansion is refined whereas the geometry definition remains the same. Elements built according to this prescription are called *super-parametric*, a term that emphasizes that unequal treatment.



Superparametric representation of triangular element.

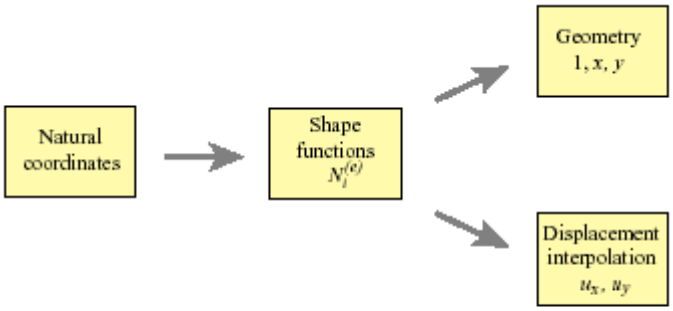
The key idea is to use the shape functions to represent *both the element geometry and the problem unknowns*, which in structural mechanics are displacements. Hence the name *isoparametric element* ("iso" means equal), often abbreviated to *iso-P element*.



Isoparametric representation of triangular elements.

[4] FELIPPA, Carlos A. *Introduction to Finite Element Methods.* University of Colorado, 2001.

To generalize to include other two-dimensional elements, the term triangular coordinates are replaced by the general term natural coordinates. Natural coordinates (triangular coordinates for triangles, quadrilateral coordinates for quadrilaterals) appear as *parameters* that define the shape functions. The shape functions then connect the geometry with the displacements.



Isoparametric representation of arbitrary 2D elements: triangles or quadrilaterals. For 3D elements, expand the geometry list to $\{1, x, y, z\}$ and the displacements to $\{u_x, u_y, u_z\}$.

The generalization to an arbitrary two-dimensional element with *n* nodes is thus presented. Two sets of relations, one for the element geometry and the other for the element displacements, are required. Both sets exhibit the same interpolation in terms of the shape functions.

Geometric relations:

$$1 = \sum_{i=1}^{n} N_i^{(e)}, \quad x = \sum_{i=1}^{n} x_i N_i^{(e)}, \quad y = \sum_{i=1}^{n} y_i N_i^{(e)}.$$

Displacement interpolation:

$$u_x = \sum_{i=1}^{n} u_{xi} N_i^{(e)}, \quad u_y = \sum_{i=1}^{n} u_{yi} N_i^{(e)}.$$

Combining both sets of expressions

$$\begin{bmatrix} 1 \\ x \\ y \\ u_x \\ u_y \end{bmatrix} = \begin{bmatrix} 1 & 1 & \dots & 1 \\ x_1 & x_2 & \dots & x_n \\ y_1 & y_2 & \dots & y_n \\ u_{x1} & u_{x2} & \dots & u_{xn} \\ u_{y1} & u_{y2} & \dots & u_{yn} \end{bmatrix} \begin{bmatrix} N_1^{(e)} \\ N_2^{(e)} \\ \vdots \\ N_n^{(e)} \end{bmatrix}.$$

Thus for the linear (3-noded) triangular element

$$\begin{bmatrix} 1 \\ x \\ y \\ u_x \\ u_y \end{bmatrix} = \begin{bmatrix} 1 & 1 & 1 \\ x_1 & x_2 & x_3 \\ y_1 & y_2 & y_3 \\ u_{x1} & u_{x2} & u_{x3} \\ u_{y1} & u_{y2} & u_{y3} \end{bmatrix} \begin{bmatrix} N_1^{(e)} \\ N_2^{(e)} \\ N_3^{(e)} \end{bmatrix}.$$

The shape functions are simply the triangular coordinates

$$N_1^{(e)} = \zeta_1, \qquad N_2^{(e)} = \zeta_2, \qquad N_3^{(e)} = \zeta_3.$$

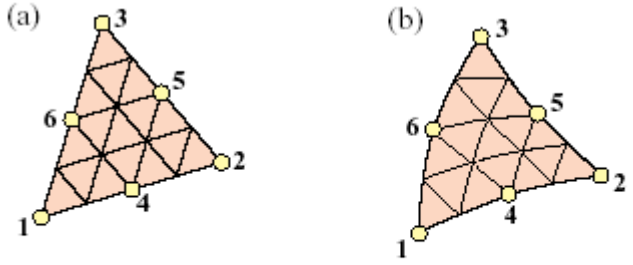
Since the resulting expression is the same as that produced by the super-parametric approach, the linear triangle is actually both super-parametric and isoparametric. It is in fact the only triangle that is both super-parametric and isoparametric. The 6-noded triangular element has the following geometric and displacement interpolation

$$\begin{bmatrix} 1 \\ x \\ y \\ u_x \\ u_y \end{bmatrix} = \begin{bmatrix} 1 & 1 & 1 & 1 & 1 & 1 \\ x_1 & x_2 & x_3 & x_4 & x_5 & x_6 \\ y_1 & y_2 & y_3 & y_4 & y_5 & y_6 \\ u_{x1} & u_{x2} & u_{x3} & u_{x4} & u_{x5} & u_{x6} \\ u_{y1} & u_{y2} & u_{y3} & u_{y4} & u_{y5} & u_{y6} \end{bmatrix} \begin{bmatrix} N_1^{(e)} \\ N_2^{(e)} \\ N_3^{(e)} \\ N_4^{(e)} \\ N_5^{(e)} \\ N_6^{(e)} \end{bmatrix}.$$

where the shape functions are

$$N_1^{(e)} = \zeta_1(2\zeta_1 - 1), \quad N_2^{(e)} = \zeta_2(2\zeta_2 - 1), \quad N_3^{(e)} = \zeta_3(2\zeta_3 - 1),$$
$$N_4^{(e)} = 4\zeta_1\zeta_2, \quad N_5^{(e)} = 4\zeta_2\zeta_3, \quad N_6^{(e)} = 4\zeta_3\zeta_1.$$

The isoparametric version can thus have curved sides defined by the location of the mid-side nodes.



The six-node quadratic triangle: (a) the superparametric version, with straight sides and midside nodes at midpoints; (b) isoparametric version.

The natural coordinates for a triangular element are the triangular coordinates ζ_1, ζ_2 and ζ_3. The natural coordinates for a quadrilateral element are ξ and η. These are called *quadrilateral coordinates*. These coordinates vary from -1 on one side to +1 at the other. This particular variation range (instead of, say, 0 to 1) was chosen by the investigators who originally developed isoparametric quadrilaterals to facilitate the use of the standard Gauss integration formulas. The isoparametric 4-noded quadrilateral has the following geometric and displacement interpolation

$$\begin{bmatrix} 1 \\ x \\ y \\ u_x \\ u_y \end{bmatrix} = \begin{bmatrix} 1 & 1 & 1 & 1 \\ x_1 & x_2 & x_3 & x_4 \\ y_1 & y_2 & y_3 & y_4 \\ u_{x1} & u_{x2} & u_{x3} & u_{x4} \\ u_{y1} & u_{y2} & u_{y3} & u_{y4} \end{bmatrix} \begin{bmatrix} N_1^{(e)} \\ N_2^{(e)} \\ N_3^{(e)} \\ N_4^{(e)} \end{bmatrix}.$$

where the shape functions are

$$N_1^{(e)} = \tfrac{1}{4}(1-\xi)(1-\eta), \qquad N_2^{(e)} = \tfrac{1}{4}(1+\xi)(1-\eta),$$
$$N_3^{(e)} = \tfrac{1}{4}(1+\xi)(1+\eta), \qquad N_4^{(e)} = \tfrac{1}{4}(1-\xi)(1+\eta).$$

The element equilibrium equations can be written for a geometrically linear element to be

$$\{f\} = \left[\int_\Omega [B^A]^T [D^A][B^A] d\Omega\right]\{d\} + \left\{\int_\Omega [B^A]^T \{\sigma\}_i d\Omega\right\} - \left\{\int_\Omega [D^A]\{\varepsilon\}_i d\Omega\right\} - \left\{\int_\Omega [N]^T \{b\} d\Omega\right\}$$

Exact integration of the above stiffness matrix and the fixed end forces often cannot be done analytically. Constant strain triangles (CST) have constant [B] and [D] matrices and are a rare exception to the rule since their stiffness and fixed end force matrices can be integrated in closed form. For the quadrilateral, the matrix [B] often depends on the location of the coordinates. This occurs because the shape functions once differentiated are still a function of the natural coordinates.

The derivation of the stiffness terms is presented for the isoparametric quadrilateral. Partial derivatives of shape functions with respect to the Cartesian coordinates *x* and *y* are required for the strain and stress calculations ultimately to derive the stiffness matrix. Since the shape functions are not directly functions of *x* and *y* but of the natural coordinates ξ and η, the determination of Cartesian partial derivatives is not trivial. We require the Jacobian of two-dimensional transformations that connect the differentials of {*x, y*} to those of {ξ,η} and vice-versa.

$$\begin{bmatrix} dx \\ dy \end{bmatrix} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix} \begin{bmatrix} d\xi \\ d\eta \end{bmatrix} = \mathbf{J} \begin{bmatrix} d\xi \\ d\eta \end{bmatrix}, \quad \begin{bmatrix} d\xi \\ d\eta \end{bmatrix} = \begin{bmatrix} \frac{\partial \xi}{\partial x} & \frac{\partial \eta}{\partial x} \\ \frac{\partial \xi}{\partial y} & \frac{\partial \eta}{\partial y} \end{bmatrix} \begin{bmatrix} dx \\ dy \end{bmatrix} = \mathbf{J}^{-1} \begin{bmatrix} dx \\ dy \end{bmatrix},$$

in which

$$\mathbf{J} = \frac{\partial(x, y)}{\partial(\xi, \eta)} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix}, \qquad \mathbf{J}^{-1} = \frac{\partial(\xi, \eta)}{\partial(x, y)} = \begin{bmatrix} \frac{\partial \xi}{\partial x} & \frac{\partial \eta}{\partial x} \\ \frac{\partial \xi}{\partial y} & \frac{\partial \eta}{\partial y} \end{bmatrix}$$

Hence, the shape function derivatives in terms of the quadrilateral coordinates are

$$\frac{\partial N_i^{(e)}}{\partial x} = \frac{\partial N_i^{(e)}}{\partial \xi}\frac{\partial \xi}{\partial x} + \frac{\partial N_i^{(e)}}{\partial \eta}\frac{\partial \eta}{\partial x},$$
$$\frac{\partial N_i^{(e)}}{\partial y} = \frac{\partial N_i^{(e)}}{\partial \xi}\frac{\partial \xi}{\partial y} + \frac{\partial N_i^{(e)}}{\partial \eta}\frac{\partial \eta}{\partial y}.$$

or in matrix form

$$\begin{bmatrix} \frac{\partial N_i^{(e)}}{\partial x} \\ \frac{\partial N_i^{(e)}}{\partial y} \end{bmatrix} = \begin{bmatrix} \frac{\partial \xi}{\partial x} & \frac{\partial \eta}{\partial x} \\ \frac{\partial \xi}{\partial y} & \frac{\partial \eta}{\partial y} \end{bmatrix} \begin{bmatrix} \frac{\partial N_i^{(e)}}{\partial \xi} \\ \frac{\partial N_i^{(e)}}{\partial \eta} \end{bmatrix} = \frac{\partial(\xi, \eta)}{\partial(x, y)} \begin{bmatrix} \frac{\partial N_i^{(e)}}{\partial \xi} \\ \frac{\partial N_i^{(e)}}{\partial \eta} \end{bmatrix} = \mathbf{J}^{-1} \begin{bmatrix} \frac{\partial N_i^{(e)}}{\partial \xi} \\ \frac{\partial N_i^{(e)}}{\partial \eta} \end{bmatrix}.$$

These terms feature in the strain matrix [B] which ultimately yields the stiffness matrix [K] for the element. The symbolic inversion of **J** for arbitrary ξ and η in general leads to extremely complicated expressions unless the element has a particularly simple geometry. This complexity was one of the factors that motivated the use of numerical integration techniques. The use of numerical integration is thus essential for evaluating element integrals of isoparametric elements. The standard practice has been to use *Gauss integration* (also known as Gauss-Legendre quadrature) because such rules use a *minimal number of sample points to achieve a desired level of accuracy*.

The Gauss integration rules in one dimension are

$$\int_{-1}^{1} F(\xi)\, d\xi \approx \sum_{i=1}^{p} w_i F(\xi_i).$$

where p is the number of Gauss integration points (equal of more than one), w_i are the integration weights and ξ_i are the sample point abscissas in the interval –1 to 1. Hence the first four one dimensional Gauss rules are

$$\text{One point:} \quad \int_{-1}^{1} F(\xi)\, d\xi \approx 2F(0),$$

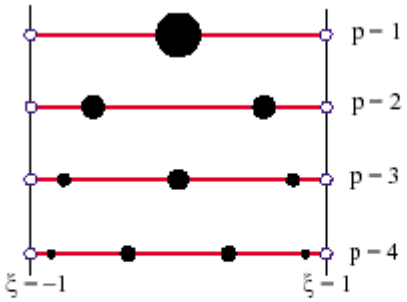
$$\text{Two points:} \quad \int_{-1}^{1} F(\xi)\, d\xi \approx F(-1/\sqrt{3}) + F(1/\sqrt{3}),$$

$$\text{Three points:} \quad \int_{-1}^{1} F(\xi)\, d\xi \approx \frac{5}{9}F(-\sqrt{3/5}) + \frac{8}{9}F(0) + \frac{5}{9}F(\sqrt{3/5}),$$

$$\text{Four points:} \quad \int_{-1}^{1} F(\xi)\, d\xi \approx w_1 F(\xi_1) + w_2 F(\xi_2) + w_3 F(\xi_3) + w_4 F(\xi_4).$$

For the 4-point rule, $\xi_3 = -\xi_2 = \sqrt{(3 - 2\sqrt{6/5})/7}$, $\xi_4 = -\xi_1 = \sqrt{(3 + 2\sqrt{6/5})/7}$, $w_1 = w_4 = \frac{1}{2} - \frac{1}{6}\sqrt{5/6}$, and $w_2 = w_3 = \frac{1}{2} + \frac{1}{6}\sqrt{5/6}$.

The four rules integrate exactly polynomials in a ξ of orders up to 1, 3, 5 and 7, respectively. In general a one-dimensional Gauss rule with *p* points integrates exactly polynomials of order up to $2p - 1$. This is called the *degree* of the formula.



The first four one-dimensional Gauss rules $p = 1, 2, 3, 4$ depicted over the line segment $\xi = -1$ to $\xi = +1$. Gauss point locations are marked with black circles. The radii of these circles are proportional to the integration weights.

A more general integral, such as $F(x)$ over $[a, b]$ in which $\ell = b - a > 0$, is transformed to the canonical interval $[-1, 1]$ as follows:

$$\int_a^b F(x)\, dx = \int_{-1}^{1} F(\xi)\, J\, d\xi,$$

in which ξ and J are defined by the mapping $x = \frac{1}{2}a(1 - \xi) + \frac{1}{2}b(1 + \xi) = \frac{1}{2}(a + b) + \frac{1}{2}\ell\xi$, that is, $\xi = (2/\ell)(x - \frac{1}{2}(a + b))$ and $J = dx/d\xi = \frac{1}{2}\ell$.

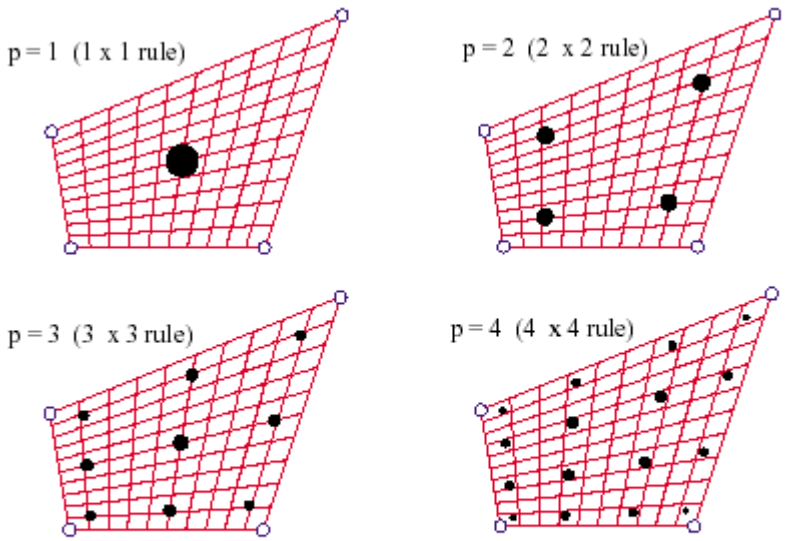
The simplest two-dimensional Gauss rules are called *product rules*. They are obtained by applying the one-dimensional rules to each independent variable in turn. To apply these rules we must first reduce the integrand to the canonical form.

$$\int_{-1}^{1}\int_{-1}^{1} F(\xi,\eta)\,d\xi\,d\eta = \int_{-1}^{1} d\eta \int_{-1}^{1} F(\xi,\eta)\,d\xi.$$

Once this is done we can process numerically each integral in turn.

$$\int_{-1}^{1}\int_{-1}^{1} F(\xi,\eta)\,d\xi\,d\eta = \int_{-1}^{1} d\eta \int_{-1}^{1} F(\xi,\eta)\,d\xi \approx \sum_{i=1}^{p_1}\sum_{j=1}^{p_2} w_i w_j F(\xi_i,\eta_j).$$

where p_1 and p_2 are the number of Gauss points in the ξ and η directions, respectively. Usually the same number $p = p_1 = p_2$ is chosen if the shape functions are taken to be the same in the ξ and η directions.



The first four two-dimensional Gauss product rules $p = 1, 2, 3, 4$ depicted over a straight-sided quadrilateral region. Gauss points are marked with black circles. The areas of these circles are proportional to the integration weights.

Hence, if the stiffness matrix is

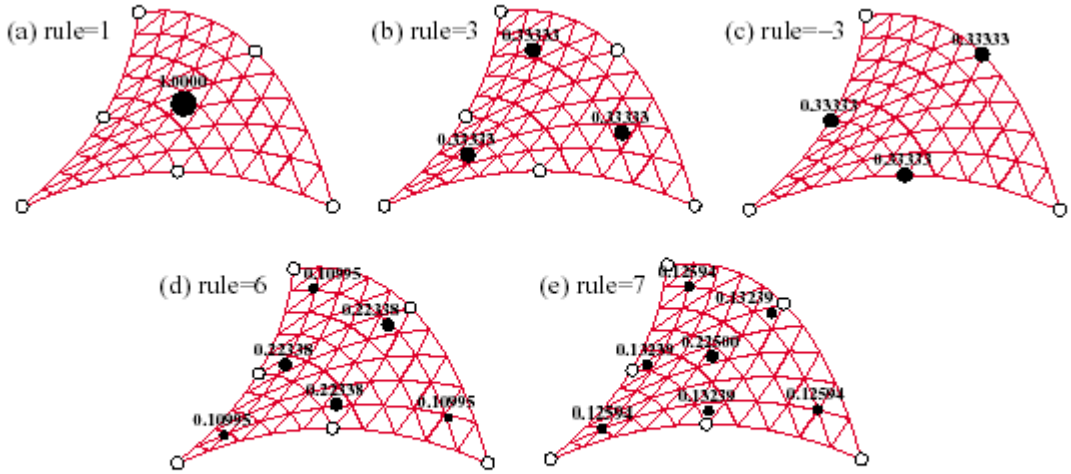
$$\mathbf{K}^{(e)} = \int_{\Omega^{(e)}} h\,\mathbf{B}^T\mathbf{E}\mathbf{B}\,d\Omega^{(e)}$$

Reducing the integrand into canonical form

$$\mathbf{K}^{(e)} = \int_{-1}^{1}\int_{-1}^{1} \mathbf{F}(\xi,\eta)\,d\xi\,d\eta. \qquad d\Omega^{(e)} = dx\,dy = \det\mathbf{J}\;d\xi\,d\eta. \qquad \mathbf{F}(\xi,\eta) = h\,\mathbf{B}^T\mathbf{E}\mathbf{B}\det\mathbf{J}.$$

This matrix function can be numerically integrated over the domain $-1 \le \xi \le +1$, $-1 \le \eta \le +1$ by an appropriate Gauss product rule. For square 4-noded quadrilaterals, the integrand $h\,\mathbf{B}T\,\mathbf{EB}J$ is at most quadratic in ξ and η, and hence 2 x 2 Gauss integration suffice to compute the integral exactly (i.e. the element is fully integrated and not under-integrated). Using a higher order Gauss integration rule, such as 3 x 3 and 4 x 4, reproduces exactly the same stiffness matrix produced by the 2 x 2 integration rule. Using a 1 x 1 rule yields a rank-deficiency matrix. For a non-square quadrilateral, there is little difference in the stiffness matrix beyond the 2 x 2 Gauss integration, with higher Gauss integration orders producing a slightly stiffer response (and hence slightly higher element natural frequencies).

As for the 6-noded triangle, the same stiffness matrix is obtained for integration rule 3, rule -3 or rule 7 as long as the triangular shape is maintained and the mid-side nodes are exactly at the mid-point between corner nodes. A highly distorted triangle on the other hand will return varying stiffness matrices for different Gauss integration rules.



Location of sample points (dark circles) of five Gauss quadrature rules for curved sided 6-node triangles. Weight written to 5 places near each sample point; sample-point circle areas are proportional to weight.

A reduced integrated element (Gauss integration of a lower p order) may suffer from matrix singularities as there are insufficient stiffness terms unlike a fully integrated element which will have all its stiffness terms calculated. Under-integrated elements may thus suffer from *hourglassing* where the deformed shape shows clear zig-zag patterns indicating DOFs which are not stiff.

However, often it is usually advantages to use **A REDUCED INTEGRATION TECHNIQUE (i.e. the minimum integration requirements that PRESERVES THE RATE OF CONVERGENCE which would result if exact integration were used)** as, for very good reasons, **A CANCELLATION OF ERRORS due to discretization and due to inexact integration** can occur [5].

[5] ZIENKIEWICZ, O.C. & TAYLOR, R.L. *The Finite Element Method. The Basis. Volume 1*. 5th Edition. Butterworth-Heinemann, Oxford, 2000.

1.3.26 Element and Nodal Stress Recovery [6]

Stress recovery concepts involve

I. stress recovery at the element Gauss points, then
II. subsequent stress extrapolation to the element grids, and
III. eventually, the (transformation into global coordinates and) averaging of the elemental stresses to yield the grid point stresses

1.3.26.1 Stress Recovery at Gauss Points

In the stiffness method, the stresses are obtained from the computed displacements, and are thus *derived quantities*. The accuracy of derived quantities is generally lower than that of primary quantities (the displacements). For example, if the accuracy level of displacements is 1% that of the stresses may be typically 10% to 20%, and even lower at boundaries. It is therefore of interest to develop techniques that enhance the accuracy of the computed stresses. These procedures receive the generic name *stress recovery* techniques in the finite element literature.

Once the global displacements {U} solved for, and hence the elemental displacements {d} derived, the elemental strain and stress can be obtained from

$$\{\varepsilon\} = [B]\{d\}$$
$$\{\sigma\} = [D][B]\{d\} + \{\sigma\}_i - [D]\{\varepsilon\}_i$$

It is of interest to evaluate and report these stresses at the *element grid* points located on the corners and possibly midpoints of the element. These are called *element grid point stresses*. It is important to realize that the stresses computed at the same grid point from adjacent elements *will not generally be the same*, since stresses are not required to be continuous in displacement assumed finite elements. This suggests some form of stress averaging can be used to improve the stress accuracy, and indeed this is part of the stress recovery technique. The results from this averaging procedure are called *grid point stresses*. Hence to summarize there are the *independent element Gauss point stresses*, the *independent element grid point stresses* and the *dependent grid point stresses.*

To obtain the *independent element grid point stresses*, two approaches are possible, namely

(a) Evaluate directly the stress at the element grid locations by substituting the natural coordinates of the grid points as arguments to the shape functions.
(b) Evaluate the stress at the Gauss integration points used in the element stiffness integration rule and then extrapolate to the element grid points.

Empirical evidence indicates that the second approach generally delivers better stress values for *quadrilateral* elements whose geometry departs substantially from the rectangular shape. This is backed up by superconvergence (i.e. values sampled at these points show an error which decreases more rapidly than elsewhere) results in finite element approximation theory. Stresses sampled at superconvergent points converge at the same rate as the unknown function or displacement and hence have errors of order $O(h^{p+1})$. For rectangular elements there is no difference. For isoparametric *triangles* both techniques deliver similar results (identical if the elements are straight sided with mid-side nodes at midpoints) and so the advantages of the second one are marginal.

Hence, on many occasions the displacement or the unknown function is most accurately sampled at the nodes defining the element and the gradients or stresses are best sampled at some interior points. The displacement or unknown function and the stress or the gradients will be exact at the nodes and within the element if the shape function contains the exact theoretical solution to the governing differential equation. When the shape function is of a polynomial of lower order or simply does not contain the exact theoretical solution, the displacement or the unknown function is most accurately sampled at the nodes than anywhere else whilst the stress or the gradients is most accurate at some interior point depending on the order of the element. This can be explained qualitatively as

[6] FELIPPA, Carlos A. *Introduction to Finite Element Methods.* University of Colorado, 2001.

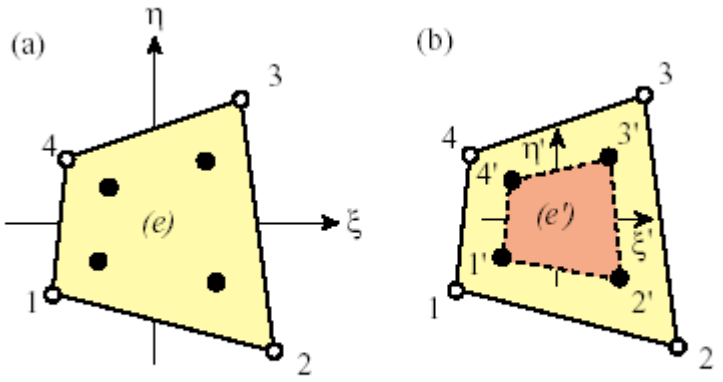
follows. Linear elements such as QUAD4 and TRIA3 have constant first gradient or stress values. If the true stress distribution is linear, the constant FE representation will be exact at the middle of the element. For the quadratic elements of QUAD8 and TRIA6 (with linear FE representation of stress), the 2-point Gauss-Legendre will provide superconvergent sampling of stress. In general, the best accuracy is obtainable for gradients or stresses at the Gauss points corresponding, in order, to the polynomial used in the solution of the unknown function. As far as the optimal points for gradient or stress recovery is concerned, it was observed experimentally by Barlow and proved by Herrmann, that the optimal (not exact because of isoparametric distortion) superconvergent points are the Gauss-Legendre points. This almost always corresponds to the locations of the Gauss-Legendre points for reduced integration (i.e. integration whilst still maintaining the rate of convergence if fully integrated).

1.3.26.2 Extrapolation of Stress From Gauss Points to Element Grids

Now that the optimum internal stresses are obtained i.e. at the Gauss points, it is necessary to extrapolate these to the corner and possibly the middle of the element in order to obtain a complete picture. Stress contours are calculated by interpolation involving an extrapolation from the Gauss points to the grids. This means that for element with high stress gradients the interpolated values will be less accurate than for elements with low stress gradients. It is important to check to see what variation of stress exists over the element and if it is large it may be necessary to refine the mesh locally to overcome this problem. Within an isoparametric finite element, stresses are calculated at Gauss points, and are converted from these Gauss point values to grid values.

1.3.26.2.1 Bilinear Extrapolation

For QUAD4, had the stresses chosen to be computed at the (**non-optimum**) 2 x 2 Gauss integration points, each stress component is "carried" to the corner nodes and the center of the element through a bilinear extrapolation based on the computed values at the Gauss points. The optimum superconvergent Gauss point for a QUAD4 is at the center. Here we are only demonstrating the principle of bilinear extrapolation. To understand the extrapolation procedure more clearly it is convenient to consider the region bounded by the Gauss points as an "internal element" or "Gauss element".



. Extrapolation from 4-node quad Gauss points: (a) 2 × 2 rule, (b) Gauss element (e')

The Gauss element, is also a four-node quadrilateral. Its quadrilateral (natural) coordinates are denoted by ξ' and η'. These ai

$$\xi = \xi'/\sqrt{3}, \quad \eta = \eta'/\sqrt{3}, \qquad \xi' = \xi\sqrt{3}, \quad \eta' = \eta\sqrt{3}.$$

Any scalar quantity w whose values w_i' at the Gauss element corners are known can be interpolated through the usua of ξ' η'

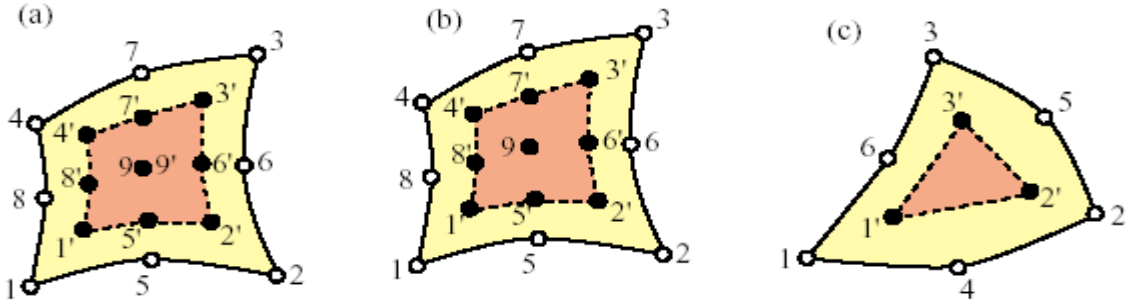
$$w(\xi', \eta') = [\, w'_1 \quad w'_2 \quad w'_3 \quad w'_4 \,] \begin{bmatrix} N_1^{(e')} \\ N_2^{(e')} \\ N_3^{(e')} \\ N_4^{(e')} \end{bmatrix},$$

$$N_1^{(e')} = \tfrac{1}{4}(1-\xi')(1-\eta'),$$
$$N_2^{(e')} = \tfrac{1}{4}(1+\xi')(1-\eta'),$$
$$N_3^{(e')} = \tfrac{1}{4}(1+\xi')(1+\eta'),$$
$$N_4^{(e')} = \tfrac{1}{4}(1-\xi')(1+\eta').$$

To extrapolate w to corner 1, say, we replace its ξ' and η' coordinates, namely $\xi' = \eta' = -3^{1/2}$, into the above formula. Doing that for the four corners we obtain

$$\begin{bmatrix} w_1 \\ w_2 \\ w_3 \\ w_4 \end{bmatrix} = \begin{bmatrix} 1+\frac{1}{2}\sqrt{3} & -\frac{1}{2} & 1-\frac{1}{2}\sqrt{3} & -\frac{1}{2} \\ -\frac{1}{2} & 1+\frac{1}{2}\sqrt{3} & -\frac{1}{2} & 1-\frac{1}{2}\sqrt{3} \\ 1-\frac{1}{2}\sqrt{3} & -\frac{1}{2} & 1+\frac{1}{2}\sqrt{3} & -\frac{1}{2} \\ -\frac{1}{2} & 1-\frac{1}{2}\sqrt{3} & -\frac{1}{2} & 1+\frac{1}{2}\sqrt{3} \end{bmatrix} \begin{bmatrix} w'_1 \\ w'_2 \\ w'_3 \\ w'_4 \end{bmatrix}$$

Note that the sum of the coefficients in each row is one, as it should be. Also, note that each stress component is extrapolated independently.



Gauss elements for higher order quadrilaterals and triangles:
(a) 9-node element with 3 × 3 Gauss rule, (b) 8-node element with 3 × 3 Gauss rule, (c) 6-node element with 3-interior point rule.

1.3.26.2.2 Superconvergent Patch Recovery (SPR) method (ZIENKIEWICZ, 2000)

This is a highly efficient method of stress extrapolation.

1.3.26.2.3 Recovery by Equilibration of Patches (REP) method (ZIENKIEWICZ, 2000)

This is a highly efficient method of stress extrapolation.

1.3.26.3 (Transformation into Global Coordinates and) Averaging of the Elemental Grid Stresses to Yield the Grid Point Stresses

The stress is calculated on an element by element basis (unlike the displacement) so it is possible, and in fact very likely, that the stresses at a grid in one element will not match up with the stress at the same grid in an adjacent element (irrespective of whether the elemental stress is calculated directly at the grids or by extrapolation from the Gauss points). In some cases, where there is a discontinuity in the material properties or the thickness of the elements, it is correct that there is a local stress discontinuity, however in other cases this stress difference will be a measure of the error in the solution. When contour plots of the stresses are required, or when a stress value is required for design purposes it is better to have a single value. The values from all the adjacent *independent elements grid point stresses* at a grid can be averaged to give a single value at the grid, i.e. the *dependent grid point stresses*. This averaging can be weighted based on the element type, geometry etc. if required. Typically the determinant of the Jacobian divided by the total element volume is used as the basis of the weighting function since this will emphasize those elements that have a large amount of material related to the node at the expense of those elements that are rather elongated. However it should be remembered that the stresses, which look smooth and continuous on the plots, are in fact calculated as a series of discontinuous stress patches. As the mesh is refined, the element stress values will tend to converge. Hence, the averaged grid point stress can be used as an error estimator. The averaging of the element grid stresses is best performed in the post-processor. Averaging **SHOULD NOT** be

performed between **dissimilar element types**, between **shells of differing thickness**, **beams of differing cross sections** nor between **elements of different material properties**.

1.3.26.4 Element Stress Recovery Procedures in MSC.NASTRAN

1.3.26.4.1 CQUAD4 Elements

CENTRE method requests element stress recovery at center only. This is the default for **QUAD4** elements and is the optimum sampling point. This is requested with ELSTRESS (CENTER).

BILINEAR stress recovery procedure requests stress recovery at center and at element grids. This is requested with ELSTRESS (CORNER). This method uses the results at the gauss points to create a linear function that is then used to extrapolate the results out to the grid points. In the case of a linear varying moment in a cantilevered engineering beam model, since the QUAD4 element is an almost constant strain curvature element, the gauss results are constant curvature in each element, thus the linear function is almost constant curvature for each element, thus the CORNER strain curvatures are almost constant in an element, hence the reason for the CENTER method being the default for QUAD4. In a cantilevered beam problem, corner stress output using the bilinear method will cause a step function from the cantilever point to the load application point, which is obviously a coarse approximation of the correct linear stress distribution of the engineering model. To conclude, the bilinear corner stress option uses the grid displacements to interpolate strains to gauss points using the element shape functions. Stresses are then calculated at these gauss points. Finally, these stresses are extrapolated to grids using a bilinear stress function.

CUBIC stress recovery requests stress recovery at the grids **directly without calculating at the Gauss points**; In MSC.NASTRAN the CUBIC method works from the grid displacements. It uses the grid displacements and rotations to curve-fit a cubic equation to develop the element strain/curvature distribution. In the case of a linear varying moment in the cantilevered beam engineering model, since the grid point rotations vary across the element, the curve fit gives the correct engineering result of a linearly varying strain curvature across the QUAD4 element, which translates to a linear varying stress. This method is much like the cubic beam stress recovery.

SGAGE stress recovery requests stress recovery at the grids directly without calculating at the Gauss points; In MSC.NASTRAN, the SGAGE method is similar to the CUBIC method in that grid displacements are used, but in-plane strains and curvatures are calculated independently. First strains are calculated in the u and v and diagonal uv directions at each grid point. The state of in-plane strain at the grid point is calculated using rosette strain gauge equations. Grid strain curvatures are done similarly. The resulting grid point strain and curvature states can then be changed to forces and stresses at the grid points.

The BILINEAR option is the default because it is the most stable in all cases, but as is shown in the linearly varying moment case, it can be much less accurate than the CUBIC method. The CUBIC method is very accurate if the mesh if fine enough to give accurate displacements. However, coarse meshes can cause the cubic curve-fit to be more inaccurate than the BILINEAR method. The SGAGE method is similar to the CUBIC method but not as accurate and is quite obsolete.

Stiffness matrix integration and stress calculation need not necessarily use the same Gauss Quadrature Rule, i.e. the stresses may be calculated at the more accurate (superconvergent) lower order Gauss Rule and the stiffness fully integrated. Because we know that the (fully integrated) stress variation of the finite element is not good enough to capture variation in reality, we use reduced integration stress recovery techniques, as recovery is best at the lower order Gauss points. This is what happens with the BILINEAR, SPR and REP recovery methods; not with CUBIC or SGAGE methods which do not extrapolate stresses from the Gauss points, instead calculates stresses from the displacements and rotations at the nodes. Hence, we can say that BILINEAR, SPR or REP are better when the fully integrated stress variation is not good enough, but CUBIC or SGAGE is better when the fully integrated stress variation is good enough to model reality. In any case, the results should converge with refinement of the mesh.

Reiterating, BILINEAR is the superconvergent reduced integrated (although can be fully integrated as well if the finite element variation is deemed accurate enough to model reality) stress recovery (i.e. stress recovery at Gauss points based on, the not so high order polynomial, shape functions defined by the displacement unknowns; then

extrapolation from Gauss points), and CUBIC is the (fully integrated) stress recovery (i.e. stress recovery not at Gauss points but throughout the element based on shape functions defined by the unknowns). Hence, we can say that the reduced integrated BILINEAR is better when the fully integrated stress variation is not good enough, but CUBIC is better when the fully integrated stress variation is good enough to model reality. In any case, the results should converge with refinement of the mesh. The ELSTRESS (CENTER) outputs the σ_x, σ_y and τ_{xy} stresses in the ***element coordinate system (MCID field of CQUAD4)*** at the ***center*** of the element on the ***top and bottom surfaces***.

S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)

ELEMENT ID.	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRESSES (ZERO SHEAR) ANGLE	MAJOR	MINOR	VON MISES
1	-5.000000E-02	-7.523499E+03	-1.793419E+03	-3.373840E+02	-86.6419	-1.773622E+03	-7.543295E+03	6.831404E+03
	5.000000E-02	7.476501E+03	1.786722E+03	3.313992E+02	3.3222	7.495738E+03	1.767485E+03	6.786861E+03
2	-5.000000E-02	-4.515933E+03	-3.544799E+02	-7.011219E+01	-89.0350	-3.532990E+02	-4.517114E+03	4.351235E+03
	5.000000E-02	4.484066E+03	3.550410E+02	6.498148E+01	.9014	4.485089E+03	3.540186E+02	4.318975E+03
3	-5.000000E-02	-1.508649E+03	3.605036E+02	1.940267E+02	84.1357	3.804320E+02	-1.528577E+03	1.750085E+03
	5.000000E-02	1.491351E+03	-3.600570E+02	-2.035071E+02	-6.1993	1.513457E+03	-3.821626E+02	1.736371E+03

Stresses in element coordinate system.

Angle of principal stress in element coordinate system.

Major and minor principal stresses.

Maximum shear stress or Hencky-von Mises stress.

Fiber distance Z1 and Z2 specified on the PSHELL property entry

The ELFORCE (CENTER) outputs the membrane forces per unit length Fx, Fy, and Fxy, bending moments per unit length Mx, My and Mxy and transverse shear forces per unit length Qx and Qy, all in the ***element coordinate system*** at the ***center*** of the element.

F O R C E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)

ELEMENT ID	- MEMBRANE FORCES - FX	FY	FXY	- BENDING MOMENTS - MX	MY	MXY	- TRANSVERSE SHEAR FORCES - QX	QY
1	-2.349886E+00	-3.348329E-01	-2.992391E-01	-1.250000E+01	-2.983451E+00	-5.573193E-01	-5.000000E-01	2.352497E-02
2	-1.593343E+00	2.805547E-02	-2.565355E-01	-7.500000E+00	-5.912674E-01	-1.125781E-01	-5.000000E-01	3.125528E-02
3	-8.648885E-01	2.233189E-02	-4.740243E-01	-2.500000E+00	6.004672E-01	3.312782E-01	-5.000000E-01	1.284359E-02
4	1.349886E+00	1.656714E-01	-4.015217E-01	-1.250000E+01	-2.983451E+00	5.573193E-01	-5.000000E-01	-2.352497E-02
5	5.933433E-01	-8.414105E-02	-4.869289E-01	-7.500000E+00	-5.912674E-01	1.125781E-01	-5.000000E-01	-3.125528E-02
6	-1.351115E-01	-4.106952E-03	-5.259757E-01	-2.500000E+00	6.004672E-01	-3.312782E-01	-5.000000E-01	-1.284359E-02

In-plane element forces in element coordinate system (force/length).

Element internal moments in element coordinate system (moment/length).

Traverse shear forces (force/length).

The ELSTRESS (CORNER) outputs the σ_x, σ_y and τ_{xy} stresses in the ***element coordinate system*** at the ***center and 4 corners*** of the element on the ***top and bottom surfaces***.

S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4) OPTION = BILIN

ELEMENT ID	GRID-ID	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRESSES (ZERO SHEAR) ANGLE	MAJOR	MINOR	VON MISES
1	CEN/4	-5.000000E-02	-7.523499E+03	-1.793269E+03	-3.396249E+02	-86.6199	-1.773210E+03	-7.543558E+03	6.831780E+03
		5.000000E-02	7.476501E+03	1.786872E+03	3.291584E+02	3.3000	7.495480E+03	1.767893E+03	6.786491E+03
	1	-5.000000E-02	-8.650413E+03	-2.595124E+03	-3.788401E+02	-86.4339	-2.571515E+03	-8.674022E+03	7.716604E+03
		5.000000E-02	8.561436E+03	2.568431E+03	3.851403E+02	3.6620	8.586085E+03	2.543781E+03	7.638754E+03
	2	-5.000000E-02	-5.493704E+03	-5.921041E+02	-1.022627E+03	-78.6755	-3.873087E+02	-5.698500E+03	5.515055E+03
		5.000000E-02	5.409805E+03	5.823387E+02	1.001319E+03	11.2654	5.609259E+03	3.828847E+02	5.427954E+03
	6	-5.000000E-02	-6.247504E+03	-1.075137E+03	-3.180060E+02	-86.4949	-1.055658E+03	-6.266982E+03	5.811513E+03
		5.000000E-02	6.242486E+03	1.089036E+03	2.907728E+02	3.2192	6.258840E+03	1.072681E+03	5.797412E+03
	5	-5.000000E-02	-9.702373E+03	-2.910712E+03	3.257812E+02	87.2600	-2.895121E+03	-9.717964E+03	8.642099E+03
		5.000000E-02	9.692277E+03	2.907683E+03	-3.254057E+02	-2.7397	9.707849E+03	2.892112E+03	8.633103E+03

Denotes CEN/4 for the center stress or the grid point ID for a corner stress.

The ELFORCE (CORNER) outputs the membrane forces per unit length Fx, Fy, and Fxy, bending moments per unit length Mx, My and Mxy and transverse shear forces per unit length Qx and Qy, all in the ***element coordinate system*** at the ***center and 4 corners*** of the element.

F O R C E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4) OPTION = BILIN

ELEMENT ID	GRID-ID	- MEMBRANE FORCES - FX	FY	FXY	- BENDING MOMENTS - MX	MY	MXY	- TRANSVERSE SHEAR FORCES - QX	QY
1	CEN/4	-2.349886E+00	-3.198563E-01	-5.233257E-01	-1.250000E+01	-2.983451E+00	-5.573193E-01	-5.000000E-01	2.352497E-02
	1	-4.448889E+00	-1.334667E+00	3.150085E-01	-1.434321E+01	-4.302963E+00	-6.366503E-01	-5.000000E-01	2.352497E-02
	2	-4.194971E+00	-4.882719E-01	-1.065422E+00	-9.086258E+00	-9.787024E-01	-1.686622E+00	-5.000000E-01	2.352497E-02
	6	-2.508836E-01	6.949542E-01	-1.361660E+00	-1.040833E+01	-1.803477E+00	-5.073156E-01	-5.000000E-01	2.352497E-02
	5	-5.048020E-01	-1.514406E-01	1.877032E-02	-1.616221E+01	-4.848663E+00	5.426558E-01	-5.000000E-01	2.352497E-02
2	CEN/4	-1.593343E+00	6.351992E-02	-4.918268E-01	-7.500000E+00	-5.912674E-01	-1.125781E-01	-5.000000E-01	3.125528E-02
	2	-2.714164E+00	-4.402977E-02	-1.611118E-01	-1.134483E+01	-1.656274E+00	-1.138151E+00	-5.000000E-01	3.125528E-02
	3	-2.864952E+00	-5.466592E-01	-9.984620E-01	-6.194139E+00	-3.528372E-03	7.047635E-01	-5.000000E-01	3.125528E-02
	7	-4.725232E-01	1.710696E-01	-8.225417E-01	-3.723025E+00	5.971142E-01	8.980479E-01	-5.000000E-01	3.125528E-02
	6	-3.217343E-01	6.736990E-01	1.480843E-02	-8.738006E+00	-1.302381E+00	-9.448670E-01	-5.000000E-01	3.125528E-02

Denotes CEN/4 for the center stress or the grid point ID for a corner stress.

Note that the options CENTER or CORNER are not applicable to elements other than QUAD4.

1.3.26.4.2 CQUAD8

The ELSTRESS outputs the σ_x, σ_y and τ_{xy} stresses in the ***element coordinate system*** at the ***center and 4 corners*** of the element on the ***top and bottom surfaces***.

S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 8)

ELEMENT ID	GRID-ID	FIBRE DISTANCE	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRESSES (ZERO SHEAR) ANGLE	MAJOR	MINOR	VON MISES
1	CEN/4	-5.000000E-02	-7.521979E+03	-2.218078E+03	-2.661170E+01	-89.7125	-2.217945E+03	-7.522112E+03	6.694611E+03
		5.000000E-02	7.478021E+03	2.211569E+03	1.614634E+01	.1757	7.478070E+03	2.211519E+03	6.653905E+03
	1	-5.000000E-02	-8.942510E+03	-2.682753E+03	-4.751408E+01	-89.5651	-2.682393E+03	-8.942870E+03	7.948703E+03
		5.000000E-02	8.860332E+03	2.658100E+03	2.187377E+01	.2021	8.860409E+03	2.658022E+03	7.875326E+03
	2	-5.000000E-02	-6.126543E+03	-1.731518E+03	-3.533652E+01	-89.5394	-1.731234E+03	-6.126827E+03	5.470668E+03
		5.000000E-02	6.048768E+03	1.721540E+03	3.490948E+01	.4622	6.049049E+03	1.721259E+03	5.398310E+03
	6	-5.000000E-02	-5.915213E+03	-1.726948E+03	-5.709310E+00	-89.9219	-1.726940E+03	-5.915221E+03	5.268484E+03
		5.000000E-02	5.909476E+03	1.738582E+03	1.041892E+01	.1431	5.909501E+03	1.738556E+03	5.260303E+03
	5	-5.000000E-02	-9.103648E+03	-2.731095E+03	-1.788687E+01	-89.8392	-2.731045E+03	-9.103699E+03	8.091559E+03
		5.000000E-02	9.093509E+03	2.728053E+03	-2.616797E+00	-.0236	9.093510E+03	2.728052E+03	8.082488E+03

Denotes CEN/4 for the center stress or a grid point ID for a corner stress.

The ELFORCE outputs the membrane forces per unit length Fx, Fy, and Fxy, bending moments per unit length Mx, My and Mxy and transverse shear forces per unit length Qx and Qy, all in the ***element coordinate system*** at the ***center and 4 corners*** of the element.

F O R C E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 8)

ELEMENT ID	GRID-ID	- MEMBRANE FORCES - FX	FY	FXY	- BENDING MOMENTS - MX	MY	MXY	- TRANSVERSE SHEAR FORCES - QX	QY
1	CEN/4	-2.197888E+00	-3.254840E-01	-5.232677E-01	-1.250000E+01	-3.691373E+00	-3.563170E-02	-5.205615E+00	8.254652E-02
	1	-4.108907E+00	-1.232672E+00	-1.282016E+00	-1.483570E+01	-4.450711E+00	-5.782321E-02	-5.205615E+00	8.254652E-02
	2	-3.888765E+00	-4.988650E-01	-2.135196E-02	-1.014609E+01	-2.877548E+00	-5.853834E-02	-5.205615E+00	8.254652E-02
	6	-2.868679E-01	5.817043E-01	2.354806E-01	-9.853907E+00	-2.887942E+00	-1.344019E-02	-5.205615E+00	8.254652E-02
	5	-5.070101E-01	-1.521030E-01	-1.025183E+00	-1.516430E+01	-4.549290E+00	-1.272506E-02	-5.205615E+00	8.254652E-02
2	CEN/4	-1.499658E+00	4.452617E-02	-4.931346E-01	-7.500000E+00	-2.112700E+00	-1.628812E-02	-1.779801E+00	2.273268E-02
	2	-2.536411E+00	-9.315875E-02	-8.093295E-01	-9.912405E+00	-2.807442E+00	-2.389918E-02	-1.779801E+00	2.273268E-02
	3	-2.650702E+00	-4.741282E-01	-4.360040E-02	-5.061935E+00	-1.334132E+00	-2.409612E-02	-1.779801E+00	2.273268E-02
	7	-4.629044E-01	1.822111E-01	-1.769397E-01	-4.938065E+00	-1.351179E+00	-8.677053E-03	-1.779801E+00	2.273268E-02
	6	-3.486136E-01	5.631805E-01	-9.426688E-01	-1.008759E+01	-2.958048E+00	-8.480112E-03	-1.779801E+00	2.273268E-02

Denotes CEN/4 for the center forces or the grid point ID for a corner force.

1.3.26.4.3 CTRIA3

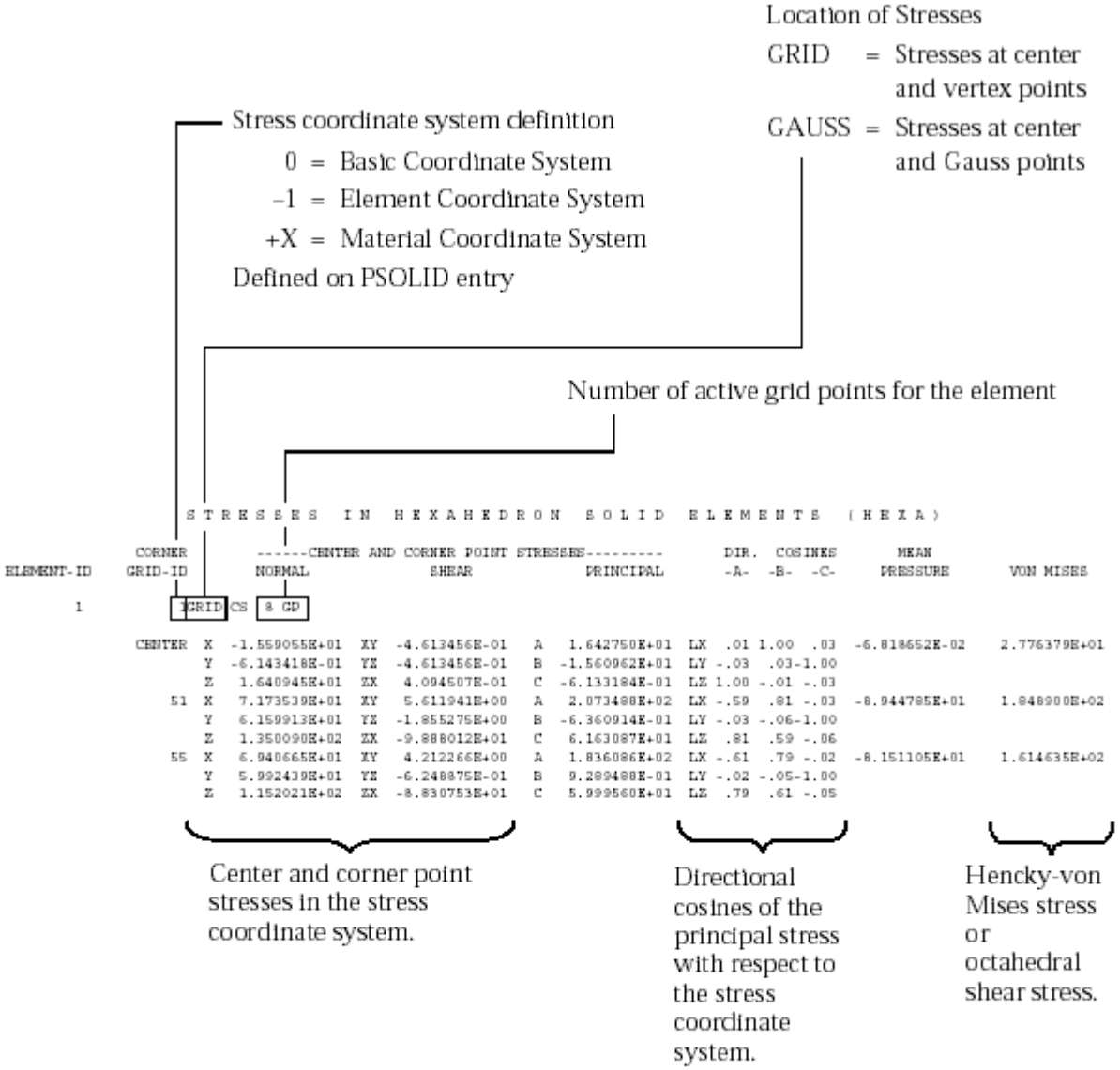
The ELSTRESS outputs the σ_x, σ_y and τ_{xy} stresses in the ***element coordinate system*** at the ***center*** of the element on the ***top and bottom surfaces***. The ELFORCE outputs the membrane forces per unit length Fx, Fy, and Fxy, bending moments per unit length Mx, My and Mxy and transverse shear forces per unit length Qx and Qy, all in the ***element coordinate system*** at the ***center*** of the element.

1.3.26.4.4 CTRIA6

The ELSTRESS outputs the σ_x, σ_y and τ_{xy} stresses in the ***element coordinate system*** at the ***center and 3 corners*** of the element on the ***top and bottom surfaces***. The ELFORCE outputs the membrane forces per unit length Fx, Fy, and Fxy, bending moments per unit length Mx, My and Mxy and transverse shear forces per unit length Qx and Qy, all in the ***element coordinate system*** at the ***center and 3 corners*** of the element.

1.3.26.4.5 CHEXA4, CHEXA20, CPENTA6, CPENTA15, CTETRA4, CTETRA10

The ELSTRESS outputs the σ_x, σ_y ,σ_z, τ_{xy} ,τ_{zx} and τ_{yz} stresses in the ***element, basic (default) or material (depending on CORDM field of the PSOLID entry) coordinate system*** at the ***center and either the corners (default) or the Gauss points (depending on PSOLID entry)*** of the element.



Note: This output is typical for the CHEXA, CPENTA, and CTETRA elements.

1.3.26.5 Grid Point Stresses in MSC.NASTRAN

Although it is usual to perform the grip point stress calculation in the post-processor, it is instructive to know that MSC.NASTRAN also has the feature of calculating grid point stresses. Moreover, the methods used within the post-processor are usually similar. The grid point stress option calculates the stresses at the grid points from the adjoining plate and solid elements in the global or a user defined coordinate system. Each stress component (e.g. σ_x) is treated independently of other stress components in the calculation of average grid point stress components. The stress invariants at grid points (i.e., principal stresses, the Hencky-von Mises stress, and the mean pressure) are evaluated from the average values of the stress components at the grid points.

Most element have vertex element stress (i.e. *element grid point stress*) values. These include CQUADR, CQUAD4 (with corner option), CQUAD8, CTRIAR, and CTRIA6 shell elements as well as the CHEXA, CPENTA, and CTETRA solid elements. But not CQUAD4 (without corner option) and CTRIA3.

Two methods are used to calculate grid point stress components for plate and shell elements namely, **topological** and **geometric**, with topological as the default method.

Topological	Geometric
Averages stresses at grids with element vertex stresses, ignores CQUAD4 (without corner option) and CTRIA3.	Averages stresses at grids with element vertex stresses, ignores CQUAD4 (without corner option) and CTRIA3.
Grids with only CQUAD4 (without corner option) and/or CTRIA3, average of center stress used. Hence no element size consideration.	Grids with only CQUAD4 (without corner option) and/or CTRIA3, average stress obtained from least square approximation. Hence size of element considered.
Recognizes stresses at interior, corner, or edge grid points to be continuous between directly connected elements. Does not recognize exception grid points. User defined separate surface sets adjacent to exception grids required.	Recognizes stresses at interior, corner, or edge grid points to be continuous between directly connected elements. Recognizes stresses at exception grid points to be discontinuous (large slope between adjacent elements cause stress discontinuities) between connected elements and, a different grid point stress may be output for each of the connected elements.

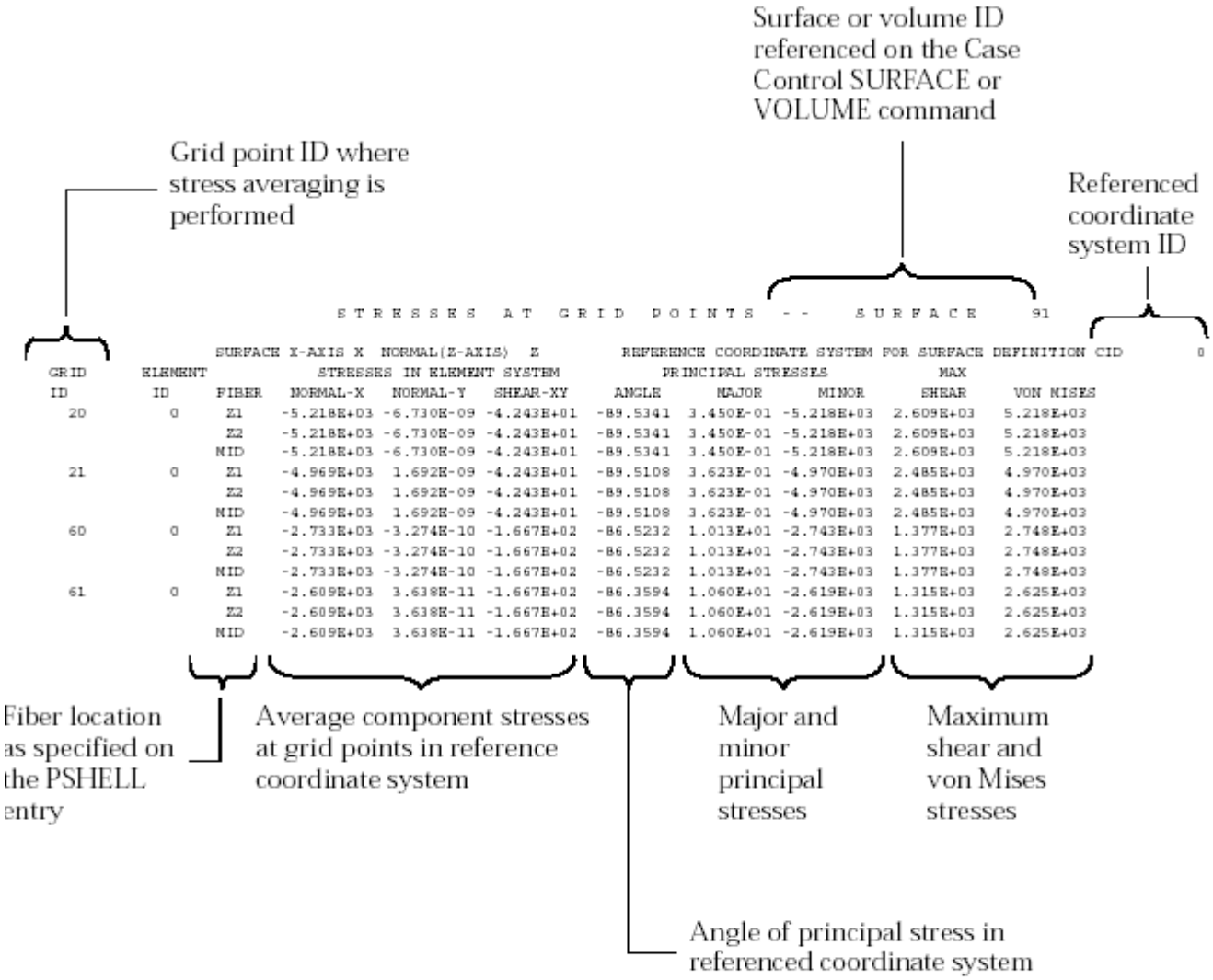
If the results are substantially different when using the two different methods, it may be an indication that the FE model is **not fine enough**. In general, if the FE model is fine enough to capture the stress gradients, the two methods should yield similar results. The NASTRAN request interface is as follows.

```
$ CASE CONTROL
ELSTRESS (PLOT) = ALL $ Required
$
SET 10 = < SURFACE IDs > say < 100, 200 >
GPSTRESS = 10 $ Printed Output of Grid Point Stresses
STRFIELD = 10 $ Graphical Output of Grid Point Stresses
$
OUTPUT(POST)
 SET 1 = < Element IDs >
 SET 2 = < Element IDs >
 SURFACE 100 SET 1 NORMAL Z
 SURFACE 200 SET 2 NORMAL Z
```

For shells, Z-normals (defining top and bottom surfaces) are required. Specify VOLUME for solids.

The element positive normal is in the direction of the z-axis of the basic coordinate system. By default, the grid point stresses are transformed to the basic coordinate system. The grid point stresses are called out by the GPSTRESS command, which points to set 10, which, in turn, points to surfaces 100 and 200.

The GPSTRESS output is presented as follows.



STRESSES AT GRID POINTS -- SURFACE 91

SURFACE X-AXIS X NORMAL(Z-AXIS) Z REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CID 0

GRID ID	ELEMENT ID	FIBER	STRESSES IN ELEMENT SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRESSES ANGLE	MAJOR	MINOR	MAX SHEAR	VON MISES
20	0	Z1	-5.218E+03	-6.730E-09	-4.243E+01	-89.5341	3.450E-01	-5.218E+03	2.609E+03	5.218E+03
		Z2	-5.218E+03	-6.730E-09	-4.243E+01	-89.5341	3.450E-01	-5.218E+03	2.609E+03	5.218E+03
		NID	-5.218E+03	-6.730E-09	-4.243E+01	-89.5341	3.450E-01	-5.218E+03	2.609E+03	5.218E+03
21	0	Z1	-4.969E+03	1.692E-09	-4.243E+01	-89.5108	3.623E-01	-4.970E+03	2.485E+03	4.970E+03
		Z2	-4.969E+03	1.692E-09	-4.243E+01	-89.5108	3.623E-01	-4.970E+03	2.485E+03	4.970E+03
		NID	-4.969E+03	1.692E-09	-4.243E+01	-89.5108	3.623E-01	-4.970E+03	2.485E+03	4.970E+03
60	0	Z1	-2.733E+03	-3.274E-10	-1.667E+02	-86.5232	1.013E+01	-2.743E+03	1.377E+03	2.748E+03
		Z2	-2.733E+03	-3.274E-10	-1.667E+02	-86.5232	1.013E+01	-2.743E+03	1.377E+03	2.748E+03
		NID	-2.733E+03	-3.274E-10	-1.667E+02	-86.5232	1.013E+01	-2.743E+03	1.377E+03	2.748E+03
61	0	Z1	-2.609E+03	3.638E-11	-1.667E+02	-86.3594	1.060E+01	-2.619E+03	1.315E+03	2.625E+03
		Z2	-2.609E+03	3.638E-11	-1.667E+02	-86.3594	1.060E+01	-2.619E+03	1.315E+03	2.625E+03
		NID	-2.609E+03	3.638E-11	-1.667E+02	-86.3594	1.060E+01	-2.619E+03	1.315E+03	2.625E+03

Note: Stress surface output is available for all plate and shell elements (CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR).

Stress volume output is available for all solid elements (CHEXA, CPENTA, and CTETRA).

1.3.27 Full Integration Quadrature, Reduced Integration Quadrature and Optimal Gauss Sampling Points

It is usually advantages to use **A REDUCED INTEGRATION TECHNIQUE (i.e. the minimum integration requirements that PRESERVES THE RATE OF CONVERGENCE which would result if exact integration were used)** as, for very good reasons, **A CANCELLATION OF ERRORS due to discretization and due to inexact integration** can occur [7].

As far as the optimal points for gradient or stress recovery is concerned, it was observed experimentally by Barlow and proved by Herrmann, that the optimal (not exact because of isoparametric distortion) superconvergent points are the Gauss-Legendre points. This almost always corresponds to the locations of the Gauss-Legendre points for reduced integration (i.e. integration whilst still maintaining the rate of convergence if fully integrated).

Element	Displacement Variation	Fully Integrated Stress Variation	Fully Integrated Gauss Quadrature	Optimum Superconvergent Sampling Points (Reduced Integration Gauss Quadrature)	Reduced Integrated Stress Variation
Two-noded C_1 Cubic Beam	y = constant + x + x^2 + x^3 Complete 3rd order	M_y, M_z linear in x; V_y, V_z constant in x; A_x constant in x; T_x constant in x;	2	2	M_y, M_z linear in x; V_y, V_z constant in x; A_x constant in x; T_x constant in x;
In-plane C_0 Linear TRIA3 (CST)	u = constant + x + y v = constant + x + y Complete 1st order	σ_x, σ_y and τ_{xy} constant is x and y		Both 1 x 1 	σ_x, σ_y and τ_{xy} constant in x and y
In-plane C_0 Parabolic TRIA6 (LST)	u = constant + x + y + x^2 + xy + y^2 v = constant + x + y + x^2 + xy + y^2 Complete 2nd order	σ_x, σ_y and τ_{xy} linear is x and y	Rule 3 or –3 	Optimal Sampling Reduced Integration 	σ_x, σ_y and τ_{xy} linear in x and y
In-plane C_0 Linear QUAD4	u = constant + x + y + xy v = constant + x + y + xy Complete 1st order; Incomplete 2nd order	σ_x constant in x, linear in y; σ_y constant in y, linear in x; τ_{xy} linear is x and y	2 x 2 	Both 1 x 1 	σ_x, σ_y and τ_{xy} constant in x and y
In-plane C_0 Parabolic QUAD8 (Serendipity)	u = constant + x + y + xy + x^2 + xy + y^2 + x^3y v = constant + x + y + xy + x^2 + xy + y^2 + xy^3 Complete 2nd order; Incomplete 3rd order	σ_x linear in x, parabolic in y; σ_y linear in y, parabolic in x; τ_{xy} parabolic in x, y	3 x 3 	Both 2 x 2 	σ_x, σ_y and τ_{xy} linear in x and y
Out-of-plane C_1 Kirchhoff QUAD4	w = constant + x + y + x^2 + xy + y^2 + x^3 + y^3 + x^2y + xy^2 + x^3y + xy^3 Complete 3rd order	σ_x linear in x σ_y linear in y			
Out-of-plane C_0 Mindlin QUAD4			2 x 2 bending 2 x 2 shear	1 x 1 or 2 x 2 bending 1 x 1 shear	σ_x, σ_y constant
Out-of-plane C_0 Mindlin QUAD8			3 x 3 bending 3 x 3 shear	2 x 2 or 3 x 3 bending 2 x 2 shear	σ_x, σ_y linear
Linear TETRA4	u = constant + x + y + z v = constant + x + y + z w = constant + x + y + z Complete 1st order	σ_x, σ_y, σ_z, τ_{xy}, τ_{yz}, τ_{zx} constant		1 x 1 x 1 	

[7] ZIENKIEWICZ, O.C. & TAYLOR, R.L. *The Finite Element Method. The Basis. Volume 1*. 5th Edition. Butterworth-Heinemann, Oxford, 2000.

Parabolic TETRA10					
Linear HEXA8	u = constant + x + y + z + xy + yz + zx + xyz v = likewise w = likewise Complete 1st order; Incomplete 2nd order		2 x 2 x 2	1 x 1 x 1	
Parabolic HEXA20	u = constant + x + y + z + x^2 + y^2 + z^2 + xy + yz + zx + x^2y + xy^2 + y^2z + yz^2 + z^2x + zx^2 + xyz + x^2yz + xy^2z + xyz^2 v = likewise w = likewise Complete 2nd order; Incomplete 3rd order		3 x 3 x 3	2 x 2 x 2	

1.3.28 Stress Interpretation

Stress recovery concepts, namely

I. stress recovery at the element Gauss points, then
II. subsequent stress extrapolation to the element grids, and
III. eventually, the (transformation into global coordinates and) averaging of the elemental stresses to yield the grid point stresses

have been covered. Here we shall focus on the engineering interpretation of these stresses.

A solid element outputs (the appropriate variations of) 3 normal stresses and 3 shear stresses σ_{xx}, σ_{yy}, σ_{zz}, τ_{xy}, τ_{yz}, τ_{zx} in its element coordinate system. A shell element outputs (the appropriate variations of) 2 normal stresses and 1 shear stress σ_{xx}, σ_{yy}, τ_{xy} in its element coordinate system. A beam element outputs (the appropriate variations of) 2 bending moments M_{yy} and M_{zz}, 2 shear forces F_y and F_z, 1 torsional moment M_{xx} and 1 axial force F_x. Stresses from solids and shells can be cumulated into forces if the design criteria are specified (as is usual for composite material such as reinforced concrete) in terms of moments, shear and axial forces. Conversely, forces from beam elements can be used to derive stresses based on simple bending theory.

Element stress recovery is (unless otherwise specified) done in the element coordinate system. To define a beam, the two end nodes define the x-direction of the element from node 1 to node 2. The third node defines either its y- or z-axis. To define shells, the direction of the node number definitions defines the element x- and y-axes. The z-axis is then automatically defined using the Right Hand Rule usually. Likewise, for solids, the numbering of the nodes will define the x-, y- and z-axes. A preprocessor should be used to plot the element coordinate systems for all the elements (ensuring that it is consistent with the element coordinate definition of the particular code).

For beams, the elements **X-, Y- and Z- axes *must be* orientated in the same manner**. Stress recovery is performed in the **beam local element system**. No transformation into a global system is performed, hence the requirement for the elements to be orientation consistently for the bending moment and shear force diagrams about and in the two directions to look consistent between elements.

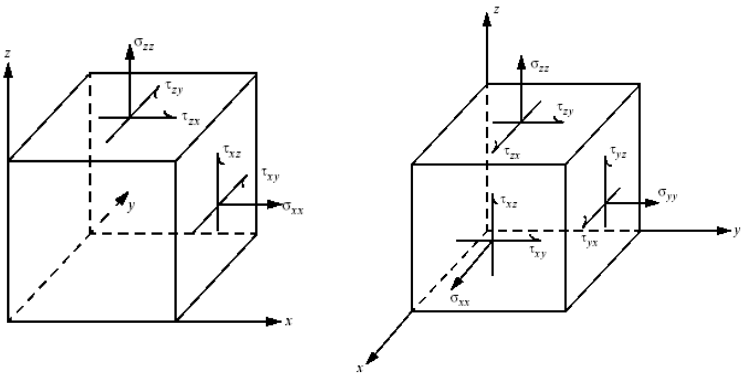
For shells, **the element Z-normals *must be* consistent**. This is because the top and bottom surface stresses need to be viewed consistently. Stress recovery is performed in the **element coordinate system by default (but defined by MCID entry on element connection card; 0 for basic projected, > 0 for user projected, < blank > for element coordinate system, > 0.0 and < 360.0 for angled from side n1-n2 of element)**. The element results are then usually transformed into the global coordinate system. Of course, unless the results are transformed into the global coordinate system, the definition of the X- and Y- element coordinate system between elements of the same type *must also* be consistent. Alternatively, a random element X- and Y- orientation (but consistent Z-normals of course) can be used between elements of the same type if after stress recovery, the output is transformed (by the post-processor) into the global coordinate system.

For solid elements, the stress recovery is performed in the **basic coordinate system by default (but defined on CORDM field of PSOLID; 0 for basic, > 0 for user-defined, -1 for element coordinate system)**. Hence, if the stress recovery is presented in the basic coordinate system, a random X-, Y-, Z- element orientation can be used.

To conclude, the element X-, Y- and Z- orientation for beams must be consistent as stress recovery is presented in the element coordinate system, the Z-normal for shells must be consistent but the element X- and Y- orientation can be random as long as the stress recovery is presented in the global coordinate system, and for solids a random X-, Y- and Z- element orientation is fine as long as the stress recovery is presented in the global coordinate system.

Note that although the element stress tensors σ_{xx}, σ_{yy}, σ_{zz}, τ_{xy}, τ_{yz}, τ_{zx} must be viewed in a consistent coordinate system, derived quantities are independent of the element orientation. These invariant quantities included maximum and minimum principal stresses, maximum shear stress and the von Mises stress.

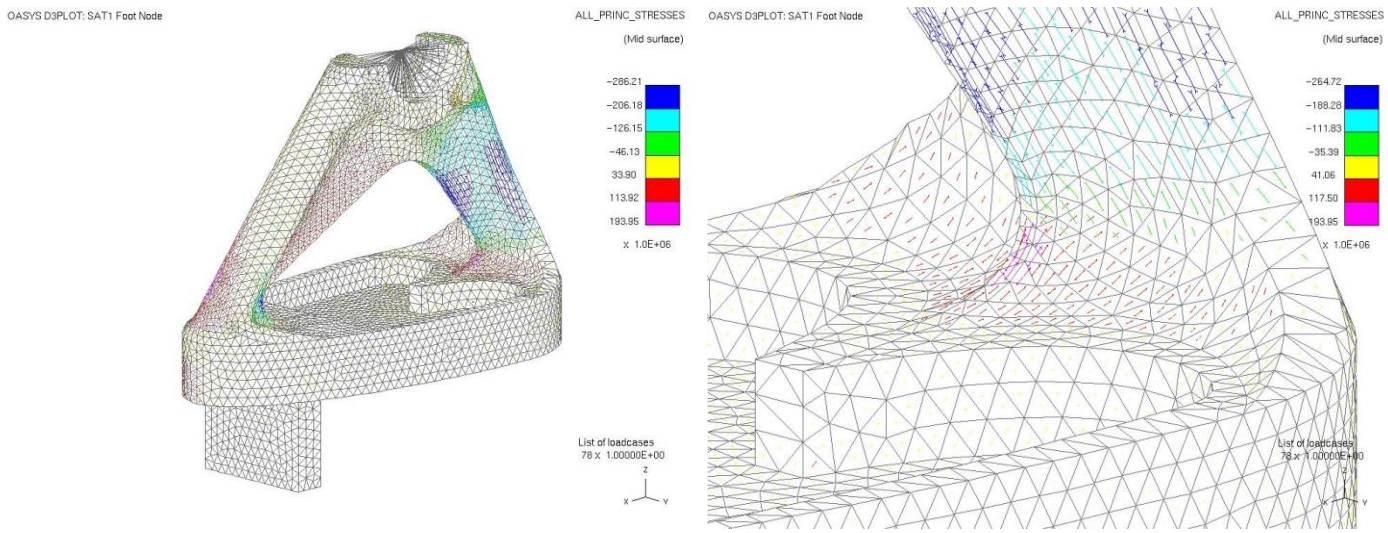
1.3.28.1 State of Stress i.e. Global Coordinate Stress Tensors σ_{xx}, σ_{yy}, σ_{zz}, τ_{xy}, τ_{yz}, τ_{zx}



Normal stresses are normal to the plane and shear stresses act on the plane. The first subscript in the stress notation denotes the direction of the normal to the plane on which the stress acts and the second subscript indicates the direction of the stress itself.

In shells and solids, stresses in the element coordinate system are usually quite meaningless. The state of stress (three normal and three shear stresses in general) at a point with reference to the **global** Cartesian system is also quite insignificant in its original form. However, the various derived stress quantities, which can be obtained, are very illustrative. These are presented as follows.

(i) The **load path** of **flow of stresses** through the structure can be ascertained very intuitively from the **principal** normal stresses σ_1, σ_2 and σ_3. When an element is orientated such that all its shear stress components are zero, the normal stresses on its 3 planes are called the principal stresses σ_1, σ_2 and σ_3 from most positive to most negative. When one of the principal stresses are zero, the stress state is biaxial (or plane stress). When two of the principal stresses are zero, the stress state is then uniaxial. Let us assume tensile as positive. σ_2 and σ_3 that is close to zero indicates a state of uniaxial tensile stress if σ_1 positive. σ_2 and σ_1 that is close to zero indicates a state of uniaxial compressive stress if σ_3 negative. Note that the principal stress orientation is a function of loading, not geometry alone. Hence, a plot of principal stresses is dependent upon **a particular load case**. The direction of flow of stresses (if tensile) is a good indication of the optimum direction of reinforcement to be provided in reinforced concrete.



Flows of Stresses i.e. Principal Stresses on a Structural Connection Casting

(ii) The state of overall stress, i.e. compressive or tensile can be ascertained from the average stress.

$$\sigma_{av} = \tfrac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$$

(iii) The susceptibility of the structure to a specified failure mode can be assessed.

1.3.28.2 Failure Criteria

The applicable failure criterion for a particular material depends on whether the material is ductile or brittle. Ductile materials yield and then undergo significant plastic deformation before ultimate fracture. Brittle materials do not yield and undergo plastic deformation, but fractures when the ultimate strength is attained. **Carbon steel** and **aluminium** are generally **ductile** materials whilst **concrete**, **masonry, quenched tempered alloy steel, hardened tool steel and gray cast iron** are brittle materials. Of course, certain composite materials such as **reinforced concrete** and **reinforced masonry** can be deemed to be ductile materials.

However, some materials become more brittle as the **strain-rate** increases and **temperature** decreases. Some metals such as **copper** and **aluminium** have a crystalline structure that enables them to **resist fast fracture** under all loading conditions and at all temperatures. This is not the case with **ferrous** alloys, particularly structural **steel** which is brittle below the **transition temperature**.

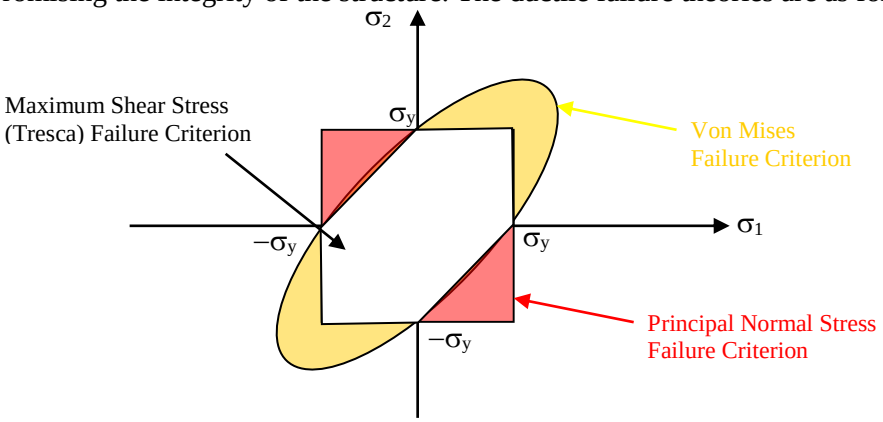
Ductile materials usually have approximately the same ultimate tensile and compressive strengths. Brittle materials on the other hand are usually stronger in compression than in tension. For instance, the ultimate compressive strength of gray cast iron can be 3 to 4 times its ultimate tensile strength. There are exceptions, for instance the brittle hardened tool steel which has similar ultimate strengths in both tension and compression.

If the **percentage elongation** in a tensile test is **less than 5%** at ultimate strength or if there is no published yield strength, then assume brittle behaviour.

It will be shown that ductile materials tend to yield **along 45° shear slip planes** at the same pure normal stress, whether in **tension or compression**, i.e. if a ductile specimen were to be loaded in tension to fracture, the fractures will all be at an angle of approximately 45° from the loading axis. On the other hand, a tensile specimen of brittle material does tend to fail in tension (i.e. fracture on a plane orthogonal to the maximum principal tensile stress) and in **shear at approximately 45° angles** when loaded to fracture in compression.

1.3.28.2.1 Ductile Yielding (Plastic Flow) Failure Criteria

A yielding failure occurs when the material yields such that it compromises the performance of the structure in its load carrying capacity. Localized yielding around stress concentrations may be acceptable as a ductile material is capable of redistributing the stresses sufficiently for yielding to stop at a distance from the concentration whilst not compromising the integrity of the structure. The ductile failure theories are as follows.



Tresca is the safest criterion, but the best match to experimental data is the von Mises criterion. Any of these failure criteria can be employed on a set of loadcases and enveloped.

Principal Normal Stresses σ_1, σ_2 and σ_3 Failure Criterion

The principal normal stresses are the maximum, minimum and intermediate normal stresses that occur when the element is rotated onto its principal plane. The shear stresses are zero on these principal planes. Visualize the 3-dimensional Mohr Circle of stresses (3 circles). The principal stresses in two dimensions are presented, as is that in three dimensions as a solution of a 3rd order equation.

$$\sigma_{1,2} = \frac{\sigma_x + \tau_{xy}}{2} \pm \sqrt{\left(\frac{\sigma_x - \sigma_y}{2}\right)^2 + \tau_{xy}^{\ 2}}$$

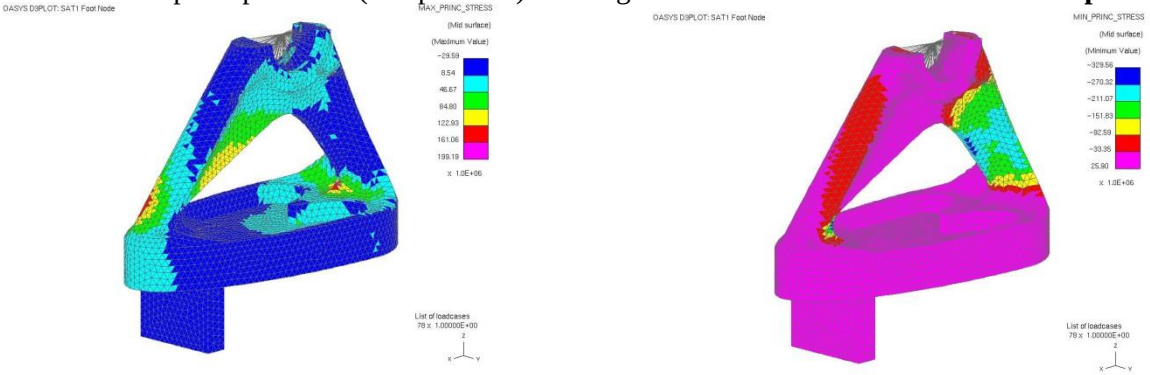
$$\sigma^3 + I_1\sigma^2 + I_2\sigma - I_3 = 0$$

where $I_1 = \sigma_x + \sigma_y + \sigma_z$

$$I_2 = \sigma_x\sigma_y + \sigma_y\sigma_z + \sigma_z\sigma_x - \tau_{xy}^{\ 2} - \tau_{yz}^{\ 2} - \tau_{zx}^{\ 2}$$

$$I_3 = \sigma_x\sigma_y\sigma_z + 2\tau_{xy}\tau_{yz}\tau_{zx} - \sigma_x\tau_{yz}^{\ 2} - \sigma_y\tau_{zx}^{\ 2} - \sigma_z\tau_{xy}^{\ 2}$$

The maximum normal stress failure criterion states that failure occurs whenever σ_1 or σ_3 equals the failure strength of the material in tension or compression, respectively. For ductile materials, failure by yielding occurs when the yield strength is attained whilst failure by fracture occurs when the ultimate strength is reached. Ductile materials have approximately the same ultimate tensile and compressive strengths. This theory however does not account for the entire complex triaxial stress state and hence assumes that yielding (flowing) occurs either along the direction of the maximum principal stress or the minimum principal stress. A plot of the maximum principal stress (tension) and the minimum principal stress (compression) can be generated for **all load cases and enveloped**.



Maximum and Minimum Principal Stresses on a Structural Connection Casting

Von Mises-Hencky (Distortion Energy) Stress $\sigma_{von\ mises}$ Failure Criterion

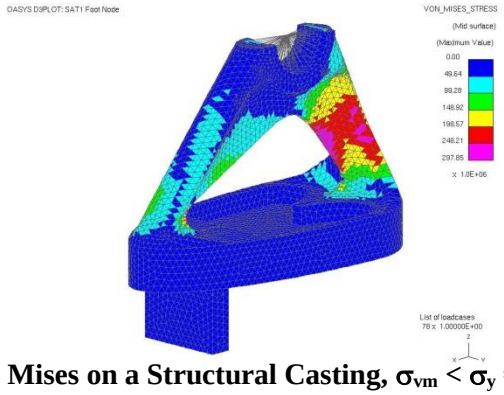
The von Mises (or effective stress) stress is a measure of the distortional stresses. It gives a good indication of how close a material is to yielding or experiencing plastic flow. It is based upon strain energy hypothesis and represents the entire complex triaxial stress state. This yield criterion is suitable for pressure independent material models i.e. materials that will not flow (fail) under the conditions of hydrostatic stress. With this material model the yield surface is independent of pressure (average stress). This is typical of materials such as steel. The material yields when the von Mises stress reaches the one-dimensional yield strength, $\sigma_{vm} = \sigma_y$

$$\text{von Mises } \sigma_{vm} = \frac{1}{\sqrt{2}}[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2))]^{1/2}$$

or, alternatively in terms of principal stresses

$$Y = \frac{1}{\sqrt{2}}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right]^{\frac{1}{2}}$$

The von Mises model provides the best match to experimental data of failure limits. A plot of the von Mises state of stress can be generated for **all load cases and enveloped**.



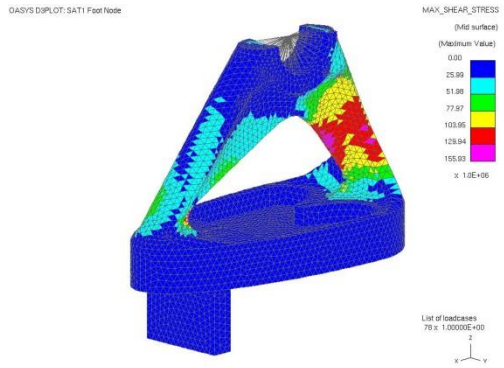
Von Mises on a Structural Casting, $\sigma_{vm} < \sigma_y = 350$MPa OK

Maximum Shear Stress (Tresca) Failure Criterion

The Tresca yield criterion is based on the maximum shear stress and is especially useful for predicting the yielding of annealed ductile materials. The Tresca criterion specifies that if any stress component exceeds the specified yield strength, the material will flow (undergo plastic deformation). It is more conservative than the principal normal stress and the von Mises stress failure criteria. The Tresca criterion states that failure by yielding occurs on a **shear slip plane i.e. at 45° to the principal planes** when the maximum shear stress equals ½ the yield strength.

$\tau_{13} = (\sigma_1 - \sigma_3)/2 = \sigma_y/2$

The theory is suggested by the fact that yielding is related to shear slip (at the plane of maximum shear stress which is 45° to the planes of the maximum and minimum principal stresses) at the atomic level of the material. A plot of the maximum shear stress τ_{13} of stress can be generated for **all load cases and enveloped**.



Tresca Failure Criterion: Max Shear Stress, $\tau_{13} = (\sigma_1 - \sigma_3)/2 < \sigma_y/2 = 175$MPa OK

1.3.28.2.2 Brittle Fracture (Frictional) Failure Criteria

Brittle fracture occurs when the material cracks such that it becomes detrimental to the structural performance in its load carrying capacity. Fracture occurs when cracks appear without significant permanent (plastic) deformation.

Principal Normal Stresses σ_1, σ_2 and σ_3 Failure Criterion

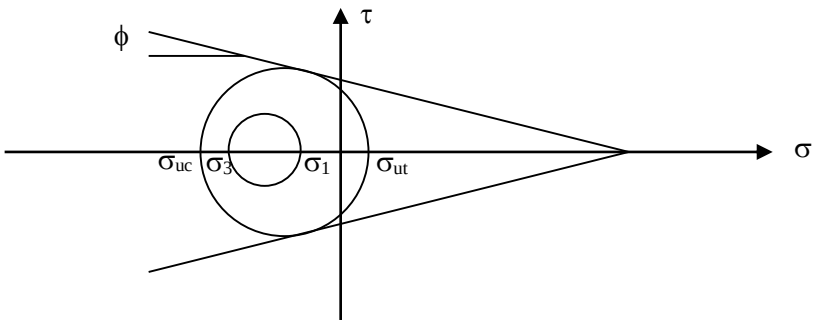
The maximum normal stress failure criterion states that failure occurs whenever σ_1 or σ_3 equals the failure strength of the material in tension or compression, respectively. For brittle materials, failure by fracture occurs when the ultimate strength (not yield strength as in ductile materials) is attained. Brittle materials are usually stronger in compression than tension. Fracture is deemed to occur either in the **plane normal to the maximum tensile stress** for a **tensile failure** or **in a plane oblique to the maximum compressive stress** as what appears to be a **shear fracture (but not as a rule on the plane of maximum shear stress)** for a failure in compression.

Mohr-Coulomb Failure Criterion

Brittle materials such as concrete, soil, gray cast iron and hardened tool steel are held together by frictional forces and they fail by sliding. The effect of the stress normal to the failure plane is important and should be included in the failure criterion. Fracture occurs when the maximum and minimum principal stresses satisfies the following equation.

$$(\sigma_1 / \sigma_{ut}) - (\sigma_3 / \sigma_{uc}) > 1.0$$

where σ_{ut} and σ_{uc} the represent the ultimate tensile and compressive strengths.



This theory is best used in brittle materials that are much stronger in compression. The Mohr-Coulomb equivalent stress is analogous to the Tresca failure criterion but incorporates the effect of the normal stress, unlike Tresca, which is pressure independent. Fracture is deemed to occur either in the **plane normal to the maximum tensile stress** for a **tensile failure** or **in a plane oblique to the maximum compressive stress** as what appears to be a **shear fracture (but not as a rule on the plane of maximum shear stress)** for a failure in compression.

A special case occurs in certain brittle materials such as **hardened tool steel** where the ultimate strengths are the **same** both in tension and compression. In this case the criteria becomes

$$(\sigma_1 / \sigma_{ut}) - (\sigma_3 / \sigma_{uc}) < 1.0$$

$$(\sigma_1 / \sigma_u) - (\sigma_3 / \sigma_u) < 1.0$$

$$\sigma_1 - \sigma_3 < \sigma_u$$

which is exactly similar (in expression only) to the ductile (maximum shear stress) Tresca $\tau_{13} = (\sigma_1 - \sigma_3)/2 = \sigma_y/2$ criteria. Of course the mode of failure remains different in the brittle material (plane normal to the maximum tensile stress for a tensile failure or in a plane 45° oblique to the maximum compressive stress) compared to ductile materials (45° to principal planes for failure in either tension or compression). But the fact remains that the maximum shear stress can be plotted on a post-processor to determine if the special equal (in tension and compression) strength brittle material meets the Mohr-Coulomb criterion or not.

BS 5950-Part1:2000 (cl. 2.4.4) Criterion

Brittle fracture is used to describe fast unstable fractures, in contrast to stable fractures such as **fatigue** or slow unstable fractures such as plastic yielding. Some metals such as **copper** and **aluminium** have a crystalline structure that enables them to **resist fast fracture** under all loading conditions and at all temperatures. This is not the case with **ferrous** alloys, particularly structural **steel**. The Liberty ships and the King Street bridge in Melbourne, the Sea Gem drilling rig for North Sea gas and the Alexander Kielland oil rig are a few examples of the casualties of brittle fracture. An important feature of steel is the **transistion temperature** between ductile and brittle fracture. To determine the risk of brittle fracture, the following are considered: -

I. minimum operating temperature
II. thickness of material to be used
III. steel grade
IV. stress level
V. detail; parent plate, weld metal or heat affected zone HAZ, holes
VI. strain-rate

In addition, the welding electrodes or other welding consumables should have a specified Charpy impact value equivalent to, or better than, that specified for the parent metal. **BS 5950-Part 1:2000 (cl. 2.4.4)** gives maximum permissible thicknesses of steel at given operating temperatures based on Charpy toughness energy absorption impact tests. The steel quality selected for each component should be such that the thickness t of each element

$$t \leq Kt_1$$

where t_1 is the limiting thickness at the appropriate minimum service temperature T_{min} for a given steel grade and quality

— if $T_{27J} \leq T_{min} + 20\ °C$:

$$t_1 \leq 50(1.2)^N \left[\frac{355}{Y_{nom}}\right]^{1.4}$$

— if $T_{27J} > T_{min} + 20\ °C$:

$$t_1 \leq 50(1.2)^N \left(\frac{35 + T_{min} - T_{27J}}{15}\right) \left[\frac{355}{Y_{nom}}\right]^{1.4}$$

in which:

$$N = \left(\frac{T_{min} - T_{27J}}{10}\right)$$

where T_{min} is the minimum service temperature, Y_{nom} is the nominal yield strength for thickness < 16mm for the steel grade used and T_{27J} is the test temperature at which 27J Charpy value was obtained as according to Table 7. Table 4 and Table 5 presents the calculated the t_1 values.

Table 7 — Charpy test temperature or equivalent test temperature T_{27J}

Steel quality	Product standard						
	BS EN 10025	BS EN 10113	BS EN 10137	BS EN 10155	BS EN 10210	BS EN 10219	BS 7668
JR	+20 °C	—	—	—	+20 °C	+20 °C	—
J0	0 °C	—	—	0 °C	0 °C	0 °C	0 °C
J2	–20 °C	—	—	–20 °C	–20 °C	–20 °C	—
K2	–30 °C[a]	—	—	–30 °C[a]	—	—	—
M	—	–30 °C[a]	—	—	—	–30 °C[a]	—
ML	—	–50 °C	—	—	—	–50 °C	—
N	—	–30 °C[a]	—	—	–30 °C[a]	–30 °C[a]	—
NL	—	–50 °C	—	—	–50 °C	–50 °C	—
Q	—	—	–20 °C[b]	—	—	—	—
QL	—	—	–40 °C[b]	—	—	—	—
QL1	—	—	–60 °C[b]	—	—	—	—
G	—	—	—	—	—	—	–15 °C

[a] Equivalent test temperature for 27 J. Product standard specifies 40 J at –20 °C.
[b] Equivalent test temperature for 27 J. Product standard specifies 30 J at the same temperature.

K is a factor that depends on the type of detail, the general stress level, the stress concentration effects and the strain conditions. Values are given in Table 3.

Table 3 — Factor *K* for type of detail, stress level and strain conditions

Type of detail or location	Components in tension due to factored loads		Components not subject to applied tension
	Stress ≥ $0.3Y_{nom}$	Stress < $0.3Y_{nom}$	
Plain steel	2	3	4
Drilled holes or reamed holes	1.5	2	3
Flame cut edges	1	1.5	2
Punched holes (un-reamed)	1	1.5	2
Welded, generally	1	1.5	2
Welded across ends of cover plates	0.5	0.75	1
Welded connections to unstiffened flanges, see **6.7.5**	0.5	0.75	1
NOTE 1 Where parts are required to withstand significant plastic deformation at the minimum service temperature (such as crash barriers or crane stops) *K* should be halved.			
NOTE 2 Baseplates attached to columns by nominal welds only, for the purposes of location in use and security in transit, should be classified as plain steel.			
NOTE 3 Welded attachments not exceeding 150 mm in length should not be classified as cover plates.			

In addition, the maximum thickness of the component should not exceed the maximum thickness t_2 at which the full Charpy impact value applies to the selected steel quality for that product type and steel grade, according to the relevant product standard, see Table 6.

Table 6 — Maximum thickness t_2[a] (mm)

Product standard	Steel grade or quality	Sections	Plates and flats	Hollow sections
BS EN 10025	S 275 *or* S 355	100	150	—
BS EN 10113-2	S 275 *or* S 355	150	150	—
	S 460	100	100	—
BS EN 10113-3	S 275, S 355 *or* S 460	150	63	—
BS EN 10137-2	S 460	—	150	—
BS EN 10155	J0WP *or* J2WP	40	16	—
	J0W, J2W *or* K2W	100	100	—
BS EN 10210-1	All	—	—	65
BS EN 10219-1	All	—	—	40
BS 7668	J0WPH	—	—	12
	J0WH *or* GWH	—	—	40
[a] Maximum thickness at which the full Charpy impact value given in the product standard applies.				

1.3.28.2.3 Large Deflections due to Insufficient Stiffness

The structure may be deemed to fail if certain sections deform considerably (static or dynamic response may be too high) under the applied excitation such that it fails to meet the required deflection tolerance criteria.

1.3.28.2.4 Buckling

Buckling is the sudden loss of stability or stiffness under the applied loads. The stress levels may not even be sufficiently high for failure by yielding to occur, in which case a linear elastic buckling analysis will predict this failure.

1.3.28.2.5 Fatigue

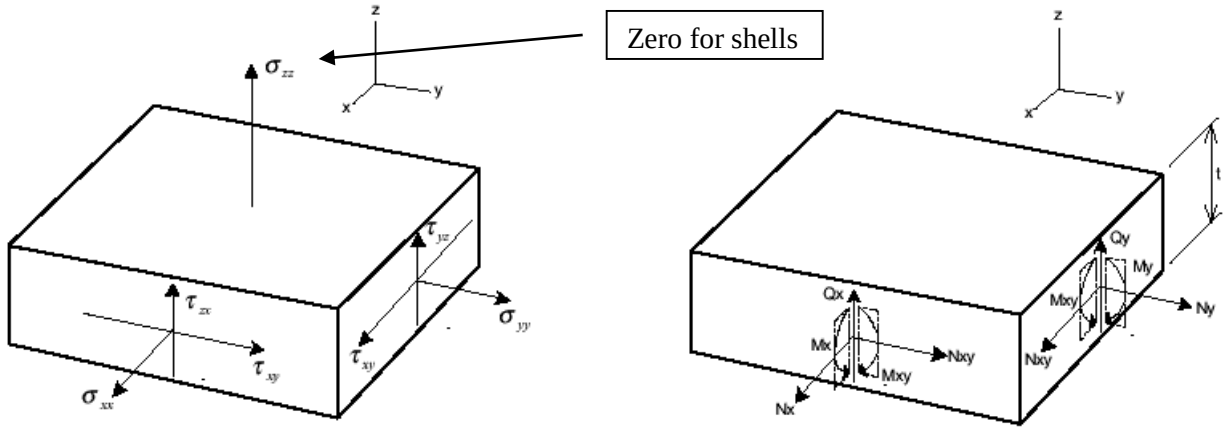
Components subject to repetitive cycles of loading may fail after a certain number of cycles. A pseudo-static or dynamic fatigue analysis is required to predict this failure.

1.3.28.2.6 Creep or Viscoelasticity

Structures under load gradually deform over time. This may be accounted for by modifying the stiffness modulus with an apparent creep modulus. Creep or viscoelasticity is a time dependent, plastic deformation under sustained load. The amount of creep experienced is a function of time, temperature and applied load. Creep is more significant for plastics even under room temperature. Creep is not significant for most metals until operating temperatures reach 35 to 70 percent of respective melting points.

1.3.28.3 Deriving Forces (Moments, Shear, Torsion & Axial Force) from Stresses in Shells

Moments, shear, torsion and axial forces are usually the design criteria for composite material such as reinforced concrete. Hence, to facilitate the design of these structural elements, often the computed stresses are summed to obtain the equivalent averaged forces. The stresses and hence forces obtained from a shell is summarized as follows.



The output stresses are

the in-plane normal stresses σ_{xx} and σ_{yy}, varying linear through the thickness
the in-plane shear stresses τ_{xy}, varying linearly through the thickness
the out-of-plane shear stresses τ_{xz} and τ_{yz}, constant through the thickness although from equilibrium considerations, they should vary parabolically

From these, the in-plane force tensors N_x, N_y and N_{xy}, the bending tensors M_x, M_y and M_{xy} and the through-thickness shear force vectors Q_x and Q_y.

$$N_x = \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{xx} dz \qquad N_y = \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{yy} dz \qquad N_{xy} = \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{yx} dz$$

$$Q_x = \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{xz} dz \qquad Q_y = \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{yz} dz$$

$$M_x = \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{xx} z\, dz \qquad M_y = \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{yy} z\, dz \qquad M_{xy} = \int_{-\frac{t}{2}}^{\frac{t}{2}} \sigma_{yx} z\, dz$$

These simplify to

$$M_x = \sigma_{bxx} t^2/6 \qquad M_y = \sigma_{byy} t^2/6 \qquad M_{xy} = \sigma_{bxy} t^2/6$$
$$N_x = \sigma_{axx} t \qquad N_y = \sigma_{ayy} t \qquad N_{xy} = \sigma_{axy} t$$
$$Q_x = \sigma_{axz} t \qquad Q_y = \sigma_{ayz} t$$

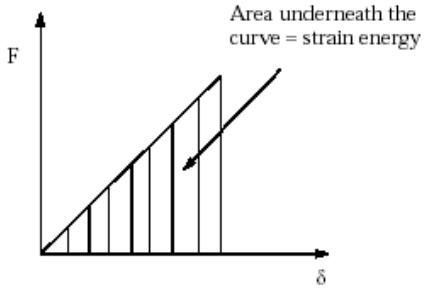
Note that these are forces and moments based on the stresses and hence act over the same width that the stresses act. For design purposes, often a per meter width design moment and force are required. Hence, the values of moments and forces of all elements over the physical width of a meter should be averaged for the design values.

On a further note on design, the complications of designing for the torsional moment (derived from the in-plane bending shear stresses) M_{xy} is avoided by increasing the design moments M_{xx} and M_{yy} as follows.

$$|M_x| + |M_{xy}| \qquad |M_y| + |M_{yx}|$$

1.3.29 The Element Strain Energy Density

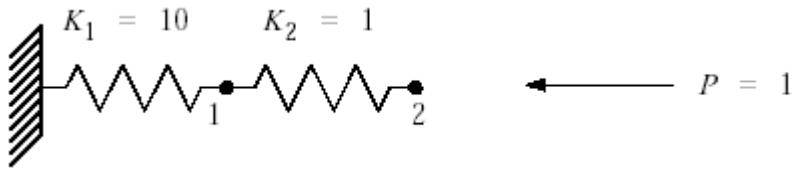
The element strain energy is basically the elastic energy stored in the structural element. As an example, if you hold onto one end of the spring and push slowly on the other end starting from rest, the load deflection curve looks as follows for small deflections.



The strain energy $U = 0.5F\delta = 0.5k\delta^2$. The strain energy distribution within a structure denotes the elements that are working the hardest, i.e. the higher the strain energy, the harder the element is working. Hence, for a given loadcase it provides information on the elements that should be stiffened to increase the overall stiffness (and hence to minimize the deflections) to that loadcase. The required request in MSC.NASTRAN is

$ CASE CONTROL SECTION
ESE(<PRINT,PUNCH,PLOT>) = ALL/<Element Set ID>

To illustrate this concept, consider the following springs in series. The goal is to reduce the tip deflection at grid point 2. Without performing any calculation, it is quite obvious that stiffening K_2 is more efficient than stiffening K_1. The next step is to calculate the strain energy and see if it also guides you in the same direction.



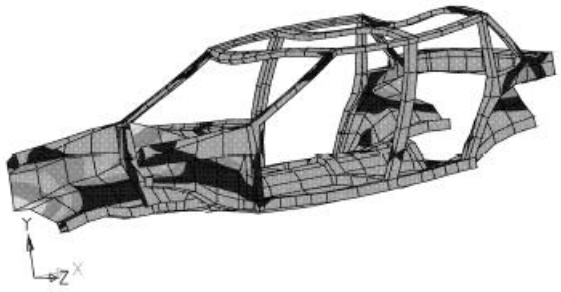
The deflections and element strain energy are

$$\delta_1 = \frac{P}{K_1};\ \delta_2 = \frac{P}{K_1} + \frac{P}{K_2}$$

$$U_1 = 0.5\,(K_1)(\delta_1)^2 = 0.5\,(10)\left(\frac{1}{10}\right)^2 = 0.05$$

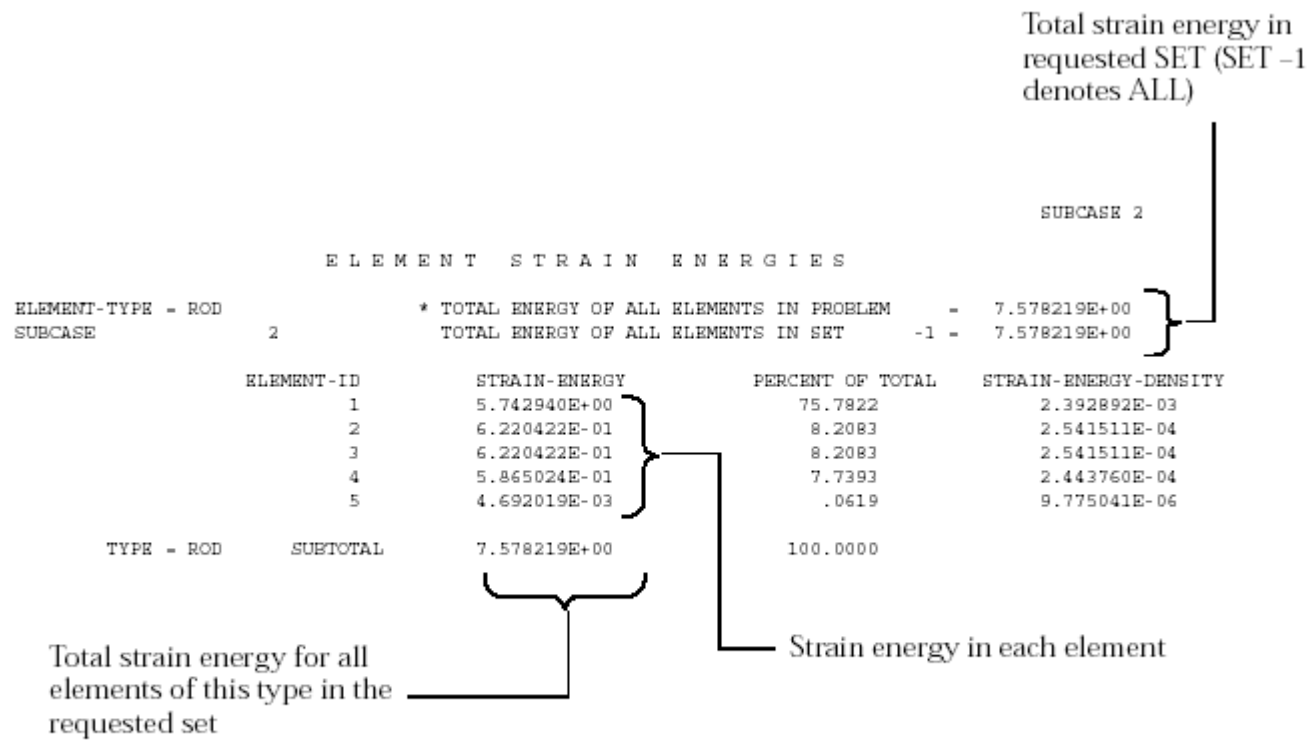
$$U_2 = 0.5(K_2)(\delta_2 - \delta_1)^2 = 0.5(1)\left[\left(\frac{1}{10}\right) + \left(\frac{1}{1}\right) - \left(\frac{1}{10}\right)\right]^2 = 0.5$$

Since U_2 is an order of magnitude greater than U_1, clearly it is better to stiffen K_2 than K_1 to reduce the deflections at the tip. The element strain energy distribution for a car model is shown below (courtesy of Lapcad Eng).



The strain energy density is an effective design tool for linear static analysis (SOL 101), buckling analysis (SOL 105 and SOL 106) and for dynamic linear modal analysis (SOL 103).

The ESE request outputs the following.



Note: This output is incorrect in the presence of thermal loads or element deformations.

1.3.30 Error Estimation

1.3.30.1 GL, ML Analysis Error Estimation: h- or p- Refinement

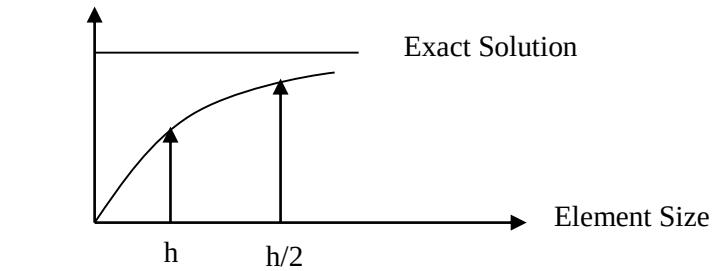
Finite elements attempt model a continuum with discrete parameters, i.e. a finite number of DOFs. As long as the following conditions are met

(i) the analysis is linear
(ii) the finite elements are based on the variational approach (or the Galerkin approach)
(iii) the geometry and material properties are represented accurately
(iv) continuity requirements of the shape functions are satisfied
(v) the integration of the discrete system of equations is accurate
(vi) all essential boundary conditions are associated with zero values

the work done by the total nodal loads due to nodal displacements as predicted by the discrete system will always be less or equal to the exact value

$$\{\{f+P)^T\{u\}\}_{discrete} \leq \{\{f+P)^T\{u\}\}_{discrete}$$

Hence, the discrete structural analysis based on a finite number of DOFs will always present a stiffer response than the exact solution provided the above conditions are met. Hence, a finer mesh will generally be associated with a more flexible response and hence a closer to exact approximation. The stiffer response occurs due to the fact the that the chosen shape functions cannot represent all the possible modes of deformation of the structure. A useful method of approximate error estimation is described as follows. The approximate order of error has been established, but not the percentage of error itself relative to the exact solution. We shall illustrate the principle for the in-plane displacement and strain or stress of a triangular (or rectangular for that matter) finite element.



An estimate of the exact solution can be obtained by performing two linear analyses with either h- or p-refinement. Employing h-refinement, we can establish the estimate of the exact displacement and stress (or strain) as

$$\frac{u_{exact}-u_1}{u_{exact}-u_2}=\frac{O(h)^2}{O(h/2)^2}=4,\ \text{hence}\ u_{exact}=\frac{4u_2-u_1}{3}$$

$$\frac{\sigma_{exact}-\sigma_1}{\sigma_{exact}-\sigma_2}=\frac{O(h)^1}{O(h/2)^1}=2,\ \text{hence}\ \sigma_{exact}=2\sigma_2-\sigma_1$$

where the subscripts 1 and 2 refer to the first and second (refined) analysis respectively. Hence, an approximate absolute percentage error in the displacement and stress (or strain) is then given by

$$\frac{u_{exact}-u_2}{u_{exact}}\,x\,100\% \qquad \text{and} \qquad \frac{\sigma_{exact}-\sigma_2}{\sigma_{exact}}\,x\,100\%$$

1.3.30.2 GNL, MNL Analysis Error Estimation: h- or p- Refinement

The mesh should be continuously refined until the variation is of no practical relevance.

1.3.30.3 Adaptive Analysis in MSC.NASTRAN with p-Elements

p-elements are elements that have variable degrees-of-freedom. The user can specify the polynomial order for each element (in the 3 direction Px, Py, and Pz) and the program will generate the degrees-of-freedom required. The p-elements are CBEAM, CTRIA, CQUAD, CHEXA, CPENTA and CTETRA. The properties of the p-elements will be specified using the PSOLID, PSHELL, PBEAM, or PBEAML entry. These elements may use either isotropic materials as defined on the MAT1 entry or anisotropic materials as defined on the MAT9 entry. One of the main applications of p-version elements is detailed stress analysis. The p-elements have higher-order polynomials, which provide better representation of complex stress fields. For these complex stress fields, the geometry, loads, and boundary conditions must be represented accurately. This includes modelling fillets instead of sharp corners, distributed loads and constraints instead of point loads and constraints, etc. A more detailed model leads to more detailed results.

The accuracy of the analysis is controlled primarily by the polynomial level, not by the element size. Generally the user needs to only use the minimum number of elements necessary to adequately model the geometry, independent of expected result characteristics. Polynomials levels can then be assigned based on areas of the model in which the user is interested, and areas of the model in which the user is not. (Of course the same polynomial level can be assigned everywhere for a minimum of user involvement, but this is not as efficient.) If a more accurate answer is necessary, the polynomial levels can be increased, either manually by the user or automatically by the program, until the answers reach the specified accuracy, without changing the mesh. Adding degrees of freedom until the error decreases to a specified level is known as adaptivity. By adding higher-order polynomials instead of refining the mesh, a faster rate of convergence, and therefore less iteration, can be achieved.

In MSC.NASTRAN, p-elements have been added to complement the existing h-elements. The error estimator requires only a single analysis, not the difference between two analyses. For linear elements, it is based on the grid-point stress discontinuity; for higher polynomial orders, it considers the contribution of the additional terms for the individual edge, face, and body functions. Therefore it provides efficient information because of the directional sensitivity within the elements.

The main advantage of implementing p-elements in MSC.NASTRAN is the ability to combine them with the existing h-elements for global/local analysis. The mesh could consist mostly of h-version elements, with p-version elements in the areas of interest. This method of global/local interface is the most accurate, because it provides both stiffness and loads information, rather than transferring just boundary displacements or tractions into the local model. h-version elements still have several advantages. They are better for global behavior, such as loads analysis, where the exact geometry is not used. They are better for strongly singular problems, such as nonlinear problems. Approximations, such as sharp corners, point loads, point constraints, MPC's, etc. may be used. For a given iteration, they tend to have shorter solution times and use less disk space. Finally, h-elements are a very mature technology. Putting both approaches in the same program combines the advantages for general problems. The p-elements may be connected directly to the existing h-elements, and continuity is automatically enforced. This provides global/local analysis in a single run. Most general problems have both some global and some local aspects; i.e., regions where h-elements and p-elements, respectively, are appropriate. Therefore the best approach is combined h- and p-adaptivity, with different criteria in different parts of the model.

For p-version elements in detailed stress analysis, it is important to use distributed loads. Concentrated forces cause singularities in the stress field, and therefore should be avoided. The PLOAD4 and GRAV entries can also be applied to p-elements. The FORCE entry can be used, but should only be used where the adjacent elements have a fixed p-level and the results are not of interest.

As with the loads, it is important to use distributed boundary conditions for p-version elements in detailed stress analysis. Concentrated boundary conditions also cause singularities in the stress field and should be avoided. The current SPC and SPC1 entries can be used, but should only be used to prevent rigid-body motion or used where the adjacent elements have a fixed p-level and the results are not of interest.

Clearly, with p-elements, a different geometry definition is required. In linear elements, only the corner grid definitions are required. In parabolic elements, the corner and mid-side grid definitions are required to map the curvature of the surface. The Bulk Data entries to define the associativity between the finite element model and the geometrical model are the **FEEDGE** and **FEFACE** entries for the edges and faces, respectively. Geometrical curves and surfaces are defined by the **GMCURV** and **GMSURF** entries. The **GMLOAD** entry has been defined to apply (non-singular) loads on these geometric surfaces for the p-version elements. The **GMSPC** and **GMBC** entries have been defined to apply (non-singular) boundary constraints for the p-version elements.

In order for an element to be recognized as a p-version element, it must be assigned the polynomial values in the new **PVAL** Bulk Data entry, which is referenced in the **ADAPT** Bulk Data entry. In order for the p-element to be adaptive, the adaptivity parameters must be assigned in the ADAPT entry. The new PVAL Bulk Data entry assigns the polynomial levels for the three directions to a single element or a set of elements, and is referenced in the ADAPT Bulk Data entry. By default, the three directions are along the element edges. If a coordinate system is specified, the polynomial level of each edge is set to a weighted combination of the coordinate directions. There may be multiple PVALs with the same ID; all of the elements do not have to use the same entry. Each p-version element must have a starting, minimum, and maximum PVAL ID; however, these do not need to be unique. Since the polynomial levels can not decrease, the starting and minimum PVAL IDs can be the same. If the starting and maximum PVAL IDs are the same, the element will have a fixed p-level. After the p-value distribution has been defined, all of the values will be resolved to ensure displacement continuity. This includes selecting the highest p-level specified for elements with common edges, decreasing the p-level for p-elements adjacent to h-elements, and increasing the p-level on curved edges to adequately map them. The new ADAPT Bulk Data entry controls the p adaptivity. The ADAPT Bulk Data entry is referenced with the ADAPT Case Control command, and there may only be one unique ADAPT command in the Bulk Data file. The ADAPT entry contains the IDs for the starting, minimum, and maximum p-value distributions and the maximum number of iterations. It also contains the adaptivity parameters, which may be assigned differently to different sets of elements. These sets of elements do not have to correspond with any sets used in the PVAL entries. The adaptivity type may be element-by-element p adaptive, uniform p-adaptive, no change, or list of p-distributions. For each set in the ADAPT entry, the error estimator method, error tolerance, and stress and strain tolerances are specified. If a p-version element is not included in one of the sets, it is not adaptive. For a non-adaptive p-version solution, the ADAPT Bulk Data entry may specify a maximum of one iteration, or may specify an adaptivity type of no change for all the elements. A third way is to use the **PSET** Bulk Data entries, which have the same format as the PVAL entry, and reference them with the ADAPT Case Control command.

Since the p-version analysis may have multiple iterations, the output control must have the capability of differing among the iterations. For this reason, the OUTPUT Bulk Data entry was added. For a given set of elements, the conventional displacements, stresses, and strains, as well as the new element errors and polynomial values, may be printed, plotted, or punched for the given iteration. The OUTPUT entry is referenced by the DATAREC Case Control command. The conventional Case Control commands still apply to all iterations, but the OUTPUT entry has precedence. The p-version elements also tend to be larger than h-version elements, and have higher order distributions of displacements, stresses, and strains. Therefore results only at the grid points are not adequate, especially for most plotting packages, which use linear interpolation. For data recovery, the p-version elements are divided into view-grids and view-elements in order to better visualize the results. The new OUTRCV Bulk Data entry defines the number of view-elements in each element direction and the output coordinate system for the view-grids and view-elements. The OUTRCV entry is referenced by the OUTRCV Case Control command. The coordinates of the view-grids may be output using the VUGRID Case Control command.

When undertaking a h- or p- refinement, it is important to remember to also alter the boundary conditions and constraints i.e. SPCs and RBEs such that the additional h-refined nodes and the p-refined mid-side nodes on the edges and faces of the supports and constraints also participate.

	h-Version Elements	p-Version Elements
Geometry		GMCURV GMSURF
Geometrical Associativity		FEEDGE FEFACE
Coordinate Systems	CORDij	CORDij GMCORD
Grids Points	GRID	GRID POINT

	h-Version Elements	p-Version Elements
Elements	CHEXA CPENTA CTETRA CQUAD CTRIA CBEAM	CHEXA CPENTA CTETRA CQUAD CTRIA CBEAM
Element Properties	PSOLID PSHELL PBEAM	PSOLID PSHELL PBEAM
Material Properties	MATi	MATi
Loads Thermal Loads Gravity Loads	 PLOAD4 FORCE TEMP, TEMPD TEMPP1 TEMPRB GRAV	GMLOAD PLOAD4 FORCE* TEMPF TEMP, TEMPD TEMPP1 TEMPRB GRAV
Constraints Boundary Conditions	 SPC, SPC1 SPCD	GMSPC SPC, SPC1* GMBC SPCD*
Polynomial Values		PVAL PSET
Adaptivity Control		ADAPT
Data Recovery		OUTPUT OUTRCV
Miscellaneous		DEQATN TABLE3D
Sets Definition		SETS DEFINITION

* Should only be used under specific conditions.

1.3.30.4 Stress Discontinuity Error Estimator

Stress recovery concepts, namely

I. stress recovery at the element Gauss points, then
II. subsequent stress extrapolation to the element grids, and
III. eventually, the (transformation into global coordinates and) averaging of the elemental stresses to yield the grid point stresses

have been covered. Here we shall discuss error estimation based on grid and element stress discontinuities.

For discussion purposes, the averaging process used to compute the stress components at the grid points can be represented in the form

$$\sigma_g = \sum_{i=1}^{N_e} (W_i \sigma_{ei})$$

σ_g = the weighted mean value of the stress component computed at the grid point.

σ_{ei} = the value of the stress component in the i-th element $i = 1, 2, \ldots, N_e$ connected to the grid point. σ_{ei} is in the same coordinate system as σ_g.

W_i = the weighting factor assigned to the *i*-th element. The sum of the values of must equal 1. This requirement assures that all computed statistics are unbiased. The attribute of being unbiased implies that the variance is equal to the mean square error. Equal weighting, i.e., $W_i = 1/N_e$, is assumed in MSC.Nastran.

1.3.30.4.1 Grid Point Stress Discontinuity

An estimate of the probable error in the stress component δ_g at the grid point is

$$\delta_g = \sqrt{\sum_{i=1}^{N_e} (W_i \delta_{ei})^2} = \frac{1}{\sqrt{N_e}} \sqrt{\frac{\sum_{i=1}^{N_e} (\delta_{ei})^2}{N_e}} \qquad \delta_{ei} = \sigma_{ei} - \sigma_g$$

Thus, the probable error in δ_g is the root mean square error in δ_{ei} divided by $\sqrt{N_e}$. The probable error δ_g provides an estimated error for each of the stress components. It is probably more useful and more desirable to combine these six (in general there are six component stresses) estimated errors into a single representative error measure at each grid point. An approximate root mean square value of the three estimated errors for each of the stress components offers such a representative error measure.

$$\text{ERROR ESTIMATE} = \sqrt{\frac{\sum_{i=1}^{N_c} (\delta_{gi})^2}{N_c}}$$

where N_C is the number of stress components, three for shells and six for solids.

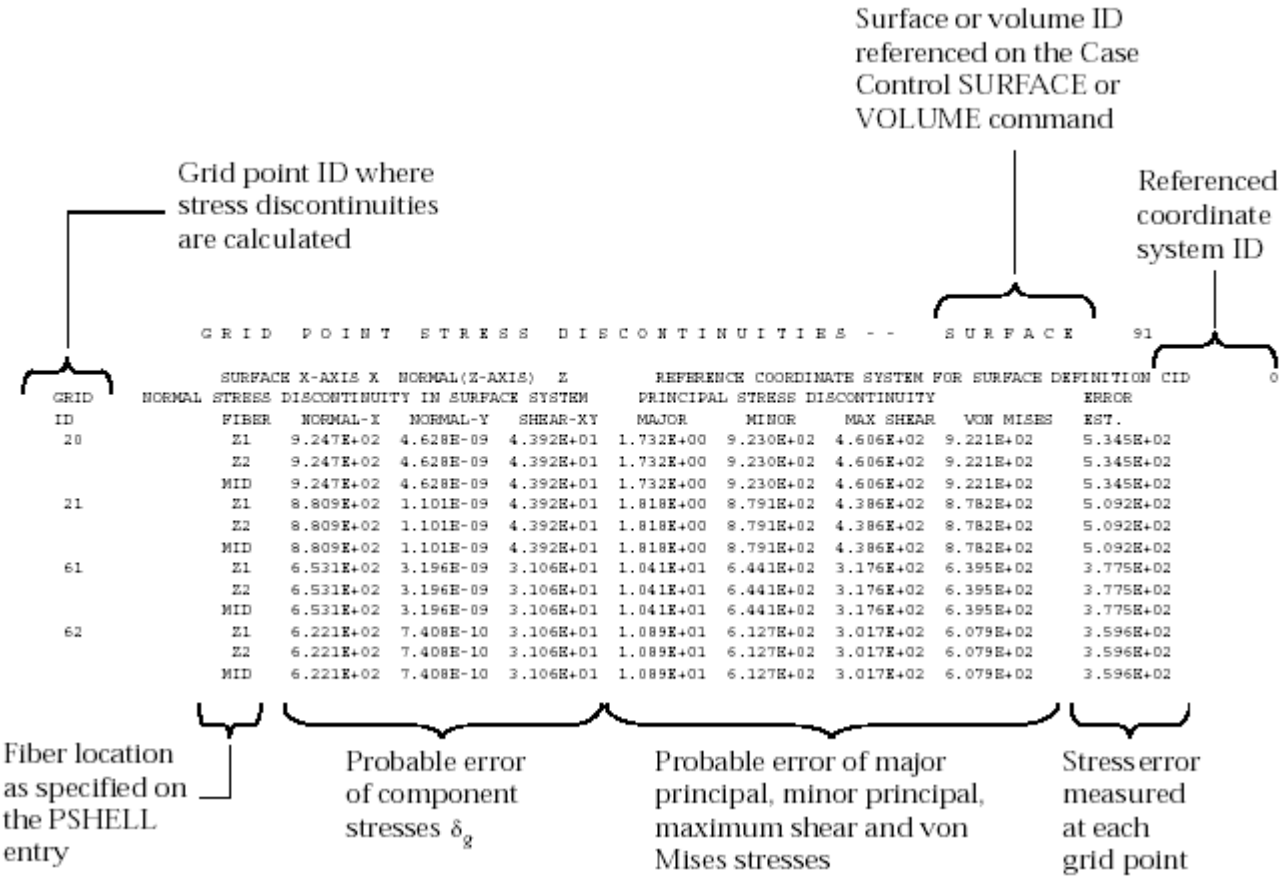
Smaller values of δ_g and ERROR ESTIMATE are usually obtained for elements with vertex stresses, i.e. most elements except CTRIA3 and CQUAD4 with STRESS (CENTER) option. CQUAD4 with STRESS (CORNER) option will usually yield a smaller δ_g and ERROR ESTIMATE. These seemingly large values of error estimators for CQUAD4 elements are not uncommon occurrences in statistical error measures. For example, the mean and variance have direct analogies in engineering mechanics. The mean is analogous to the centroid of a body and is thus a measure of where the mass is centered. The variance is the second moment about the mean, and it tends to be

small if the majority of the mass is concentrated about the centroid. As the mass is dispersed further from the centroid, the moment of inertia tends to increase. If the mass is concentrated at the centroid (as it might be in some idealized lumped mass models), the moment of inertia becomes zero. Thus, the error estimates for CQUAD4 elements tend toward larger values because the data used to compute the estimators is dispersed relatively far from the mean. For those elements that provide stress data at vertices, the estimators tend toward smaller values because the stress data tends to be concentrated about the mean in well-designed finite element models.

You are cautioned that very inaccurate values of these error estimators may occur at the edges of defined SURFACEs and on the faces of defined VOLUMEs. In summary, the error estimators under discussion can, in some cases, be highly inaccurate. Nevertheless, these data are quite useful when interpreted properly.

Provided that grid point stresses are requested (printed by GPSTRESS and plotted by STRFIELD, with SURFACE and/or VOLUME definition), grid point stress discontinuities can be requested with GPSDCON = < Set of Surface / Volume IDs >. Note that GPSTRESS, STRFIELD and GPSDCON all reference the same set.

Note that stress continuity data is not provided when both plate and solid elements are connected to a grid point that is involved in stress discontinuity calculations. The printed output is presented as follows.



Note: This output is obtained by using the STRFIELD, STRESS, GPSDCON, and SURFACE (or VOLUME) Case Control command.

Grid point stress discontinuity surface output is available for all plate and shell elements (CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR).

Grid point stress discontinuity volume output is available for all solid elements (CHEXA, CPENTA, and CTETRA).

1.3.30.4.2 Element Stress Discontinuity

The root mean square errors for each stress component of an element may be computed from the values of δ_{ei} that are computed for each of the N_g connected vertex grid points where $\delta_{ei} = \sigma_e - \sigma_{gi}$. This computation is done for each stress component for all of the elements of interest.

$$\delta_e = \sqrt{\frac{\sum_{i=1}^{N_g} (\delta_{ei})^2}{N_g}}$$

It is desirable to combine these six (in general there are six stress component) estimated errors into a single representative error estimate for each element. The root mean square value of the stress error estimates for each of the stress components offers such a representative error measure.

$$\text{ERROR ESTIMATE} = \sqrt{\frac{\sum_{i=1}^{N_c} (\delta_{ei})^2}{N_c}}$$

Provided that grid point stresses are requested (printed by GPSTRESS and plotted by STRFIELD, with SURFACE and/or VOLUME definition), element stress discontinuities can be requested with ELSDCON = < Set of Surface / Volume IDs >. Note that GPSTRESS, STRFIELD and ELSDCON all reference the same set.

Surface or volume ID referenced on the Case Control SURFACE or VOLUME command

Element ID where element stress discontinuities are calculated

Referenced coordinate system ID

E L E M E N T S T R E S S D I S C O N T I N U I T I E S - - S U R F A C E 91

SURFACE X-AXIS X NORMAL(Z-AXIS) Z REFERENCE COORDINATE SYSTEM FOR SURFACE DEFINITION CID 0

ELEMENT ID	ELEMENT TYPE	NORMAL STRESS DISCONTINUITY IN SURFACE SYSTEM FIBER	NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRESS DISCONTINUITY MAJOR	MINOR	MAX SHEAR	VON MISES	ERROR EST.
19	QUAD4	Z1	1.322E+03	7.758E-09	6.212E+01	5.662E+00	1.317E+03	6.562E+02	1.315E+03	7.643E+02
		Z2	1.322E+03	7.758E-09	6.212E+01	5.662E+00	1.317E+03	6.562E+02	1.315E+03	7.643E+02
		MID	1.322E+03	7.758E-09	6.212E+01	5.662E+00	1.317E+03	6.562E+02	1.315E+03	7.643E+02
20	QUAD4	Z1	1.277E+03	4.757E-09	6.212E+01	5.936E+00	1.272E+03	6.333E+02	1.269E+03	7.382E+02
		Z2	1.277E+03	4.757E-09	6.212E+01	5.936E+00	1.272E+03	6.333E+02	1.269E+03	7.382E+02
		MID	1.277E+03	4.757E-09	6.212E+01	5.936E+00	1.272E+03	6.333E+02	1.269E+03	7.382E+02
59	QUAD4	Z1	1.321E+03	7.630E-09	6.212E+01	1.698E+02	1.194E+03	5.395E+02	1.110E+03	7.635E+02
		Z2	1.321E+03	7.630E-09	6.212E+01	1.698E+02	1.194E+03	5.395E+02	1.110E+03	7.635E+02
		MID	1.321E+03	7.630E-09	6.212E+01	1.698E+02	1.194E+03	5.395E+02	1.110E+03	7.635E+02
60	QUAD4	Z1	1.274E+03	4.520E-09	6.212E+01	1.787E+02	1.140E+03	5.094E+02	1.050E+03	7.365E+02
		Z2	1.274E+03	4.520E-09	6.212E+01	1.787E+02	1.140E+03	5.094E+02	1.050E+03	7.365E+02
		MID	1.274E+03	4.520E-09	6.212E+01	1.787E+02	1.140E+03	5.094E+02	1.050E+03	7.365E+02

Fiber location as specified on the PSHELL entry

Probable error of component stresses δ_e

Probable error of major principal, minor principal, maximum shear and von Mises stresses

Stress error measured at each element

Note: This output is obtained by using the STRFIELD, STRESS, ELSDCON, and SURFACE (or VOLUME) Case Control commands.

Element stress discontinuity surface output is available for all plate and shell elements (CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR).

Element stress discontinuity volume output is available for all solid elements (CHEXA, CPENTA, and CTETRA).

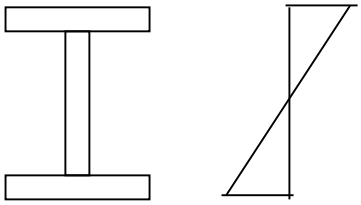
1.3.30.5 Hand Verification: Deriving Stresses From Forces (Axial, Moments, Shear and Torsion) in Beam Elements and Stresses Due to External Loading of General Shell and Solid Continuum

1.3.30.5.1 Beam – Axial Force

Axial forces cause uniform longitudinal **normal** stresses according to $\sigma = F/A$ **(Engineers Axial Theory $\sigma = F/A = Edu/dx$)**.

1.3.30.5.2 Beam – Bending Moment and Shear Force

Assuming linear elastic theory and small displacement theory (i.e. that plane sections remain plane and orthogonal to the neutral axis on bending), **bending moments** cause longitudinal **normal** bending stresses varying linearly across the section according to $\sigma = My/I$ **(Engineers Bending Theory $\sigma/y = M/I = E/R = Ed^2u/dx^2$)**, hence zero at the neutral axis and maximum at the flanges.



Longitudinal normal stresses $\sigma = My/I$

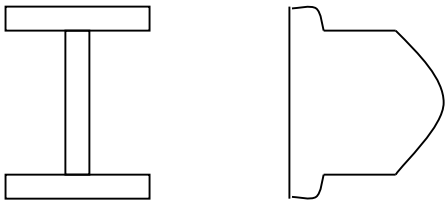
Defining x as the longitudinal axis, the general symmetrical bending equation is

$$\sigma = \frac{M_z}{I_{zz}} y + \frac{M_y}{I_{yy}} z$$

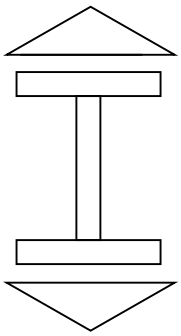
Whenever bending occurs about an unsymmetrical axis, the unsymmetrical bending equation should be used.

$$\sigma = \frac{\left(M_z I_{yy} + M_y I_{yz}\right)}{\left(I_{yy} I_{zz} - I_{yz}^2\right)} y + \frac{\left(M_z I_{yz} + M_y I_{zz}\right)}{\left(I_{yy} I_{zz} - I_{yz}^2\right)} z$$

Shear forces cause vertical and complimentary longitudinal horizontal **shear** stresses varying parabolically across the vertical dimension of the section according to $\tau = VQ/Ib$ **(Engineers Shear Theory $\tau = VQ/(Ib)$ or on average $V/(GA_s) = Gdu/dx$)** where b is the width, V is the shear force, I the second moment of area about the neutral axis and Q the first moment of area about the neutral axis of the area away from the point of concern. The shear stress is hence maximum at the neutral axis and is carried primarily by the web. Shear forces also cause horizontal and complimentary longitudinal shear stresses varying linearly in the flange according to $\tau = VQ/Ib$.



Vertical and complimentary longitudinal shear stresses $\tau = VQ/Ib$



Horizontal and complimentary longitudinal shear stresses $\tau = VQ/Ib$

The shear deformation expression is often simplified from VQ/(GIb) to V/(GA$_s$) i.e. V/(GK$_s$A). **Section 1.4.2.2** gives values of K$_s$ for typical beam sections which could be used to derive **averaged (not maximum)** shear stresses.

1.3.30.5.3 Circular Closed (Hollow or Solid) Beam – Torsional Moment

A member is subjected to **torsion** from a shear force when it is applied not at its **shear center**. For a **round section (hollow or solid)**, if plane and parallel cross sections remain plane and parallel, **torsion** causes complimentary **shear** stresses (a state of pure shear) to develop at a radius r according to $\tau = Tr/J$ **(Engineers Torsion Theory $\tau/r = T/J = Gd\phi/dx$)** where the torsional constant J is equal to the sections polar moment of inertia for this round section. The maximum hence occurs at the outermost radius.

1.3.30.5.4 Non-circular Closed (Hollow) Beam – Torsional Moment

That of a **thin-walled arbitrary closed** cross-section, $\tau = T / (2A_e t)$ where A_e is the area enclosed by mid thickness.

1.3.30.5.5 Non-circular Open Beam – Torsional Moment

The torsional stress distribution for a noncircular section is much more complicated with J often replaced by K$_t$. **Section 1.4.2.1** gives values of K$_t$ for typical beam sections which could be used to derive torsional shear stresses according to τ = function (T, K$_t$, geometry)[8]. The expression $\tau = Tr/K_t$ **CANNOT** be used for general non-circular sections.

1.3.30.5.6 Cylinder – Internal and/or External Pressure

A **capped cylinder** subject to internal or external pressure will develop tangential (hoop), radial and longitudinal **normal** stresses. Denoting p_i and p_o as internal and external pressure, r_i and r_o as internal and external radii, and r as the general radius, these stresses are

$$\sigma_{tangential} = \frac{p_i r_i^2 - p_o r_o^2 - r_i^2 r_o^2 (p_o - p_i)/r^2}{r_o^2 - r_i^2}$$

$$\sigma_{radial} = \frac{p_i r_i^2 - p_o r_o^2 + r_i^2 r_o^2 (p_o - p_i)/r^2}{r_o^2 - r_i^2}$$

$$\sigma_{longitudinal} = \frac{p_i r_i^2 - p_o r_o^2}{r_o^2 - r_i^2}$$

[8] ROARK, R.J. and YOUNG, W.C. *Roark's Formulas for Stress and Strain 7th Edition.* McGraw-Hill, London.

The tangential and radial stresses are a long way away from the end caps and the longitudinal stress is constant along the length.

1.3.30.5.7 Thin Walled Cylinder – Internal Pressure

For a thin-walled cylinder with diameter d, and thickness t less than 1/20 of its radius, with negligible outside pressure, the maximum **tangential** (positive for tension) and constant **longitudinal** (positive for tension) **normal** stresses are

$$\sigma_{\text{tangential, max}} = \frac{p_i (d_i + t)}{2t}$$

$$\sigma_{\text{longitudinal}} = \frac{p_i d_i}{4t}$$

whilst the **radial normal** stress would simply be the internal pressure (negative for compression) on the element on the inside surface of the cylinder and zero on the element on the outside surface.

1.3.30.5.8 Thin Walled Sphere – Internal Pressure

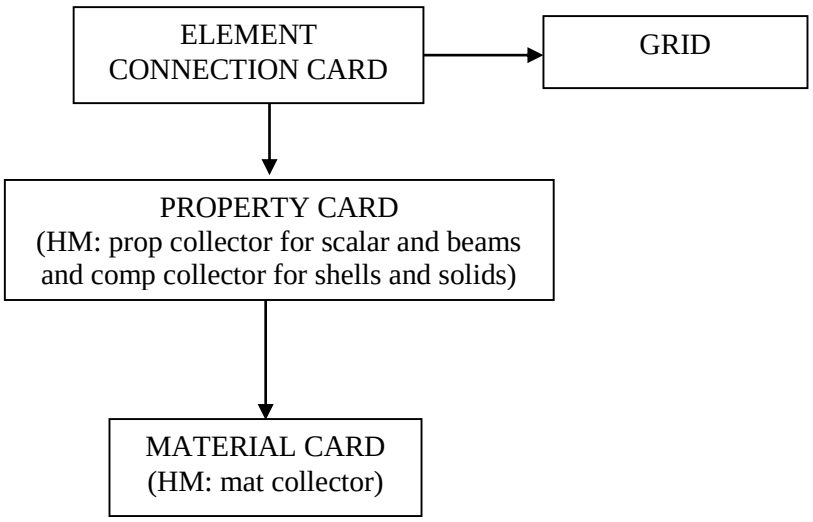
For a thin-walled sphere with diameter d and thickness t, the biaxial (in two orthogonal directions) **tangential** (positive for tension) **normal** stresses are

$$\sigma_{\text{tangential, max}} = \frac{p_i d}{2t}$$

whilst the **radial normal** stress would simply be the internal pressure (negative for compression) on the element on the inside surface of the cylinder and zero on the element on the outside surface.

1.4 Stiffness Element Cards

The basic MSC.NASTRAN element connectivity structure is shown below.



Basic MSC.NASTRAN element and property card structure

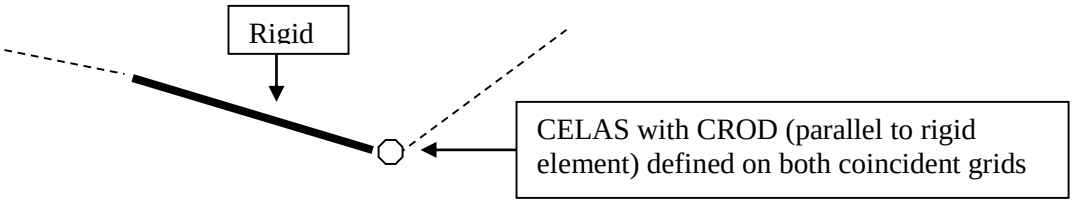
For beams, the element definition and element stress recovery are performed in the element coordinate system.

For shells, the element definition and element stress recovery are performed in the **element coordinate system by default (but defined by MCID entry on element connection card; 0 for basic projected, > 0 for user projected, < blank > for element coordinate system, > 0.0 and < 360.0 for angled from side n1-n2 of element)**.

For solid elements, the element definition and element stress recovery are performed in the **basic coordinate system by default (but defined on CORDM field of PSOLID; 0 for basic, > 0 for user-defined, -1 for element coordinate system)**.

1.4.1 Scalar CELAS Element

CELAS element is used to model springs (that provide translational or rotational stiffnesses) connecting single DOFs at two grid points that should be coincident. If one of the grid points is omitted, then the spring is assumed to be grounded (i.e. a grid which is constrained in displacement). As with any scalar element, the CELAS element **MUST BE DEFINED** as a **ZERO LENGTH** element (as a scalar element should not have a dimension anyway), and not one with a finite length. Otherwise, any forces applied to the grids by the scalar element **WILL INCORRECTLY** induce moments on the structure if the direction of the scalar stiffness (and hence force) is not parallel with the finite dimension. If the grids are not coincident, a difference in displacement may be caused by a rigid body rotation. This situation causes forces in springs and a net moment on the structure, i.e. and internal constraint. Internal constraints can be detected when there is a discrepancy between the OLOAD resultants and SPCFORCE resultants. Although the results with internal constraints could be plausible for static linear analysis, they can be terribly incorrect for eigenvalue analysis. For best practice, always utilize rigid elements to add finite length, even if scalar stiffness parallel to dimension, note that CBUSH elements generated internal rigids.



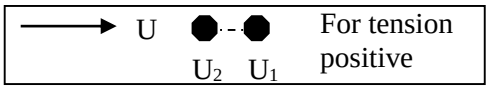
The DOFs connected by the spring are in the coordinate system defined at the grids. It is convenient if the springs are oriented parallel to the Basic System. In general the direction of the scalar stiffness value will not be coincident with a global DOF. This necessitates the definition of a local coordinate system using CORD1R or CORD2R at the grid where the scalar stiffness is required. This system should be specified in Field 7 of both connected GRID cards. In such situations using a CROD element is sometimes easier than using a spring although with CROD elements, coincident nodes cannot be defined.

$ BULK DATA									
CELAS1	**EID**	**PID**	**GA**	**DOF C1**	**GB**	**DOF C2**	**Structural Damping GE**	**S**	
PELAS	**PID1**	**K1**	**PID2**	**K2**	**PID3**	**K3**	**PID4**	**K4**	

CELAS2	**EID**	**K**	**GA**	**DOF C1**	**GB**	**DOF C2**	**Structural Damping GE**	**S**	

When you use CELASi elements to represent concentrated springs between two components of translation, the directions of the two components must be coaxial. Even small deviations in direction can induce a significant moment to your model that does not exist in your physical structure. It is recommended that when a CELASi element is used, the locations of the two end points be coincident in order to avoid this type of problem. If the two end points are not coincident, you should consider using a CROD, CBUSH or CBUSH1D element instead. **The CELAS element has somewhat been superceded by the CBUSH and CBUSH1D element to avoid problems with the generation of internal constraints.**

It is worth noting that the equivalent stiffness of springs in parallel is the addition of the individual stiffnesses and the equivalent stiffness of springs in series is the inverse of the addition of the inverse of the individual stiffnesses. The force in CELAS spring element is calculated as $F = K(u_1-u_2)$, and hence the sign is dependent upon how the element topology is defined, and not on the physical sense of element force, whether tension or compression. This is not the case in other elements such as CROD, CBAR and CBEAM. The output from the ELFORCE (PRINT) request is as follows.



F O R C E S I N S C A L A R S P R I N G S (C E L A S 2)

ELEMENT ID.	FORCE	ELEMENT ID.	FORCE	ELEMENT ID.	FORCE	ELEMENT ID.	FORCE
1	-6.366724E+02	2	3.633277E+02	3	-5.537512E+02	4	3.653757E+02

1.4.2 One-Dimensional Element Cards

1.4.2.1 Axial and Torsional Stiffness CROD, CONROD, CTUBE Elements

A CROD element connects two grid points straight and can provide axial and torsional stiffnesses, hence providing stiffness in only two DOFs at either grid. No bending nor shear stiffnesses are provided. Also, CROD elements do not consider torsional inertia in its mass matrix. NSM is the non-structural mass per unit length.

For **solid circular sections**, the torsional constant K_t be equal to the polar moment of inertia, $J = \int r^2 dA = \pi R^4/2$.

For **thin circular tube sections**, $K_t = J = \pi(r_1^4 - r_2^4)/2$.

For non-circular sections, the torsional constant is not be the polar moment of area. Equivalent torsional constants K_t for non-circular sections must be used. An approximation to the torsional constant of noncircular thin walled section depends on whether the **thin** beam section is closed or open. A **thin walled closed** section such as a SHS will have $K_t = 4A_e^2 / \left(\oint ds/t\right)$ where A_e is enclosed area by the mid thickness and the integral is carried out over the circumference. A **thin** walled **open** section such as an I-, T- or a channel should be treated as a series of rectangles each contributing to $K_t = (1/3)(ds)(t^3)$.

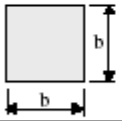
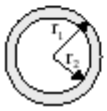
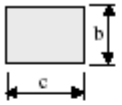
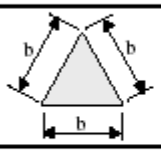
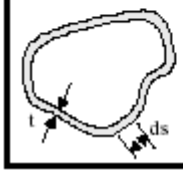
For **general rectangles**, $K_t = \frac{1}{3}\left[1 - 0.63\frac{b}{b_{max}}\left(1 - \frac{b^4}{12b_{max}^4}\right)\right]b^3 b_{max}$.

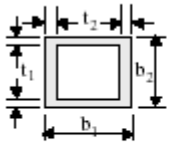
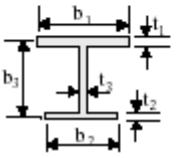
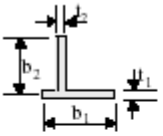
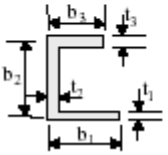
For **narrow rectangles** ($b_{max} >> 2b$), $K_t = \frac{1}{3}\left[1 - 0.63\frac{b}{b_{max}}\right]b^3 b_{max}$.

For **very narrow rectangles** ($b_{max} >> b$), $K_t = \frac{1}{3}b^3 b_{max}$.

For **squares**, $K_t = 0.1406b^4$.

For **equilateral solid triangles**, $K_t = \sqrt{3}b^4/80$

Section	Torsion constant
	$J = \frac{\pi r^4}{2}$
	$J = 0.1406 b^4$
	$J = \frac{\pi\left(r_1^4 - r_2^4\right)}{2}$
	$J = cb^3\left[\frac{1}{3} - 0.21\frac{b}{c}\left(1 - \frac{b^4}{12c^4}\right)\right]$
	$J = \frac{\sqrt{3}b^4}{80}$
	$J = \frac{4a^2}{\oint \frac{ds}{t}}$ where a is the area enclosed by a line through the centre of the thickness and the integral is carried out over the circumference.

Section	Torsion constant
	$J = \frac{2t_1 \cdot t_2 (b_1 - t_2)^2 \cdot (b_2 - t_1)^2}{b_1 t_2 + b_2 t_1 - t_2^2 - t_1^2}$
	$J = \frac{b_1 t_1^3 + b_2 t_2^3 + b_3 t_3^3}{3}$
	$J = \frac{b_1 t_1^3 + b_2 t_2^3}{3}$
	$J = \frac{1}{3}\sum b_i t_i^3$

For **general cross sections**, extruding a solid or shell model into a beam in an FE program and subjecting it to a torsion, the torsional constant K_t can be found. To do this, from $\tau/r = T/J = Gd\phi/dx$ it can be deduced that the rotation of a bar in **constant** (throughout the length of the beam) torsion T is $\phi = TL/GJ$ and replacing J with K_t, the torsional constant K_t can be found. Ensure that there is no other deformation (shear, bending, axial) which contribute to the total deformation, otherwise they must be removed. A suitable system will simply be a encastre cantilever (which is thus statically determinate as far as the equivalent beam is concerned) subject to a torsion at the free end. Hence model the complicated model with one end fixed in all DOFs and the other subject to a torsion T. Of course there will be many forms of deformation (be they axial, bending, shear or torsion) in the local elements of the complicated model, but the equivalent beam idealization will only be subject to a constant torsion T. Hence the resulting twist at the free end ϕ will be the deformation that will be observed in the idealized beam model due to only torsional deformation and $\phi = TL/GJ$ can be used to find J (or K_t).

$ BULK DATA									
CROD	**EID**	**PID**	**GA**	**GB**					
PROD	**PID**	**MID**	**Area**	**J**	**C**	**NSM**			

The CONROD element defines the axial and torsional stiffness in the connection card itself.

$ BULK DATA									
CONROD	**EID**	**GA**	**GB**	**MID**	**Area**	**J**	**C**	**NSM**	

The CTUBE element defines the axial and torsional stiffnesses by specifying the outer diameter and the thickness of a circular tube with a PTUBE entry. The element coordinate system is defined by the direction of the two grids.

A positive force is tensile by definition and the positive torsion is represented by torsion in a direction according to the Right Hand Rotation Rule about the longitudinal axes directed outwards. These elements are useful to define truss elements where there is no bending. The output from the ELFORCE (PRINT) request is as follows.

F O R C E S I N R O D E L E M E N T S (C R O D)

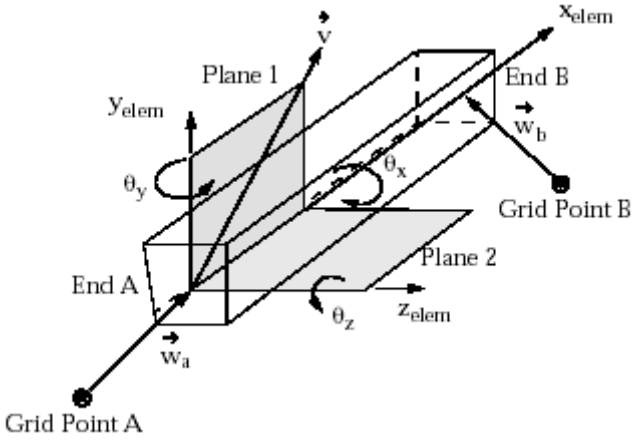
ELEMENT ID.	AXIAL FORCE	TORQUE	ELEMENT ID.	AXIAL FORCE	TORQUE
1	2.495156E+03	0	2	2.554449E+03	.0
3	-2.544570E+03	.0	4	-2.504844E+03	.0
5	4.990313E+02	.0			

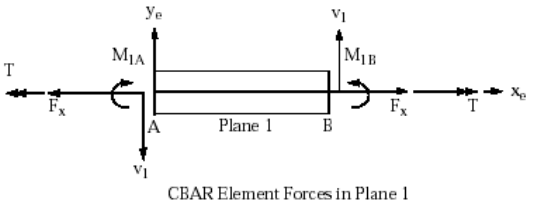
1.4.2.2 CBAR Element

A CBAR element is a straight prismatic element and provides axial, torsional, bending stiffness in two perpendicular planes and shear stiffness in two perpendicular planes, hence providing stiffness in all six DOFs on either grid. Note that CBAR elements do not consider torsional inertias. It is derived from classical beam theory (which assumes that plane sections remain plane in deformation). If this is not deemed to be the case, shell or solids should be used. Since, plane sections must remain plane in deformation, torsional stiffening of out of plane cross-sectional warping is neglected. Also, since the shear center and neutral axis must coincide, the CBAR element cannot model warping of open sections. NSM is the non-structural mass per unit length.

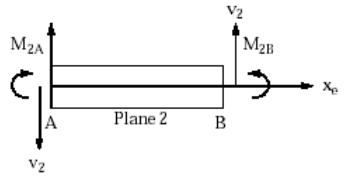
The CBAR element provides exact results for end loads of any kind, i.e. no element loading. Hence, it models linear bending moment variation exactly. Thus its transverse displacement interpolation function is a cubic polynomial since the bending moment is the second derivative of the transverse displacement in linear beam bending theory. Hence, if there are internal point loads in a structural beam member, discretized the member with a number of finite CBAR elements so that at least all the loads occur at the ends of the CBAR elements. This will prove to yield exact results for a static analysis since the displaced shape for sections with a linear variation of bending moment is cubic polynomial. For modal analyses, it is prudent to use a finer mesh as the mode shapes are not cubic polynomials, in fact are not polynomials at all.

The element coordinate system is defined by either an orientation vector or an orientation node, both of which defines a plane *in which* bending with the second moment of area of I1 occurs. Hence, if I1 is the major principal second moment of area, the orientation node will define the minor axis of bending. The element coordinate system is then defined as shown.





CBAR Element Forces in Plane 1



CBAR Element Forces in Plane 2

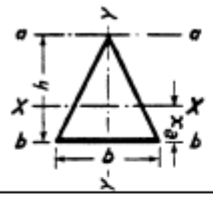
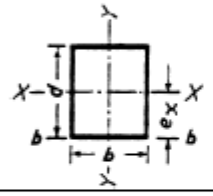
Some of the connections of the bar element can be removed by pin-flags. Pin flags can be specified for translational degrees-of-freedom (creating sliding joints), or to rotational degrees-of-freedom (creating hinges). Note that the joints created by the pin-flags occur at the ends of the CBAR' s neutral axis, not at the grid points. This is akin to disconnecting the bar element from its supporting structure and connecting only certain DOFs. Hence, rotational pin-flags is akin to having two coincident nodes at the end of the bar element (one attached to the bar and the other to the supporting structure) with only the translational DOFs constrained between them.

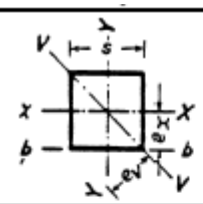
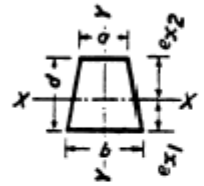
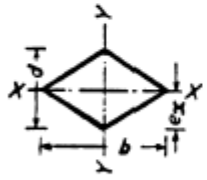
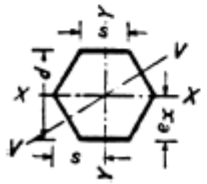
The neutral axis may be offset from the grid point, which is akin to attaching a rigid link between the end of the bar to the support. This is useful to model stiffened plates or grid-works.

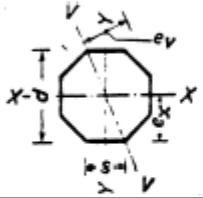
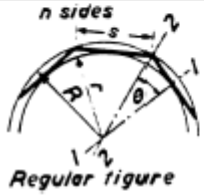
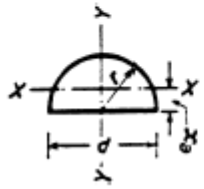
$ BULK DATA									
CBAR	**EID**	**PID**	**GA**	**GB**	**X1**	**X2**	**X3**		
	PA	**PB**	**W1A**	**W2A**	**W3A**	**W1B**	**W2B**	**W3B**	
CBAR	**EID**	**PID**	**GA**	**GB**	**G0**				

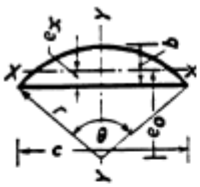
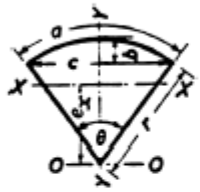
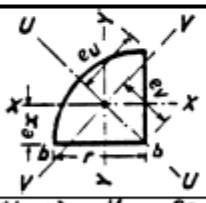
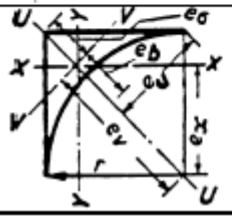
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
PBAR	PID	MID	Area	I1	I2	J	NSM		
	C1	C2	D1	D2	E1	E2	F1	F2	
	K1	K2	I12						

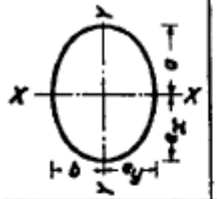
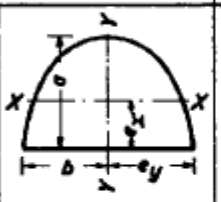
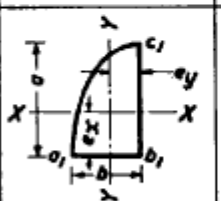
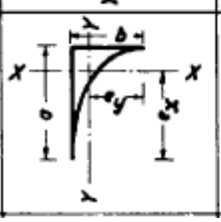
X1, X2 and X3 define the orientation vector components in the displacement coordinate system of grid G1. The alternate is to define an orientation grid id G0, but this method is not recommended as alterations to the model will cause the definition to defer. PA and PB are the pin-flags. The Wij are the components of the offset vectors in the displacement coordinate system of G1 and G2. The offset is effectively a rigid element from the grid to the element end. If the CBAR is offset from the grid points and the components of orientation vector are entered (i.e. field 6 of the CBAR entry is a real number), then the tail of orientation vector is at end A, not grid point GA. Conversely, if the orientation vector is defined with the use of another grid point G0 (field 6 of the CBAR entry is an integer number), then the orientation vector is defined as the line originating at grid point GA, not end A, and passing through G0. The Ci, Di, Ei and Fi are the stress recovery coefficients for stress estimation using bending $\sigma = My/I$ and from the axial forces. Non-symmetric beams will have I12 component. In a beam with a symmetric section, the bending properties depend only in I1 and I2 since they correspond to the principal axes. A load in plane 1 or plane 2 will cause a deflection in only plane 1 or 2. When the section is not symmetric, and is loaded in either plane 1 or plane 2, there is a component of deflection orthogonal to the loading. This occurs because the loading is not in the principal planes. In this case, the bending properties depend on I1, I2 and I12. If in this non-symmetric section, the principal planes were found (by Mohr Circle since I is a second order tensor) and the loading was applied in one of these principal planes, then there will be no orthogonal component to the deflection.

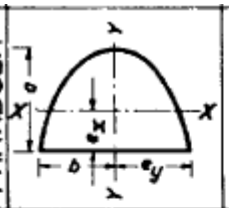
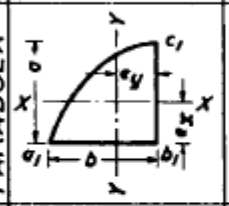
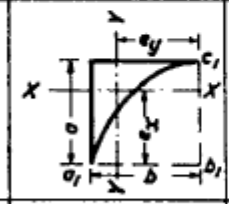
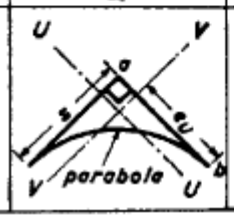
section		Area	Position of centroid	Moments of inertia	Section Moduli
Triangle		$A=\frac{bh}{2}$	$e_x=\frac{h}{3}$	$I_{xx}=bh^3/36$ $I_{yy}=hb^3/48$ $I_{aa}=bh^3/4$ $I_{bb}=bh^3/12$	Z_{xx} $base=bh^2/12$ $apex=bh^2/24$ $Z_{yy}=bh^2/24$
Rectangle		$A=bd$	$e_x=\frac{h}{2}$	$I_{xx}=bd^3/12$ $I_{yy}=db^3/12$ $I_{bb}=bd^3/3$	$Z_{xx}=bd^2/6$ $Z_{yy}=db^2/6$
Rectangle		$A=bd$	$e_x=\frac{bd}{\sqrt{b^2+d^2}}$	$I_{xx}=\frac{b^3d^3}{6(b^2+d^2)}$	$Z_{xx}=\frac{b^2d^2}{6\sqrt{b^2+d^2}}$
Rectangle		$A=bd$	$e_x=\frac{b\sin\theta+d\cos\theta}{2}$	$I_{xx}=\frac{bd(b^2\sin^2\theta+d^2\cos^2\theta)}{12}$	$Z_{xx}=\frac{bd(b^2\sin^2\theta+d^2\cos^2\theta)}{6(b\sin\theta+d\cos\theta)}$

Square		$A = s^2$	$e_x = \frac{s}{2}$ $e_v = \frac{s}{\sqrt{2}}$	$I_{xx} = I_{yy} = s^4/12$ $I_{bb} = s^4/3$ $I_{vv} = s^4/12$	$Z_{xx} = Z_{yy} = \frac{s^3}{6}$ $Z_{vv} = \frac{s^3}{6\sqrt{2}}$
Trapezium		$A = \frac{d(a+b)}{2}$	$e_{x1} = \frac{d(2a+b)}{3(a+b)}$	$I_{xx} = \frac{d^3(a^2+4ab+b^2)}{36(a+b)}$ $I_{yy} = \frac{d(a^3+a^2b+ab^2+b^3)}{48}$	$Z_{xx} = \frac{I_{xx}}{d-e_x}$ (*two values*) $Z_{yy} = \frac{2I_{yy}}{b}$
Diamond		$A = \frac{bd}{2}$	$e_x = \frac{d}{2}$	$I_{xx} = \frac{bd^3}{48}$ $I_{yy} = \frac{db^3}{48}$	$Z_{xx} = \frac{bd^2}{24}$ $Z_{yy} = \frac{db^2}{24}$
Hexagon		$A = 0.866d^2$	$e_x = 0.866s$ $= \frac{d}{2}$	$I_{xx} = I_{yy} = I_{vv}$ $= 0.0601d^4$	$Z_{xx} = 0.1203d^3$ $Z_{yy} = Z_{vv}$ $= 0.1042d^3$

Section		Area	Position of centroid	Moments of inertia	Section Moduli
Octagon		$A = 0.8284d^2$ $s = 0.4142d$	$e_x = \frac{d}{2}$ $e_v = 0.541d$	$I_{xx} = I_{yy} = I_{vv}$ $= 0.0547d^4$	$Z_{xx} = Z_{yy}$ $= 0.1095d^3$ $Z_{vv} = 0.1011d^3$
Polygon		$A = \frac{ns^2\cot\theta}{4}$ $A = nr^2\tan\theta$ $A = \frac{nR^2\sin 2\theta}{2}$	*e = r or R depending on the axis and value of n*	$I_1 = I_2$ $= \frac{A(6R^2-s^2)}{24}$ $= \frac{A(12r^2+s^2)}{48}$	$Z = \frac{I}{e}$
Circle		$A = \pi r^2$ $A = 0.7854d^2$	$e = r = \frac{d}{2}$	$I = \frac{\pi d^4}{64}$ $I = 0.7854r^2$	$Z = \frac{\pi d^3}{32}$ $Z = 0.7854r^3$
Semi-Circle		$A = 1.5708r^2$	$e_x = 0.424r$	$I_{xx} = 0.1098r^4$ $I_{yy} = 0.3927r^4$	Z_{xx} = *base* $= 0.2587r^3$ *crown* $= 0.1907r^3$ $Z_{yy} = 0.3927r^3$

Segment		$A = \frac{r^2}{2}\left(\frac{\pi\theta^\circ}{180^\circ} - \sin\theta\right)$	$e_0 = \frac{c^3}{12A}$ $e_x = e_0 - r\cos\frac{\theta}{2}$	$I_{xx} = \frac{r^4}{16}\left(\frac{\pi\theta^\circ}{90^\circ} - \sin 2\theta\right) - \frac{20r^4(1-\cos\theta)^3}{\pi\theta^\circ - 180^\circ\sin\theta}$ $I_{yy} = \frac{r^4}{48}\left(\frac{\pi\theta^\circ}{30^\circ} - 8\sin\theta + \sin 2\theta\right)$	Z_{xx} $base = I_{xx}/e_1$ $crown = \frac{I_{xx}}{b - e_1}$ $Z_{yy} = \frac{2I_{yy}}{c}$
Sector		$A = \frac{\theta^\circ}{360^\circ}\pi r^2$	$e_x = \frac{2}{3}r\frac{c}{a}$ $e_x = \frac{r^2 c}{3A}$	$I_{xx} = I_o - \frac{360^\circ}{\theta^\circ\pi}\sin^2\frac{\theta}{2}\frac{4r^4}{9}$ $I_{yy} = \frac{r^4}{8}\left(\frac{\pi\theta^\circ}{180^\circ} - \sin\theta\right)$ $I_0 = \frac{r^4}{8}\left(\frac{\pi\theta^\circ}{180^\circ} + \sin\theta\right)$	Z_{xx} $centre = \frac{I_{xx}}{e_x}$ $crown = \frac{I_{xx}}{r - e_x}$ $Z_{yy} = \frac{2I_{yy}}{c}$
Quadrant		$A = \frac{\pi r^2}{4}$	$e_x = 0.424r$ $e_v = 0.6r$ $e_u = 0.707r$	$I_{xx} = I_{yy} = 0.0549r^4$ $I_{bb} = 0.1963r^4$ $I_{uu} = 0.0714r^4$ $I_{vv} = 0.0384r^4$	*Minimum Values* $Z_{xx} = Z_{yy} = 0.0953r^3$ $Z_{uu} = 0.1009r^3$ $Z_{vv} = 0.064r^3$
Complement		$A = 0.2146r^2$	$e_x = 0.777r$ $e_v = 1.098r$ $e_a = 0.707r$ $e_\sigma = 0.316r$ $e_b = 0.391r$	$I_{xx} = I_{yy} = 0.0076r^4$ $I_{uu} = 0.012r^4$ $I_{vv} = 0.0031r^4$	*Minimum Values* $Z_{xx} = Z_{yy} = 0.0097r^3$ $Z_{uu} = 0.017r^3$ $Z_{vv} = 0.0079r^3$

	Section	Area	Position of Centroid	Moments of Inertia	Section Moduli
ELLIPSE		$A = \pi ab$	$e_x = a$ $e_y = b$	$I_{XX} = 0.7854ba^3$ $I_{YY} = 0.7854ab^3$	$Z_{XX} = 0.7854ba^2$ $Z_{YY} = 0.7854ab^2$
SEMI-ELLIPSE		$A = \frac{\pi ab}{2}$	$e_x = 0.424a$ $e_y = b$	$I_{XX} = 0.1098ba^3$ $I_{YY} = 0.3927ab^3$ $I_{base} = 0.3927ba^3$	Z_{XX} - base $= 0.2587ba^2$ Z_{XX} - crown $= 0.1907ba^2$ $Z_{YY} = 0.3927ab^2$
1/4 ELLIPSE		$A = 0.7854ab$	$e_x = 0.424a$ $e_y = 0.424b$	$I_{XX} = 0.0549ba^3$ $I_{YY} = 0.0549ab^3$ $I_{b_1 a_1} = 0.1963ba^3$ $I_{b_1 c_1} = 0.1963ab^3$	Z_{XX} - base $= 0.1293ba^2$ Z_{XX} - crown $= 0.0953ba^2$ Z_{YY} - base $= 0.1293ab^2$ Z_{YY} - crown $= 0.0953ab^2$
COMPLEMENT		$A = 0.2146ab$	$e_x = 0.777a$ $e_y = 0.777b$	$I_{XX} = 0.0076ba^3$ $I_{YY} = 0.0076ab^3$	Z_{XX} - base $= 0.0338ba^2$ Z_{XX} - apex $= 0.0097ba^2$ Z_{YY} - base $= 0.0338ab^2$ Z_{YY} - apex $= 0.0097ab^2$

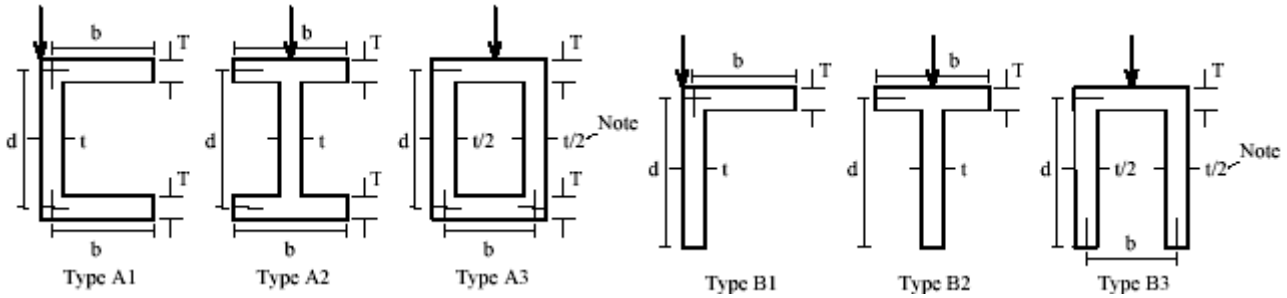
FULL PARABOLA		$A = \frac{4ab}{3}$	$e_x = \frac{2a}{5}$ $e_y = b$	$I_{XX} = 0.0914ba^3$ $I_{YY} = 0.2666ab^3$ $I_{base} = 0.3048ba^3$	Z_{XX} - base $= 0.2286ba^2$ Z_{XX} - crown $= 0.1524ba^2$ $Z_{YY} = 0.2666ab^2$
SEMI-PARABOLA		$A = \frac{2ab}{3}$	$e_x = \frac{2a}{5}$ $e_y = \frac{3b}{8}$	$I_{XX} = 0.0457ba^3$ $I_{YY} = 0.0396ab^3$ $I_{b_1a_1} = 0.1524ba^3$ $I_{b_1c_1} = 0.1333ab^3$	Z_{XX} - base $= 0.1143ba^2$ Z_{XX} - crown $= 0.076ba^2$ Z_{YY} - base $= 0.1055ab^2$ Z_{YY} - crown $= 0.0633ab^2$
COMPLEMENT		$A = \frac{ab}{3}$	$e_x = \frac{7a}{10}$ $e_y = \frac{3b}{4}$	$I_{XX} = 0.0176ba^3$ $I_{YY} = 0.0125ab^3$ $I_{a_1b_1} = 0.181ba^3$ $I_{b_1c_1} = 0.2ab^3$	Z_{XX} - base $= 0.0587ba^2$ Z_{XX} - apex $= 0.0252ba^2$ Z_{YY} - base $= 0.05ab^2$ Z_{YY} - apex $= 0.0167ab^2$
FILLET		$A = \frac{s^2}{6}$	$e_U = e_V = \frac{4s}{5}$	$I_{UU} = I_{VV} = 0.00524s^4$ $I_{ab} = 0.1119a^4$	$Z_{UU} = Z_{VV}$ base $= 0.0262a^3$ apex $= 0.0066a^3$

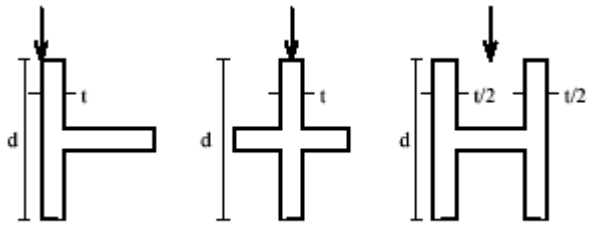
Ki are the area factors for shear deformation. Shear stress on a beam is given by $\tau = VQ/Ib$. A simplified expression would simply be $\tau = V/A_s$ where the shear area A_s is obtained by multiplying the shear area factor K_s to the total area. The shear area factors Ki are as follows.

Solid Rectangular	K1 = K2 = 5/6
Solid Circular	$K1 = K2 = \frac{6}{7 + \frac{\nu^2}{(1+\nu)^2}}$
CHS	K1 = K2 = 1/2
I- and T- Beams	
Major Axis	K1 = (5.72/6 to 5.81/6) (A_{web} / A)
Minor Axis	K2 = (5/6) (A_{flange} / A)

For those rare structures where the shear deformation is very important it may be necessary to use a more exact value for the area. The following gives formulae for *F* for the cross-sections shown below noting that

shear deformation area = $Fdt/6$





Type C

Bending in strong direction	
UB	$F = 5.72$ to 5.81
Joist	$F = 5.17$ to 5.78
UBP	$F = 5.25$ to 5.28
UC	$F = 5.28$ to 5.53
SHS	$F = 5.0$
RHS	$F = 5.0$ to 5.49 (for $d/b = 1$ to 2)
Channel	$F = 5.06$ to 5.60
Angle	$F = 3.68$ to 4.62 (for $d/b = 1$ to 2)
Tee from UB	$F = 4.78$ to 4.97
Tee from UC	$F = 4.11$ to 4.35
Cruciform	$F = 5$
Bending in weak direction	
Channel	$F = 2.12$ to 3.78
RHS	$F = 4.02$ to 5.0 (for $d/b = 0.5$ to 1)
Angle	$F = 2.55$ to 3.68 (for $d/b = 0.5$ to 1)
I's, T's, & cruciforms	$F = 5.0$

Special Case of B3 with constant thickness $T = t/2$, $\lambda = 2\phi$, $\beta = 2/\phi$ gives:

$$F = \frac{20\phi(2+\phi)^2}{5+16\phi+14\phi^2+4\phi^3}$$

Type C

$$F = 5$$

$$\lambda = \frac{dt}{bT} \qquad \beta = \frac{bt}{dT} \qquad \phi = \frac{d}{b}$$

Shear Deformation Area = $F\dfrac{dt}{6}$

Type A1

$$F = \frac{(6+\lambda)^2}{4\beta+6+2\lambda+0.2\lambda^2}$$

Types A2 & A3

$$F = \frac{(6+\lambda)^2}{\beta+6+2\lambda+0.2\lambda^2}$$

Special case of A3 with constant wall thickness so that $T = t$, $\lambda = 2\phi$, $\beta = 2/\phi$ gives:

$$F = \frac{10\phi(3+\phi)^2}{5+15\phi+10\phi^2+2\phi^3}$$

Type B1

$$F = \frac{(4+\lambda)^2}{2\beta+3.2+1.4\lambda+0.2\lambda^2}$$

Special Case of B1 with $T = t$, $\lambda = \phi$, $\beta = 1/\phi$ gives:

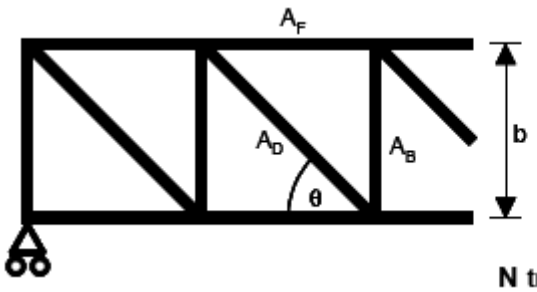
$$F = \frac{5\phi(4+\phi)^2}{(1+\phi)(10+6\phi+\phi^2)}$$

Types B2 & B3

$$F = \frac{(4+\lambda)^2}{0.5\beta+3.2+1.4\lambda+0.2\lambda^2}$$

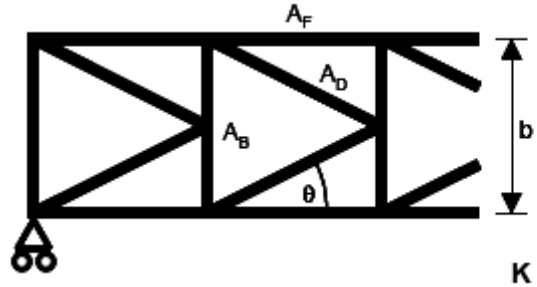
The shear stiffness of a castellated beam, or of a truss modelled by a single beam element, is significantly less than for the equivalent solid beam and so shear deformations always need to be allowed for, even with slender beams. It is often convenient in a large analysis to represent trusses by beams with shear flexibility. The axial stiffnesses, EA, of the chords of a truss determine the bending stiffness, whereas the axial stiffnesses of bracing members, or the flexural stiffnesses, EI, of all members, control the shear stiffness. For all trusses and frames with parallel chords of area A_F and centroids separated by b, the **flexural stiffness $EI = EA_F.b^2/2$**. The **shear stiffnesses** and shear area factors for trusses and for Vierendeel frames are presented.

Beam Type	Bending stiffness	Shear area
Castellated Beams It=I for T section at hole s = pitch of holes	(Isolid + I at hole)/2	$\frac{E/G \times 192 \times It}{s^2}$
Vierendeel girders where elements are slender (element span/depth > 10) A = pitch of bracing members b = depth between flanges If = I for flange member Ib = I for bracing member	(I for top and bottom flanges)	$\frac{E/G}{\frac{a^2}{24 \times If} + \frac{a \times b}{12 \times Ib}}$
Vierendeel girders where elements are stocky (element span/depth < 10) Af = area of flange Kf = shear factor for flange Ab = area for bracing member Kb = shear factor for bracing member	(I for top and bottom flanges)	$\frac{E/G}{\frac{a^2}{24 \times If} + \frac{a \times b}{12 \times Ib} + \frac{E/G}{2 \times Af \times Kf} + \frac{E/G}{2 \times Ab \times Kb}}$
N-truss (pin jointed) θ = angle of diagonal to horizontal Ad = area of diagonal Av = area of vertical	(I for top and bottom flanges)	$\frac{E/G \times \cos\theta}{\frac{1}{Ad \times \sin^2\theta} + \frac{\sin\theta}{Av}}$
Warren Truss (pin jointed) θ = angle of diagonal to horizontal Adc = area of compression diagonal Adt = area of tension diagonal	(I for top and bottom flanges)	$\frac{E/G \times 2 \times \sin^2\theta \times \cos\theta}{\frac{1}{Adc} + \frac{1}{Adt}}$



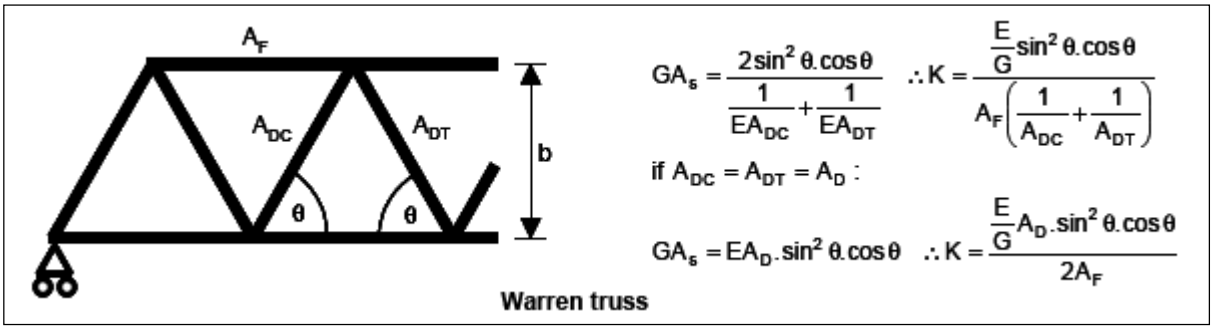
$$GA_s = \frac{\sin^2\theta.\cos\theta}{\frac{1}{EA_D} + \frac{\sin^3\theta}{EA_B}} \quad \therefore K = \frac{\frac{E}{G}\sin^2\theta.\cos\theta}{2A_F\left(\frac{1}{A_D} + \frac{\sin^3\theta}{A_B}\right)}$$

N truss



$$GA_s = \frac{2\sin^2\theta.\cos\theta}{\frac{1}{EA_D} + \frac{\sin^3\theta}{EA_B}} \quad \therefore K = \frac{\frac{E}{G}\sin^2\theta.\cos\theta}{A_F\left(\frac{1}{EA_D} + \frac{\sin^3\theta}{EA_B}\right)}$$

K truss

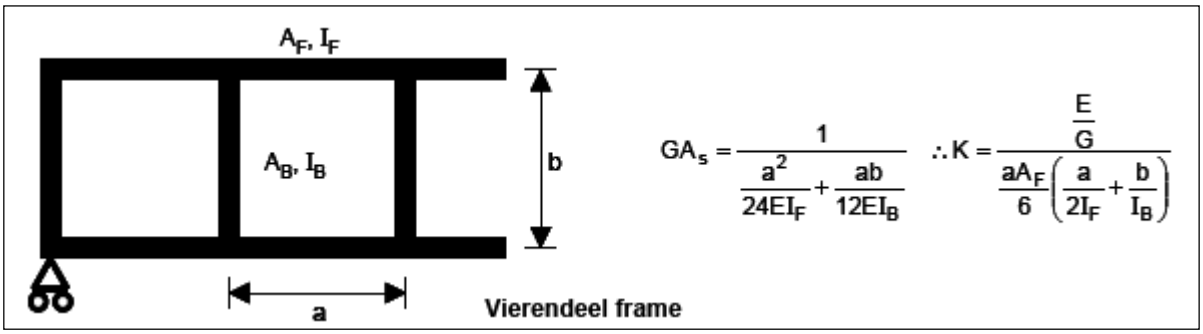


$$GA_s = \frac{2\sin^2\theta.\cos\theta}{\frac{1}{EA_{DC}} + \frac{1}{EA_{DT}}} \quad \therefore K = \frac{\frac{E}{G}\sin^2\theta.\cos\theta}{A_F\left(\frac{1}{A_{DC}} + \frac{1}{A_{DT}}\right)}$$

if $A_{DC} = A_{DT} = A_D$:

$$GA_s = EA_D.\sin^2\theta.\cos\theta \quad \therefore K = \frac{\frac{E}{G}A_D.\sin^2\theta.\cos\theta}{2A_F}$$

Warren truss



$$GA_s = \frac{1}{\frac{a^2}{24EI_F} + \frac{ab}{12EI_B}} \quad \therefore K = \frac{\frac{E}{G}}{\frac{aA_F}{6}\left(\frac{a}{2I_F} + \frac{b}{I_B}\right)}$$

Vierendeel frame

If the shear stiffnesses of the members and joints are also considered, then the expressions for Vierendeel frames becomes

$$GA_s = \frac{1}{\frac{a^2}{24EI_F} + \frac{ab}{12EI_B} + \frac{1-w/a}{2GK_FA_F} + \frac{a(1-h/b)}{bGK_BA_B} + \frac{a(1-w/a-h/b)}{2S}}$$

where S is the rotational stiffness of the joint, and K_F and K_B are the shear area factors associated with A_F and A_B respectively and w and h are defined on the joint figure below.

$$\therefore K = \frac{1}{\frac{aGA_F}{6E}\left(\frac{a}{2I_F} + \frac{b}{I_B}\right) + \frac{1-w/a}{K_F} + \frac{2a(1-h/b)A_F}{bK_BA_B} + \frac{a(1-w/a-h/b)GA_F}{S}}$$

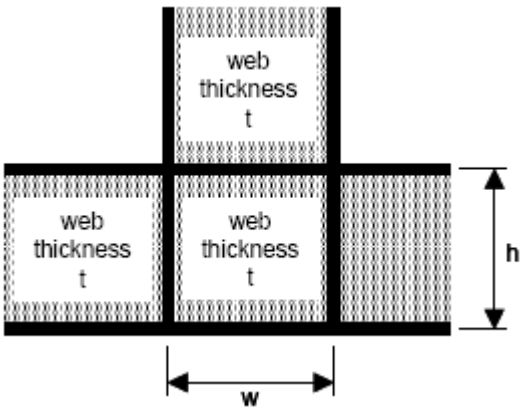
For a Vierendeel frame comprising flanged sections with a common web thickness, much of the shear flexibility results from joint deformation. For the joint shown, S, K_FA_F and K_BA_B are given by:

$$S = \frac{Gwht}{1-\frac{w}{a}-\frac{h}{b}}$$

$$K_FA_F = ht$$

$$K_BA_B = wt$$

$$\therefore K = \frac{1}{\frac{aGA_F}{6E}\left(\frac{a}{2I_F} + \frac{b}{I_B}\right) + \frac{aA_F}{wht}\left(1-\frac{w}{a}+\frac{2wh}{ab}-\frac{h^2}{b^2}\right)}$$



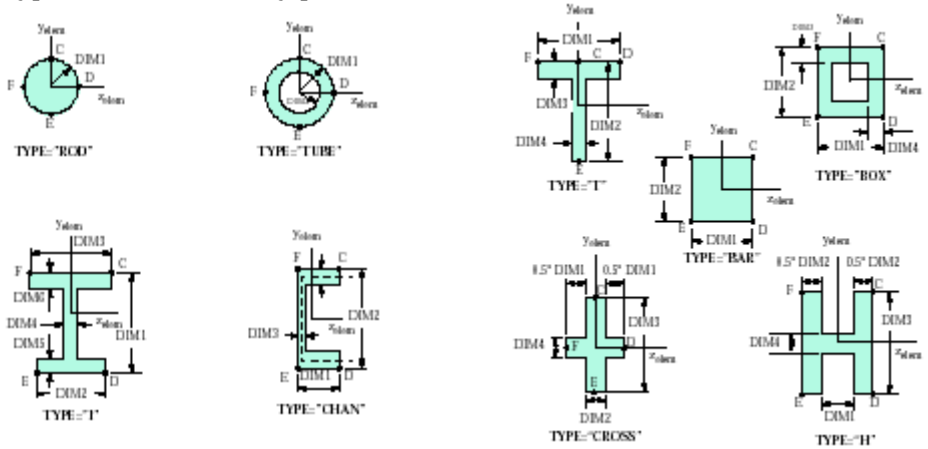
It should be noted that although a frame analysis would not normally include joint deformations, it would include the shear deformations between the face of the perpendicular members and their centre lines. If the above value of K is rewritten as

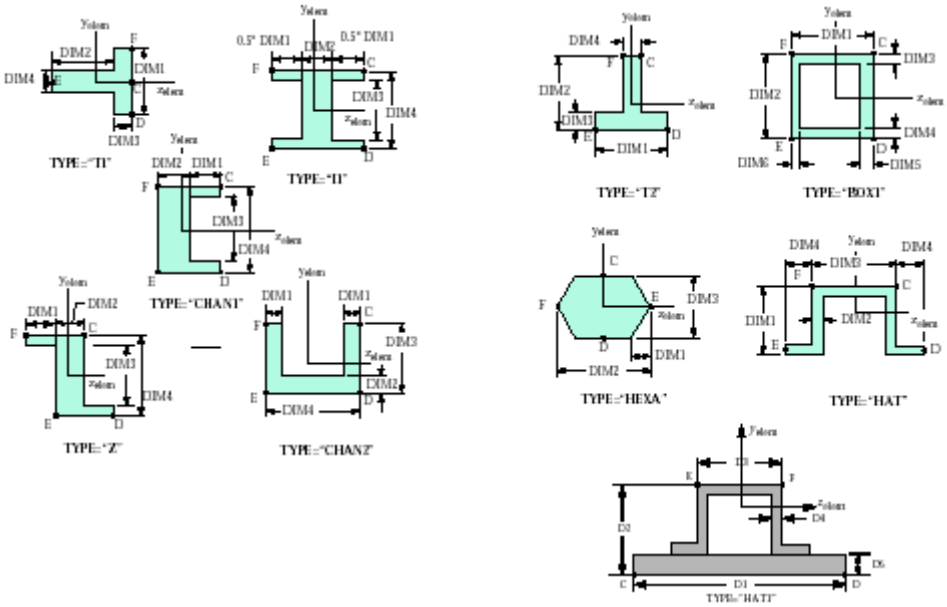
$$K = \frac{1}{\frac{aGA_F}{6E}\left(\frac{a}{2I_F}+\frac{b}{I_B}\right)+\frac{A_F}{ht}+\frac{2aA_F}{bwt}+\frac{a[2(1-w/a)(1-h/b)-1-(h/b)^2]A_F}{wht}}$$

Whether such a model over or under estimates the shear deformations depends on the last term of the denominator and hence the ratios w/a and h/b.

For **general cross sections**, extruding a solid or shell model into a beam in an FE program and subjecting it to a shear force, the shear area can be found. To do this, from $\tau = V/A_s = Gdu/dx$ it can be deduced that the shear deflection of a beam in **constant** (throughout the length of the beam) shear V is $u = VL/(GA_s)$, and hence the shear area A_s can be found. Ensure that there is no other deformation (bending, axial, torsion) which contribute to the total deformation, otherwise they must be removed. A suitable system will simply be a encastre cantilever (which is thus statically determinate as far as the equivalent beam is concerned) subject to a concentrated force at the free end. Hence model the complicated model with one end fixed in all DOFs and the other subject to a force V. Of course there will be many forms of deformation (be they axial, bending, shear or torsion) in the local elements of the complicated model, but the equivalent beam idealization will only be subject to a constant shear force V and a linearly varying bending moment. Hence the resulting deflection at the free end will be the deformation that will be observed in the idealized beam model due to both shear and bending deformations, the cantilever will not be subject to any axial nor torsional forces. The component of the deflection which is due to bending deformation must be known. This must be calculated from classical methods, i.e. $u_{bending} = VL^3/(3EI)$ where I is calculated from a closed form formula. The remaining deflection after $u_{bending}$ is subtracted is the shear deflection which can then be used to back calculate the shear area A_s from $u = VL/(GA_s)$.

The **PBARL** is an alternative method of defining the PBAR entry. The PBARL Bulk Data entry allows you to input cross section types along with their characteristic dimensions. You can choose from19 different cross section shapes. These shapes are as follows: ROD, TUBE, I, CHAN (channel), T, BOX, BAR (rectangle), CROSS, H, T1, I1, CHAN1, Z, CHAN2, T2, BOX1, HEXA (hexagon), HAT (hat section), and HAT1. Because of the knowledge of the section type, the stress recovery points are automatically defined.





$ BULK DATA									
PBARL	**PID**	**MID**	**Group (MSCBML0)**	**TYPE**					
	DIM1	**DIM2**	**DIM3**	**DIM4**	**DIM5**	**DIM6**	**DIM7**	**DIM8**	
	DIM9	**etc..**	**NSM**						

The output from the ELFORCE (PRINT) request is as follows.

F O R C E S I N B A R E L E M E N T S (C B A R)

ELEMENT ID.	BEND-MOMENT END-A PLANE 1	BEND-MOMENT END-A PLANE 2	BEND-MOMENT END-B PLANE 1	BEND-MOMENT END-B PLANE 2	- SHEAR - PLANE 1	- SHEAR - PLANE 2	AXIAL FORCE	TORQUE
1	2.586560E+01	-1.519694E+01	1.619619E+00	4.542250E+00	2.168636E-02	-1.765535E-02	3.726698E+03	2.810185E+00
2	2.586560E+01	1.519694E+01	1.619619E+00	-4.542250E+00	2.168636E-02	1.765535E-02	3.726698E+03	-2.810185E+00
3	-5.172854E+01	.0	-3.240443E+00	.0	-4.336907E-02	.0	-7.453428E+03	.0

1.4.2.3 CBEAM Element

A CBEAM element includes the CBAR element capabilities and separate neutral axis and axis of shear centers, effect of cross-sectional warping on torsional stiffness (as is important in the case of open sections), cross-sectional properties specified on both ends and interior point (tapered element), effect of taper on traverse shear stiffness (shear relief), separate axis for the center of non-structural mass and torsional inertias are included i.e. can be offset from shear center (for dynamic analyses) and also has nonzero rotational mass moment of inertia about its neutral axis. The CBEAM element can be loaded at interior points by means of the PLOAD1 card. But again, the CBEAM element is only exact if loads are applied at its ends, i.e. it models a linear bending moment distribution. NSM is the non-structural mass per unit length. The element x-axis is along the shear center unlike CBAR where it is along the neutral axis.

$ BULK DATA									
CBEAM	**EID**	**PID**	**GA**	**GB**	**X1**	**X2**	**X3**		
	PA	**PB**	**W1A**	**W2A**	**W3A**	**W1B**	**W2B**	**W3B**	
	SA	**SB**							
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)**	
	K1	**K2**	**I12**						
	…	…	…	…	…	…	…	…	
	S0	**X/XB**	**A**	**I1**	**I2**	**I12**	**J**	**NSM**	
	C1	**C2**	**D1**	**D2**	**E1**	**E2**	**F1**	**F2**	
	…	…	…	…	…	…	…	…	
	K1	**K2**	**S1**	**S2**	**NSI(A)**	**NSI(B)**	**CW(A)**	**CW(B)**	
	M1(A)	**M2(A)**	**M1(B)**	**M2(B)**	**N1(A)**	**N2(A)**	**N1(B)**	**N2(B)**	

For the CBAR element, the element stiffness matrix is generated directly from the input data. For instance, if I1 for a CBAR element is zero, then the corresponding element stiffness matrix term is null, which is not necessarily a problem. On the other hand, the CBEAM element uses the input data to generate an element flexibility matrix, which must be inverted to produce the element stiffness matrix. Therefore, positive values for A, I1, and I2 must be entered.

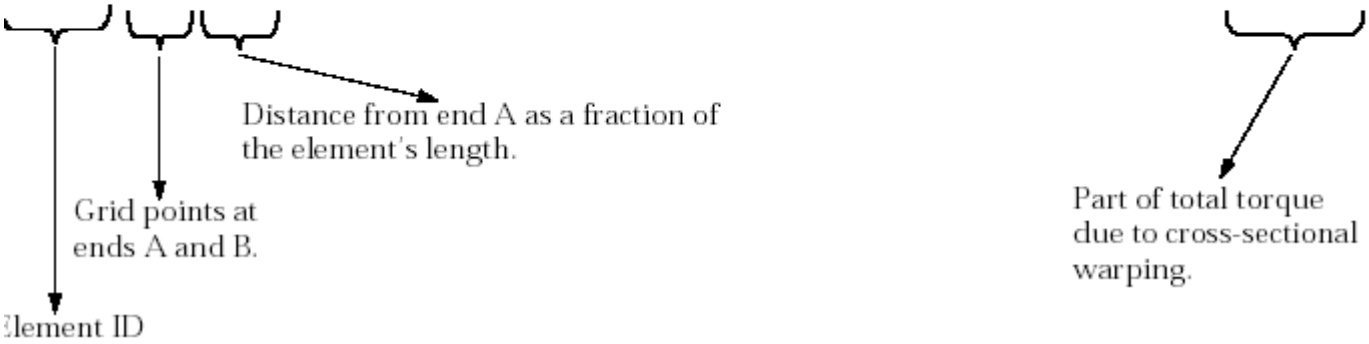
One difference between the CBAR element and the CBEAM element, that is not obvious, is the default values used for the transverse shear flexibility. For the CBAR element, the default values for K1 and K2 are infinite, which is equivalent to zero transverse shear flexibility. For the CBEAM element, the default values for K1 and K2 are both 1.0, which includes the effect of transverse shear in the elements. If you want to set the transverse shear flexibility to zero, which is the same as the CBAR element, use a value of 0.0 for K1 and K2.

Analogous to PBARL, PBEAML can be used to generate the section properties from the dimensions.

The output from the ELFORCE (PRINT) request is as follows.

F O R C E S I N B E A M E L E M E N T S (C B E A M)

ELEMENT-ID	GRID	STAT DIST/ LENGTH	- BENDING MOMENTS - PLANE 1	PLANE 2	- WEB SHEARS - PLANE 1	PLANE 2	AXIAL FORCE	TOTAL TORQUE	WARPING TORQUE
1									
	1	.000	8.653494E+03	-9.295851E+03	3.013959E+02	-1.985699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	0	.200	6.791337E+03	-7.331033E+03	2.765989E+02	-1.985699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	0	.400	4.929180E+03	-5.366216E+03	2.518019E+02	-1.985699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	0	.600	3.067023E+03	-3.401398E+03	2.270049E+02	-1.985699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	0	.800	1.204866E+03	-1.436580E+03	2.022079E+02	-1.985699E+02	9.060291E+01	-7.902837E+01	9.954489E-17
	2	1.000	-6.572902E+02	5.282371E+02	1.774109E+02	-1.985699E+02	9.060291E+01	-7.902837E+01	9.954489E-17



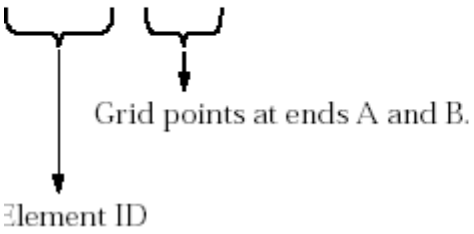
1.4.2.4 CBEND Element

A CBEND element is a circular arc between 2 grids with extensional, torsional, bending stiffness in two perpendicular planes and shear stiffness in two perpendicular planes. The PBEND card has two alternate forms, corresponding to a curved beam of arbitrary cross-section or a curved pipe (tube). The curved beam has its principal bending axes parallel and perpendicular to the plane of the element, the geometric centre of the element may be offset in two perpendicular directions, the offset of the neutral axis from the geometric centre due to curvature is calculated automatically. For the curved beam form of the element a user override is possible. For the curved pipe form, the effect of internal pressure on stiffness and stress can be accounted for. Axial stresses can be output at four cross-sectional points at each end of the element. Forces and moments are output at both ends. Distributed loads may be applied along the element using the PLOAD1 card. NSM is the non-structural mass per unit length.

The output from the ELFORCE (PRINT) request is as follows.

F O R C E S I N B E N D E L E M E N T S (C B E N D)

ELEMENT-ID	GRID	END	- BENDING MOMENTS - PLANE 1	PLANE 2	- SHEARS - PLANE 1	PLANE 2	AXIAL FORCE	TORQUE
991	1	A	-9.507964E+02	-1.118982E+02	-7.252229E+01	4.371233E+00	6.426835E+01	-3.338711E+02
	2	B	-6.589756E+02	-4.884176E+02	6.426834E+01	4.371233E+00	7.252229E+01	2.664446E+02



1.4.3 Two-Dimensional Element Cards

For linear analysis, shells assume the classical assumptions of this plate behavior, i.e. the thickness must be much smaller than the other dimensions, the deflection at the mid-surface must be small compared to its thickness and the mid-surface remains unstrained during bending by lateral loads.

The force, moment and stress are output. The Element Coordinate System is defined implicitly by the locations and order of the connected grid points in the Element Coordinate System. The element x-direction is defined by a vector from the first to the second grid point defined in the connectivity card. The element y-direction is perpendicular to the x-direction, and lies in the plane of the element. The element z-direction is defined by the right-hand rule. It is not really necessary to orient the X- and Y- axes of the elements in a common fashion so long as the element stresses are transformed into global coordinates in the post-processor. But it is always necessary to orient the Z-axis of the elements consistently as the stresses at the top and bottom surfaces are viewed separately. Forces and moments are output at the element center whilst stresses are output at distances Z1 and Z2 from the centroid surface. If Zi = ½ thickness (by default), then the stresses are output at the surfaces.

1.4.3.1 Transverse Bending, Transverse Shear and In-Plane Membrane (Plane Stress or Plane Strain) CQUAD4, CQUAD8, CTRIA3, CTRIA6 Elements

In general, the quadrilateral elements CQUAD4 and CQUAD8 should be preferred over the triangular elements CTRIA3 and CTRIA6, which are stiffer particularly for membrane strain. The latter are mainly useful to model portions of a structure when quadrilateral elements are impractical. Shell elements have 5 DOFs at each grid point, i.e. they do not have in plane rotational freedoms. Variable thickness can be defined, hence tapering is possible. For CTRIA6 and CQUAD8, the mid-side grid point need not be located on a straight line joining the adjacent corner points (because of the nature of isoparametric elements), hence these elements can model curved surfaces.

The shell elements have the capabilities of membrane stiffness, transverse bending stiffness and transverse shear stiffness. The membrane stiffness can be calculated using either the plane stress (default) or plane strain theories. Any of these capabilities may be omitted by the user. The elements CQUAD4, CQUAD8, CTRIA3 and CTRIA6 employ either the **PSHELL** or **PCOMP** property cards. Both the PSHELL and PCOMP cards can refer to either MAT1, MAT2 or MAT8. (Note that PSOLID cards refer to MAT1 or MAT9). The **PSHELL** card is described.

$ BULK DATA									
PSHELL	**PID**	**MID1**	**T**	**MID2**	**12I/T^3**	**MID3**	**TS/T**	**NSM**	
	Z1	**Z2**	**MID4**						

MID1 - Material for membrane stiffness.
MID2 - Material for bending stiffness.
MID3 - Material for transverse shear stiffness (thick plate theory for thicker shells)
MID4 - Material for coupling between membrane and bending deformation to model offsets.
$12I/T^3$ is the bending stiffness parameter, by default 1.0
TS/T is the transverse shear thickness divided by the membrane thickness, by default .833333
NSM is the non-structural mass per unit area

For solid homogenous plates, the default values of $12I/T^3$ and TS/T are correct. To obtain pure membrane behaviour only the MID1 should be specified; To obtain pure plate bending behaviour only the MID2 and MID3 fields should be specified. In reality, transverse load is resisted by both bending and membrane action, the latter when the geometry is curved or becomes curved as the nonlinear analysis progresses. Membrane action is the tension that occurs in the plane of the plate once some deformation has taken place, and this acts to resist the transverse force. For thin shells, membrane action is stiff compared to bending. Finally, transverse shear flexibility should be omitted only for comparison with academic solutions; Including the transverse shear flexibility always

gives a better answer for real-world structures. An element with shear flexibility will deflect more than an element without shear flexibility, hence the term shear flexibility instead of shear stiffness. The thicker the plate, the more significant is the contribution from the shear flexibility term, analogous to short deep beams. But for thin or curved surfaces, it is advisable not to use MID3.

The MID4 is simulating a similar effect to having an offset i.e. if you apply a tension (membrane load) it will cause bending deflections if you have this material set. It is rarely used but can be automatically generated on an unbalanced PCOMP entry. This has nothing to do with differential stiffness, which is accounted for nevertheless (with appropriate elements). Other methods of modelling plate offsets are by using ZOFFS on the element card or by using explicit rigid RBAR elements. Of the three methods MID4, ZOFFS and RBAR, the recommended approach is by using ZOFFS.

The **PCOMP** property card is used for modelling a composite material consisting of layers. This information is used internally within NASTRAN to compute equivalent PSHELL cards.

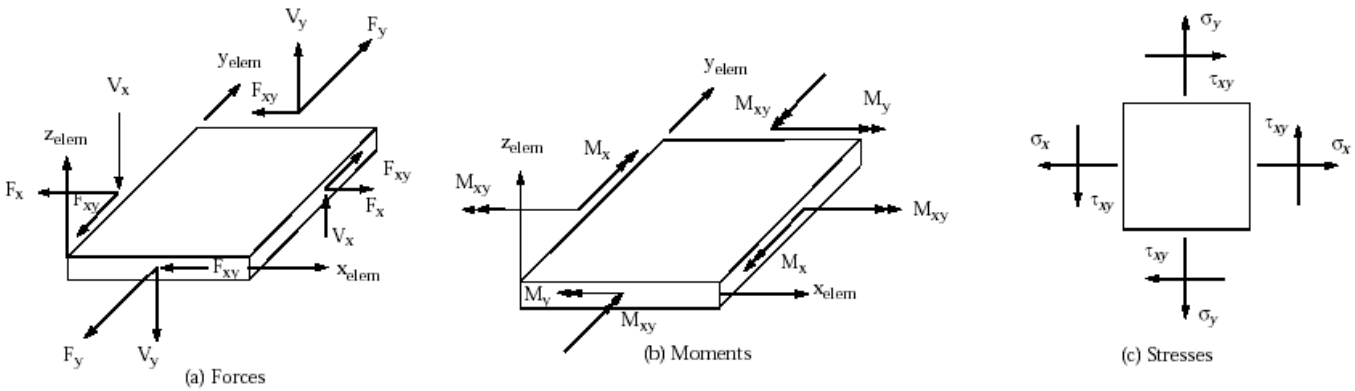
1.4.3.1.1 CQUAD4 Element

CQUAD4 element is a modified isoparametric element. Its membrane part uses reduced order integration for in-plane shear. Its bending part gives nearly exact results when the curvature varies linearly over the surface of the element. This element is the most commonly used element in NASTRAN. CQUAD4 are better for doubly-curved shells such as spheres. It is a robust element and behaves well when its shape is irregular. For example good results have been obtained with skew angles up to 45 degrees. There is no aspect ratio limit. The corner points are not required to lie in the same plane. It is recommended to avoid warping angles larger than 10 degrees.

$ BULK DATA									
CQUAD4	**EID**	**PID**	**G1**	**G2**	**G3**	**G4**	**THETA OR MCID**	**ZOFFS**	
			T1	**T2**	**T3**	**T4**			

Ti represents the shell thickness at the 4 corners, which can be omitted, in which case they will be set to the thickness defined in the property entry card. **ZOFFS** is used to model the **plate offset**. A positive ZOFFS implies that the element reference plane is offset a distance ZOFFS from the grid points along the positive **z-axis of the element coordinate system**. Naturally, MID1 and MID2 must be specified if offsets are used.

It is important to note that the forces generated for the elements are forces per unit length. A common error for new users is to assume that the force shown in the output is the total force acting on the element—it is not! It is the force per unit length.



1.4.3.1.2 CQUAD8 Element

The CQUAD8 is a modified isoparametric element with four corner grid points and four edge grid points. Compared with the CQUAD4 element, the CQUAD8 element usually provides greater accuracy at the same cost for flat plates and for singly-curved shells (such as a cylinder). The CQUAD4 however tends to perform better for doubly-curved shells (such as a sphere). For modelling cylindrical surfaces, the CQUAD8 should not sweep more than 30 degrees of the surface. CQUAD8 for mesh transition is not recommended. Deleting mid-side nodes is NOT recommended. For the CQUAD8 elements, if mid-side nodes are present, they should be located within the middle third of the edge. If a mid-side node is located at one-fourth the distance of the edge as measured from either corner node on the edge, the internal strain field becomes singular at the corners of the element. For best results, it is recommended that the mid-side node be located as close to the center of the edge as possible. If mid-side nodes are desired, the general recommendation is to include all of them. A CQUAD8 element with mid-side nodes deleted is excessively stiff and therefore is inferior to a CQUAD4 element.

$ BULK DATA									
CQUAD8	**EID**	**PID**	**G1**	**G2**	**G3**	**G4**	**G5**	**G6**	
	G7	**G8**	**T1**	**T2**	**T3**	**T4**	**THETA OR MCID**	**ZOFFS**	

QUAD8 is better than QUAD4 when there is in-plane bending because QUAD8 can curve because of the existence of mid-side nodes and its isoparametric formulation whilst the QUAD4 is straight at its edges, exhibiting shear locking behavior. QUAD8 is better than QUAD4 when there is out-of-plane bending because the stress recovery is linear (σ_x in x) in QUAD8 whilst constant in QUAD4.

1.4.3.1.3 CTRIA3 Element

CTRIA3 element is a constant strain, constant curvature triangle. CTRIA3 elements have constant membrane stresses. They are therefore not suitable for modelling areas with stress gradients (for example in the web of an I-beam) and **SHOULD BE MINIMIZED**. Quadrilateral elements (CQUAD4, CQUAD8) or CTRIA6 triangles are preferable. The primary application is to model portions of the structure where quadrilaterals are impractical, such as the center of a circular plate. Avoid using CTRIA3 in locations where the membrane stresses are changing rapidly, for example, in the web of an I-beam. Since CTRIA3 has constant membrane stresses, a large number of them may be needed to obtain acceptable accuracy. It is better to use quadrilateral elements or CTRIA6 elements. In fact, two CTRIA3 is not better than one QUAD4. To illustrate this, consider the membrane CQUAD4, which has 8 DOFs, i.e. capable of representing 8 independent deformation states, 3 of which are rigid body modes, leaving 5 independent stress-inducing deformation states. Contrast with the membrane CTRIA3, which has 6 DOFs, i.e. capable of representing 6 independent deformation states, 3 of which are rigid body modes, leaving 3 independent stress-inducing deformation states. Hence, two deformation states cannot be represented accurately by the CTRIA3 that can be by the CQUAD4.

$ BULK DATA									
CTRIA3	**EID**	**PID**	**G1**	**G2**	**G3**	**THETA OR MCID**	**ZOFFS**		
			T1	**T2**	**T3**				

Ti represents the shell thickness at the 4 corners, which can be omitted, in which case they will be set to the thickness defined in the property entry card.

1.4.3.1.4 CTRIA6 Element

CTRIA6 is a linear strain, linear curvature triangle with 3 corner nodes and 3 mid-side nodes. It is used in a similar way to CTRIA3, in areas where QUAD elements are unsuitable, in meshes with other elements having edge grids.

1.4.3.2 In-Plane Shear and Extension CSHEAR Elements

The CSHEAR element is a quadrilateral with 4 grid points that supports in-plane shear force only. It is used in the analysis of thin reinforced panels, especially if the panels are **curved** or **buckled**. The panels are then modeled by CROD elements carrying extensional loads and CSHEAR elements carrying in-plane shear. The CSHEAR element can also carry extensional force between adjacent grid points using the optional parameters F1 and F2 on the PSHEAR card. This offers an efficient way to model the stiffness of the panel for extensional loads. Typically you use the CSHEAR element in situations where the bending stiffness and axial membrane stiffness of the plate is negligible. The use of CQUAD4 element in such situations results in an overly stiff model. The most important application of the CSHEAR element is in the analysis of thin reinforced plates and shells, such as thin aircraft skin panels. In such applications, reinforcing rods (or beams) carry the extensional load, and the CSHEAR element carries the in-plane shear. This is particularly true if the real panel is buckled or if it is curved. Do not use plate or shell elements (CQUADi, CTRIAi) in stiffened shell structures with very thin panels that can buckle. Shear panels (CSHEAR) should be used in this case or in any situation where direct stresses cannot be supported, such as in a very thin curved panel.

$ BULK DATA									
CSHEAR	**EID**	**PID**	**G1**	**G2**	**G3**	**G4**			
PSHEAR	**PID**	**MID**	**T**	**NSM**	**F1**	**F2**			

The optional parameters F1 and F2 on the PSHEAR entry are useful in representing an effective stiffness of the panel for extensional loads by means of equivalent rods on the perimeter of the element.

1.4.3.3 CRACK2D

The CRAC2D element is used to model surfaces with a discontinuity due to a crack.

1.4.4 Three-Dimensional Element Cards

Solid elements connect only translational DOFs at grid points.

$ BULK DATA									
PSOLID	PID	MID	CORDM	IN	STRESS	ISOP	FCNT		

CORDM is the material coordinate system
IN is the integration network
STRESS is the location for stress output
ISOP is the integration scheme
FCNT is the fluid element flag, SMECH for structural element and PFLUID for fluid element.

Unlike plate and shell elements where stress components are always recorded in the element coordinate system, the stresses for the HEXA and PENTA elements are recorded in the material coordinate system. The user has a several ways of defining the material coordinate system specified on Field 4, either the Basic Coordinate System (0 or blank as default), the Element Coordinate System (-1), or a Local Coordinate System (Integer > 0).

By default, the stress output for the solid elements is at the center and at each of the corner points. For CHEXA and CPENTA, if no mid-side nodes are used, you may request the stress output at the Gauss points instead of the corner points by setting Field 6 of the PSOLID entry to “GAUSS” or 1. Gauss point output is available for the CTETRA element with or without mid-side nodes.

The default integration schemes may be overridden with Field 5 and Field 7 of the PSOLID entry. In general, it is recommended that you use the default value. An override of the default values is available if you are familiar with element theory and have a particular modelling need; otherwise, use the defaults.

CHEXA	Integration	IN	STRESS (Default: GRID)	ISOP (Default: See Remarks 5 and 7.)	Nonlinear Capability
8 Node	2x2x2 Reduced Shear with Bubble Function (default)	BUBBLE or Blank or 0 (Default)	Blank or GRID or GAUSS or 1	Blank or REDUCED (Default*)	Yes
	2x2x2 Reduced Shear Only	TWO or 2			
	2x2x2 Standard Isoparametric			FULL or 1	
	3x3x3 Reduced Shear Only	THREE or 3	Blank or GRID	Blank or REDUCED	No
	3x3x3 Standard Isoparametric			FULL or 1	
9–20 Node	2x2x2 Reduced Shear Only	TWO or 2	Blank or GRID	Blank or REDUCED	No
	2x2x2 Standard Isoparametric			FULL or 1	
	3x3x3 Reduced Shear Only (default)	Blank or THREE or 3 (Default)		Blank or REDUCED (Default*)	
	3 x3x3 Standard Isoparametric			FULL or 1	
p-elements	Reduced (p-order) Bubble	0 or 1	Not applicable	0	No
	Bubble, P+ISOP Integration	1		$-10 \le ISOP \le 10$	
	No Bubble, P+ISOP Integration	2 or 3			

*REDUCED is the default only for structural elements (FCTN=“SMECH”).

CPENTA	Integration	IN	STRESS (Default: GRID)	ISOP (Default: See Remarks 5 and 7.)	Nonlinear Capability
p-elements Bubble Function	2x3 for p=1, 1, 1 3x7 for p=2, 2, 2 (p+1)x(p) for all other	0 or 1	Not applicable	0	No
	2x3 for p=1, 1, 1 3x7 for p=2, 2,2	1		0	
	3x7 for p=1, 1, 1			1	
	(p+ISOP+1)x (p +ISOP) for all other			$-10 \le ISOP \le -1$ or $2 \le ISOP \le 10$	
p-elements Standard Isopara-metric: (no bubble function)	2x3 for p=1, 1, 1 3x7 for p=2, 2, 2	2 or 3		0	
	3x7 for p=1, 1, 1			1	
	(p+ISOP+1)x (p +ISOP) for all other			$-10 \le ISOP \le -1$ or $2 \le ISOP \le 10$	

*REDUCED is the default only for structural elements (FCTN=“SMECH”).

CPENTA	Integration	IN	STRESS (Default: GRID)	ISOP (Default: See Remarks 5 and 7.)	Nonlinear Capability
6 Node	2x3 Reduced Shear with Bubble Function (Default)	Blank or 0 or BUBBLE (Default)	GAUSS or 1 or Blank or GRID	Blank or REDUCED (Default*)	Yes
	2x3 Reduced Shear Only	TWO or 2			
	2x3 Standard Isoparametric			FULL or 1	
	3x7 Reduced Shear Only	THREE or 3	Blank or GRID	Blank or REDUCED	No
	3x7 Standard Isoparametric			FULL or 1	
7—15 Node	2x3 Reduced Shear Only	TWO or 2	Blank or GRID	Blank or REDUCED	No
	2x3 Standard Isoparametric			FULL or 1	
	3x7 Reduced Shear Only (default)	Blank or THREE or 3 (Default)		Blank or REDUCED (Default*)	
	3x7 Standard Isoparametric			FULL or 1	

CTETRA	Integration	IN	STRESS (Default: GRID)	ISOP	Nonlinear Capability
4 Node	1-Point Standard Isoparametric (Default)	Blank or TWO or 2 (Default)	GAUSS or 1 or Blank or GRID	Blank or FULL	Yes
	5-Point Standard Isoparametric	THREE or 3	Blank or GRID		No
5—10 Node	5-Point Standard Isoparametric	Blank or THREE or 3 (Default)	GAUSS or 1 or Blank or GRID	Blank or FULL	No
p-elements	1-Point; P=1,1,1 5-Point; P=2,2,2 P+1 Cubic Point; P>2	0 or 1	Not applicable	0	No
	5-Point; P=1,1,1 P+1 Cubic for all other			1	
	P+ISOP Cubic			$-10 \le ISOP \le -1$ or $2 = ISOP \le 10$	

1.4.4.1 CHEXA8 or CHEXA20 Element

The six-faced CHEXA element has 8 corner nodes forming a rectangular brick shaped element. If mid-side nodes are specified, then there will be an additional 12 grids. This element is more recommended than other degenerate solid elements. Its performance degrades when subjected to prominent bending. The mid-side grids G9 to G20 are optional. Any or all of them may be deleted, but not recommended. The equations are modified to take into account the available grids. Of course, the corner grids cannot be omitted. As in the case of plate elements, if mid-side nodes are present for the solid elements, they should be located as close to the center of the edge as possible.

$ BULK DATA									
CHEXA	**EID**	**PID**	**G1**	**G2**	**G3**	**G4**	**G5**	**G6**	
	G7	**G8**	**G9**	**G10**	**G11**	**G12**	**G13**	**G14**	
	G15	**G16**	**G17**	**G18**	**G19**	**G20**			

1.4.4.2 CPENTA6 or CPENTA15 Element

The five-faced CPENTA element has 6 corner grids and has 9 available mid-side edge grids, forming a wedge shape. It is commonly used to model transitions from solids to shells. The triangular faces should be on the exposed faces of the shells, to avoid excessive stiffness. Any or all of the mid-side grids may be omitted, but is not recommended.

$ BULK DATA									
CPENTA	**EID**	**PID**	**G1**	**G2**	**G3**	**G4**	**G5**	**G6**	
	G7	**G8**	**G9**	**G10**	**G11**	**G12**	**G13**	**G14**	
	G15								

1.4.4.3 CTETRA4 or CTETRA10 Element

The four-faced CTETRA is a quadratic (linear strain) isoparametric solid element with 4 corner grids and 6 available mid-side grid points. A linear (constant strain) CTETRA may be obtained by omitting the edge grid points. Partial omission of edge grid points is allowed, but is not recommended. The recommended application of the CTETRA element is to fill in odd-shaped holes in meshes made with CHEXA and CPENTA elements. If large portions of the model must be modeled using CTETRA elements, then a very fine mesh density is recommended in regions of steep stress gradients. If you use edge nodes, it is recommended that you include all six nodes. The accuracy of the element degrades if some but not all of the edge grid points are used. In bending situations, two or more tetrahedrons are required through the wall thickness to capture behaviour. It is a good modelling practice to keep the use of **CTETRA4 TO A MINIMUM**, especially in the areas of high stress. CTETRA10 should be used as the standard TETRA element, even before h-refinement is undertaken as CTETRA4 is overly stiff as it is a constant strain element. Studies show that properly converged second order CTETRA10 can provide the same accuracy as a linear brick element. CTETRA has the advantage of being very easy to mesh with an automated mesh generator and is the norm for CFD type applications. A skin of TRIAs are generated with an automated mesh generator on defined surface geometry, then the tetra mesh is employed to fill the volume. The skin can be remained to act for stress recovery purposes on the surface of the component where the stress is most likely to be the highest. Note that the thickness of the skin is usually in the order of around 0.01mm for standard components and connections of 1m nominal dimensions say.

1.4.4.4 CRACK3D

The CRAC3D element is used to model volumes with a discontinuity due to a crack.

1.5 Mass Element Cards

The CMASS1 (with PMASS) and CMASS2 elements define a scalar mass element between two grid points (and across particular DOF pairs in the CD displacement coordinate system of the grid point). The grid IDs G1 and G2 must be distinct, however a zero or blank field will indicate a grounded terminal.

$ BULK DATA									
CMASS1	EID	PID	Grid ID G1	DOF C1	Grid ID G2	DOF C2			
PMASS	PID1	M1	PID2	M2	PID3	M3	PID4	M4	

$ BULK DATA									
CMASS2	EID	M	Grid ID G1	DOF C1	Grid ID G2	DOF C2			

The CONM1 element allows input of a fully coupled 6 x 6 mass matrix. Half the terms are defined and symmetry is assumed.

$ BULK DATA									
CONM1	EID	Grid ID	CID	M11	M21	M22	M31	M32	
	M33	M41	M42	M43	M44	M51	M52	M53	
	M54	M55	M61	M62	M63	M64	M65	M66	

The CONM2 element allows the specification of a concentrated mass at a grid point. This element also allows the definition of a center of gravity offset from the grid point and moments and products of inertia about the center of gravity.

$ BULK DATA									
CONM2	EID	Grid ID	CID	M	X1	X2	X3		
	I11	I21	I22	I31	I32	I33			

The mass matrix [M] is automatically computed from

(i) the mass density (mass per unit volume) in the property cards
(ii) the non-structural mass (mass/length or mass/area) in the property cards
(iii) concentrated mass elements

The mass matrix [M] is obtained by either the lumped mass method or the coupled mass method (based on the classical consistent mass method). The lumped mass model contains uncoupled components of translational mass and uncoupled components of rotational or torsional inertia components. The lumped mass model thus generates diagonal mass matrices. The coupled mass method on the other hand contains coupled components of mass between translational DOFs and coupled components of inertia between rotational DOFs, but no coupling between translational and rotational DOFs. The coupled mass matrix is thus not diagonal. It formulates the mass matrix by a procedure similar to the formulation of the stiffness matrix by applying unit accelerations at successive nodal freedoms and investigating the force generated at all the freedoms. Although the coupled mass method is more accurate, the lumped mass method is generally preferred for its computational speed. Coupled mass formulation is especially more accurate to model high frequency modes. To specify coupled mass rather than the default lumped mass formulation,

```
$ BULK DATA
PARAM, COUPMASS, 1
```

The dynamic mass often includes the medium surrounding the structure. For instance, in the analysis of offshore gravity platforms, some degree of the mass of the water surrounding the shafts must be included within the mass. Noting that the mass acts in 3 translational orthogonal directions (likewise inertia acts about 3 orthogonal axes), it is essential that the correct component be modeled. The added mass on the shafts is a horizontal mass component whilst the added mass on the base of the gravity base structure is a vertical mass component. The inclusion of added masses on the dynamic model also means that a uniform global acceleration applied to all parts of the structure to model gravitational force cannot be employed, as the added mass will incorrectly increase the weight of the structure. Instead, the gravitational force must be modeled using explicit nodal loads or more accurately, gravitational accelerations applied to a set of masses that excludes the added masses.

Another important consideration when modelling non-structural mass is that if rigid links are used to model the center of gravity of the non-structural mass from the structure, then a coupled mass formulation (kinematically equivalent mass matrix) must be used. This is because if lumped mass formulation is used, the rigid links simply will transfer the non-structural mass onto the structure, and hence effectively not incorporating the offset. This will cause significant errors in the lowest frequencies [9].

[9] NAFEMS. *A Finite Element Primer.* NAFEMS Ltd., Great Britain, 1992.

1.6 Damping Element Cards

1.6.1 Viscous Damping Elements CDAMP and CVISC

The **CDAMP1** (with PDAMP) and CDAMP2 elements define a **scalar** damping between two grid points (and across particular DOF pairs in the CD displacement coordinate system of the grid point). The damping constant is expressed in units of force/velocity. The grid ids G1 and G2 must be distinct, however a zero or blank field will indicate a grounded terminal. In general the direction of the scalar viscous damping will not be coincident with a global DOF. This necessitates the definition of a local coordinate system using CORD1R or CORD2R at the grids (and defined on Field 7 of the two GRID entries) where the scalar viscous damping is required. As with any scalar element, the CDAMP element **MUST BE DEFINED** as a **ZERO LENGTH** element (as a scalar element should not have a dimension anyway), and not one with a finite length in order to avoid internal constraints. To this end, the **CDAMP element has been somewhat superceded by the CBUSH or CBUSH1D element.**

$ BULK DATA									
CDAMP1	**EID**	**PID**	**Grid ID G1**	**DOF C1**	**Grid ID G2**	**DOF C2**			
PDAMP	**PID1**	**C1**	**PID2**	**C2**	**PID3**	**C3**	**PID4**	**C4**	

$ BULK DATA									
CDAMP2	**EID**	**C**	**Grid ID G1**	**DOF C1**	**Grid ID G2**	**DOF C2**			

The **CVISC** element is used to define a viscous damping between two grid points in the direction defined by the vector joining the grids and has an associated PVISC card to define the translational CE and rotational CR viscous damping properties. The grid ids G1 and G2 must be distinct. **The CVISC element has been somewhat superceded by the CBUSH or CBUSH1D element.**

$ BULK DATA									
CVISC	**EID**	**PID**	**Grid ID G1**	**Grid ID G2**					
PVISC	**PID1**	**CE1**	**CR1**		**PID2**	**CE2**	**CR2**		

1.6.2 Structural Damping Elements

Elemental structural damping is defined as follows. If the model consists of a single material, such as a casting, then only one damping value is needed in which case the PARAM, G can be used

$ BULK DATA
PARAM, G, < Damping constant >

The other way of specifying structural damping is to use GE in the MAT1 card enabling different values to be used for the various components of the model. Structural damping can also be specified using CELAS or CBUSH.

1.7 General Nonlinear Excitation Frequency Dependent Spring And Linear Excitation Frequency Dependent Damper CBUSH (PBUSH and PBUSHT) Element

Note that the CELASi and CDAMPi elements are non-metric elements. This means that grid point geometry is not used when assembling them into the system stiffness matrix. They will always produce a 2 by 2 matrix, regardless of offsets, coordinate systems, or other modelling details. This can result in internal constraints in the element, which in turn leads to constraint forces to ground that do not appear in the SPC forces. An internal constraint means that the element is not in equilibrium. The effect of internal constraints can be investigated by checking the applied load resultant (OLOAD RESULTANT) and the reaction resultant (SPCFORCE RESULTANT), a discrepancy indicating the existence of internal constraints. The CBUSH element can be thought of as the modern replacement for the CELASi and CDAMPi scalar elements. It is a metric element, and will properly account for offsets, coordinate systems, and any other modelling data. It is always in equilibrium. It is recommended that all scalar elements in models be replaced with CBUSH elements, especially if the input file is passed onto a different modeller. If the new modeller modifies the grid point geometry the CBUSH element will still be in equilibrium. This may not be true for scalar elements. Must **NOT** use local coordinate systems on GRIDs with CBUSH. The CBUSH element requires the definition of a local coordinate system in itself. The element force is given by

$$f_e = k(f,u)u + c(f)\dot{u} + iG_E(f)ku$$

where f is the excitation frequency. Frequency dependent stiffness and damper properties are applicable only to forced frequency response analyses SOL 108, SOL 111 and SOL 200. Nonlinear spring properties are only applicable to SOL 106 and SOL 129. Note however that the CBUSH element is not geometrically nonlinear. CBUSH is the recommended element enabling up to 6 uncoupled stiffness and damping values to be inputted. PBUSH provides nominal values and a PBUSHT (with the same ID as a PBUSH card) provides the frequency dependency and nonlinear characteristics.

CBUSH	EID	PID	GA	GB	GO/X1	X2	X3	CID	
	S	OCID	S1	S2	S3				

PBUSH	PID	"K"	K1	K2	K3	K4	K5	K6	
		"B"	B1	B2	B3	B4	B5	B6	
		"GE"	GE1						
		"RCV"	SA	ST	EA	ET			

PBUSHT	PID	"K"	TKID1	TKID2	TKID3	TKID4	TKID5	TKID6	
		"B"	TBID1	TBID2	TBID3	TBID4	TBID5	TBID6	
		"GE"	TGEID1						
		"KN"	TKNID1	TKIND2	TKNID3	TKIND4	TKIND5	TKIND6	

The third node G0, the orientation vector X1, X2 and X3 or the user defined coordinate system CID defines the element coordinate system. Note that the stiffness and damping values are defined at the elastomer center of elasticity, from which the properties are transferred to the grid nodes using rigid element equations. The output from the ELFORCE (PRINT) request (in the element coordinate system) is as follows.

F O R C E S I N B U S H E L E M E N T S (C B U S H)

ELEMENT-ID	FORCE-X	FORCE-Y	FORCE-Z	MOMENT-X	MOMENT-Y	MOMENT-Z
10000	-6.489702E+00	-5.897736E+00	6.265953E-01	3.955022E+00	-9.723748E-02	1.147097E-02
10001	-3.201495E+00	-1.851144E+00	2.724363E-01	8.105669E+00	-1.866592E-01	2.313610E-02
10002	9.125268E-01	2.533982E+00	2.790101E-01	8.008885E+00	-1.804384E-01	2.278207E-02
10003	2.938818E+00	5.093895E+00	3.161362E-01	7.944401E+00	-1.380170E-01	2.264773E-02
10004	3.530923E+00	7.028401E+00	3.253268E-01	7.911948E+00	-8.659782E-02	2.265486E-02
10005	6.848238E+00	4.654799E+00	3.589581E-01	7.842767E+00	-2.292452E-02	2.255161E-02
10006	6.788301E+00	3.347241E+00	3.445033E-01	7.912354E+00	4.136241E-02	2.287587E-02
10007	4.012078E+00	2.324608E+00	3.328075E-01	7.965162E+00	1.027839E-01	2.302680E-02
10008	5.789543E-01	-4.556357E-01	3.059701E-01	8.035738E+00	1.431401E-01	2.305412E-02
10009	-4.496515E+00	-5.173875E+00	2.923176E-01	8.129048E+00	1.063870E-01	2.287990E-02
10010	-7.422128E+00	-7.604536E+00	5.459386E-01	3.958032E+00	-3.207400E-02	1.047710E-02

1.8 One Dimensional Nonlinear Spring And Nonlinear Damper CBUSH1D (PBUSH1D) Element

The CBUSH1D element is a 1D version of CBUSH but with added features. It is materially and geometrically nonlinear (for stiffness and damping) for SOL 106 and SOL 129. The element force is given by

$$f_e = c(u)\text{sign}(\dot{u})\dot{u}^{EXPV} + k(u)u + c(\dot{u})\dot{u}$$

CBUSH1D	EID	PID	GA	GB	CID				

PBUSH1D	PID	K	C	M		SA	SE		
	"SHOCKA"	TYPE	CVT	CVC	EXPVT	EXPVC	IDTS		
			IDETS	IDECS	IDETSD	IDECSD			
	"SPRING"	TYPE	IDT	IDC	IDTDU	IDCDU			
	"DAMPER"	TYPE	IDT	IDC	IDTDV	IDCDV			
	"GENER"		IDT	IDC	IDTDU	IDCDU	IDTDV	IDCDV	

1.9 Material Cards

1.9.1 Linear, Elastic, Isotropic Material Card for 1-D, 2-D and 3-D Elements MAT1

The user needs to specify in the MAT1 card 2 of the 3 elastic constants: elastic modulus E, shear modulus G and Poisson' s ratio NU. The value of the third constant is calculated by NASTRAN using $G = E/[2(1+\nu)]$. Note that specifying all 3 constants can give incorrect results.

$ BULK DATA									
MAT1	**MID**	**E**	**G**	**NU**	**RHO**	A	TREF	GE	
	ST	SC	SS	MCSID					

RHO density
A thermal expansion coefficient
TREF reference temperature for the calculation of thermal loads
GE structural damping coefficient
ST, SC, SS stress limits for tension, compression and shear used only to compute margins of safety in certain elements
MCSID material coordinate system id

Element	E	NU	G
CROD, CBAR, CBEAM	Extension and bending	N/A	Torsion and transverse shear
CQUAD4, CQUAD8, CTRIA3, CTRIA6	Membrane and bending		Transverse shear
CSHEAR	N/A		Shear
CHEXA, CPENTA, CTETRA	All terms		N/A

	E (N/m^2)	ρ (kg/m^3)	ν	Thermal Expansion (Strain/°C)
Steel	2.05E11	7850	0.3	1.2E-5
Concrete Short Term	2.8E10	2400	0.2	1.0E-5
Concrete Long Term	1.4E10	2400	0.2	1.0E-5
Aluminium	7.0E10	2710	0.34	2.3E-5

Modelling of concrete as a linear elastic material assumes that the concrete does not crack. The cracking of concrete leads to a redistribution of load. Since the cracking occurs at very low strains, concrete is a highly nonlinear material. In linear analyses, usually a short-term high stiffness concrete and long-term low stiffness concrete stiffness is assumed in separate analyses to bound the results.

1.9.2 Linear, Elastic, Anisotropic Elastic Material for Shell MAT2 and Solid Elements MAT9

Anisotropic refers to properties that differ in two or more directions.

1.9.3 Linear, Elastic, Orthotropic Elastic Material for Shell Elements MAT8 and Solid Elements MAT9

Orthotropic refers to a specific type of anisotropy in which planes of extreme values are orthogonal.

1.10 Rigid Element Cards

Rigid elements are equations of motion connecting DOFs. They therefore do not have physical properties such as cross-section, stiffness or mass.

RROD	A pin-ended rod which is rigid in extension, i.e. 1 constraint equation
RBAR	A rigid bar with six DOFs at each end, i.e. 6 constraint equations
RTRPLT	A rigid triangular plate with six DOFs at each vertex, i.e. 12 constraint equations
RBE1	A rigid element connected to an arbitrary number of grid points. The independent and dependent DOFs can be arbitrarily selected by the user.
RBE2	A rigid element connected to an arbitrary number of grid points. The independent DOFs are the six components of motion at a single grid point. The dependent DOFs at the other grid points all have the same component numbers.

$ BULK DATA									
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4	GM5	
	GM6	GM7	GM8	…etc…					
RBAR	EID	GNA	GMB	CNA	CNB	CMA	CMB		

GN-independent grid; GM-dependent grids; CN-independent grid components; CM-dependent grid components;

Note that a dependent DOF in one element cannot also be a dependent DOF in another rigid element since this will produce conflicting equations of motion. The dependent DOF should also have no SPC specified to it. Note however that an independent DOF can be a dependent DOF in another rigid element.

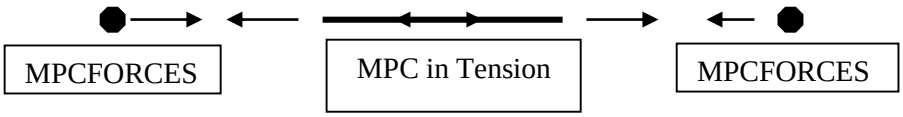
Rigid elements are used to

I. model connections such as spot welds,
II. define hinges or sliding joints,
III. distribute loads,
IV. connect dissimilar elements such as solids to shells
V. model stiffeners on plates; the stiffener can be a CBAR element offset by rigid elements from a shell structure to a distance equal to half the thickness of the stiffener
VI. replace very stiff elements, which will produce numerical ill conditioning.

Rigid elements, however, should be used with great care. Rigid elements can transfer large loads, but since they are not real elements they do not provide load or stress output.

Application	R-Type Entries
Triangular Bell Crank	RTRPLT
Rigid Engine Blocks	RBE1
Tripod with Hinged Rigid Legs	RROD
Rigid Bulkhead	RBE2
Evaluation of Resultant Loads	RBE2
Connection of a Bar Element to a Shell	RBE2 or RBE3
Hinge Between Two Plates	RBAR
Recording Motion in a Nonglobal Direction	RBAR
Relative Motion	MPC
Incompressible Fluid in an Elastic Container	MPC
"Beaming" Loads and Masses	RBE3
Change in Mesh Size	RSPLINE
Transitions Between Plate and Solid Elements	RSSCON

Rigid element and MPC grid point forces are requested with MPCFORCES (PRINT, PLOT). Note that akin to GPFORCE, MPCFORCES output the force acting on the grid point, NOT ON THE ELEMENT. In fact, MPCFORCES is the resultant force on the grid point from all adjoining rigid or MPC elements. This is clearly not very useful unless there is only one MPC on the grid, so that the MPCFORCES will be the forces acting on the grid due to ONLY that particular MPC. In that case, we can establish the force within the MPC by switching the directions of the forces as illustrated below.



This is very much akin to GPFORCES (in the sense that it is the force on the grid not on the element(s)), hence the reason for omitting forces from rigid or MPC elements from GPFORCE output. In the situation where there are many MPC elements connecting into a grid, the use of a stiff CELAS or CBUSH is necessary to find out the force going through an MPC element. The output from MPCFORCEs is the summation of all MPC-related forces at each GRID point. If only R-elements and MPCs are used to connect a point to the model, NASTRAN will provide 0.0 as the MPCFORCE at that GRID. This may be misleading, as the output does not provide forces for each MPC and R-element separately, rather it provides only the summation of all MPC and R-element forces at the GRID point. For example, if the following model of a beam with a 100.0 unit axial force is used,

```
100→ x------x------x------x RBAR x RBAR x------x------x ←100.0
       1      2      3      4        5        6      7      8
```

the MPCFORCE at GRID 5 will show as 0.0, even although each of the connected RBAR elements has an axial force of 100.0 units. If you replace one of the RBARs by a CBUSH element, the MPC-forces will now show the components from the remaining RBAR (100.0 units), which are equal and opposite to the forces in the BUSH element (which will not show up as MPCFORCE output but ELFORCE output).

Two methods for rigid elements are available, i.e. the **linear method** and the **Lagrange method** (see **Section 1.3.23**). The rigid elements are not real elements, they are internally represented by a set of MPC equations. By using these MPC equations, the dependent degrees-of-freedom (the m-set) are eliminated from the solution set. The rigid elements linear method has the following limitations; do not compute thermal load, do not have a differential stiffness matrix; therefore, the solutions are incorrect for the buckling analysis or other solution sequences where the differential stiffness matrix is required, use the small rotation theory in the geometrical nonlinear analysis so that the solutions are incorrect in this type of analysis, use the elimination method for solution, resulting in very dense stiffness matrices. These dense matrices cannot take advantages of the sparse matrix algorithm. The Lagrange method does not have the above limitations. With the Lagrange method, the rigid elements become "real" finite elements, similar to, for example, a QUAD4 element. Instead of using MPC equations, the element stiffness matrix is computed for each rigid element. Two solution methods are available, namely the **Augmented Lagrange Multiplier Method** (for this method, the solution is obtained with the independent degrees-of-freedom, the dependent degrees-of-freedom, and the Lagrange multipliers degrees-of-freedom left in the solution set. Thus, the sparse characteristic of the stiffness matrix is maintained and sparse matrix algorithms can be used) and the **Lagrange Elimination Method** (for this method, additional operations of eliminating both the dependent and Lagrange multiplier degrees-of-freedom from the solution set are performed after the global stiffness matrix is assembled. The solution is performed on the independent set (the n-set), creating dense matrices after elimination. Once again, these dense matrices cannot take advantage of the efficiency features of sparse methods). For most problems, the augmented Lagrange multiplier method is preferred. The tangent stiffness matrix using this method, however, is not positive definite, and can potentially pose numerical difficulties. The Lagrange elimination method is introduced as a backup for eigenvalue problems. The Lagrange elements are only available for linear static analysis (SOL 101), normal modes analysis (SOL 103), buckling analysis (SOL 105) and general nonlinear analysis (SOL 400) but **not** in SOL 106 and SOL 129. Note that the application of proper nonlinear rigid elements is one of the major benefits of SOL 400 to SOL 106 and SOL 129. The Lagrange elimination method is only recommended for eigenvalue analysis, it is not recommended in nonlinear static analysis (but Augmented Multiplier Method is fine for general nonlinear scheme SOL 400) for the following reasons. For nonlinear analysis,

the tangent stiffness matrix must be updated at each nonlinear iteration for the Lagrange elimination method. This implies that only the full Newton-Raphson method can be used, which conflicts with the general iteration solution scheme employed in MSC.Nastran. The Control Case command **RIGID** selects the type of rigid element. It has the following format:

$$\text{RIGID} = \begin{Bmatrix} \text{LINEAR} \\ \text{LAGR} \\ \text{LGELIM} \end{Bmatrix}$$

LINEAR will select the linear rigid elements, LAGR will select the Lagrange rigid element with the Lagrange multiplier method, and LGELIM will select the Lagrange element with the Lagrange elimination method. If RIGID command does not exist in the user Case Control file, the linear rigid element will be used for all solution sequences except SOL 400. For SOL 400, the default rigid element type is the Lagrange rigid element with the Lagrange multiplier method. The parameter **LANGLE** selects the method to represent the lager rotations, having the following values:

LANGLE,1 -- use the Gimbal angle method.
LANGLE,2 -- use the left rotation method.
LANGLE,3 -- use the right rotation method.

LANGLE=1 or 2 is the existing method. LANGLE=3 is the new method implemented for the Lagrange rigid elements and is the only method available for the Lagrange rigid elements. Therefore, if Lagrange rigid elements are present in a structural model, MSC.Nastran will automatically use LANGLE=3, and ignore user input on the LANGLE parameter. The parameters **LMFACT** and **PENFN** are the scale factor *k* and penalty function *p* for the Lagrange rigid element with the Lagrange multiplier method. The purpose of LMFACT and PENFN is to make the values of the stiffness matrix for the Lagrange rigid elements approximately equal in relative magnitude to those of the other elements in a structural model. Too small a value produces inaccurate results and too large a value poses numerical difficulties. Usually, the same value is assigned to both LMFACT and PENFN. Under special requirements, the user may assign different values to LMFACT and PENFN. For example, if and PENFN = 0.0, then the solution method becomes the pure Lagrange multiplier method instead of the augmented Lagrange multiplier method. However, the user must exercise caution when assigning different values to LMFACT and PENFN. The default value is 1.0e+5. for all solution sequences except SOL 400. For the nonlinear solution sequence SOL 400, MSC.Nastran will compute the appropriate default value in nonlinear iterations.

1.11 Boundary Conditions

1.11.1 Single Point Constraints (SPCs)

It is important to ensure that the SPCs restrain the rigid body motion of the model in all 6 degrees-of-freedom (translations in x,y,z directions, rotations about x,y,z axes). Failure to do so would cause singularity in the stiffness matrix in linear static analyses. Note that this applies even if the user is applying a balanced set of loads on the model (e.g. applying internal pressure in a closed pipe). In this case restraints in 6 degrees-of-freedom must be specified, although they would carry negligible loads.

A special case of rigid body motion occurs when at least a part of the structure is weakly coupled to the structure, in which case a mechanism occurs causing the matrix to again be singular.

Another application of SPCs is for applying symmetric or anti-symmetric boundary conditions by restraining the appropriate degrees-of-freedom on the plane of symmetry. However in dynamic analysis, a symmetric boundary will not incorporate all the modes. For instance, for a beam, symmetric boundary conditions will capture odd bending modes, whilst antisymmetric boundary conditions are required for the even modes.

The **SPC** card can be used to specify constraints and enforced displacement on individual grids, whilst **SPC1** cards are used to specify a similar constraint on many grids simultaneously.

$ CASE CONTROL									
SPC = < SPC SID in Bulk Data Section >									
$ BULK DATA									
SPC	**SID**	**G1**	**C1**	**D1**	**G2**	**C2**	**D2**		

$ CASE CONTROL									
SPC = < SPC SID in Bulk Data Section >									
$ BULK DATA									
SPC1	**SID**	**C**	**G1**	**G2**	**G3**	**G4**	**G5**	**G6**	
	G7	**G8**	**THRU**	**G47**	**G78**	…**etc**…			

Gi – grid ids
Ci – component numbers
Di – enforced displacement

1.12 Linear Optimization SOL 200

SOL 200 is applicable to SOL 101, SOL 103, SOL 105, SOL 108, SOL 111, SOL 112 and static aeroelastic. The analysis command in the Case Control Section specifies the type of analysis to be undertaken.

```
$ EXECUTIVE CONTROL SECTION
SOL 200
$ CASE CONTROL SECTION

SUBCASE 1
ANALYSIS = STATIC
SUBCASE 2
ANALYSIS = MODES
```

1.12.1 Objective Function and Constraints

Response quantities that can be optimized for include weight, volume, natural frequencies, buckling load factors, displacements, velocities, accelerations, stresses, strains, forces, frequency or transient response displacements, velocity, acceleration, force and stress, trim, stability derivatives in a static Aeroelastic analysis, damping levels in flutter and whatever user-defined analytical equation supplied. The case control commands are as follows.

```
$ CASE CONTROL SECTION
DESOBJ(MAX or MIN) = < ID of DRESP1 or DRESP2 >
DESGLB = < ID of global constraints defined by DCONSTR >
SUBCASE 1
DESSUB = < ID of subcase dependent constraints defined by DCONSTR >
SUBCASE 2
DESSUB = < ID of subcase dependent constraints defined by DCONSTR >
```

The bulk data entries are described as follows.

DRESP1	defines responses computed directly by the analysis
DRESP2	defines synthesized responses for design problems
DCONSTR	defines the response constraints
DCONADD	defines constraint set combinations
DEQATN	defines the user-defined equations
DTABLE	defines a table of real constants that are used in equations in DEQATN

Do not use 0.0 for lower or upper bounds on DCONSTR.

1.12.2 Design Variables and Constraints

DVPREL1 and DVPREL2 defines the field of the property entries (model parameters) that needs to be optimized referring to DESVAR entries to define how the values are to be altered in order to be optimized. DVPREL1 defines linear relations the model parameters and the design variables whilst DVPREL2 defines nonlinear relations. DVPREL1 thus defines the basic relationship between the model parameters P_i and the various design variables as

$$P_i = C0 + \sum_i \text{COEFFi x DVIDi}$$

DESVAR	defines the design variables
DLINK	defines the dependent design variables

Shape optimization is performed using the DVGRID bulk data entry (instead of DVPREL1 or DVPREL2) which references the design variables DESVAR. It defines the direction and magnitude of a grid variation for a given change in a design variable.

1.12.3 Optimization Control Parameters

DSCREEN specifies measures of constraint screening
DOPTPRM defines the optimization process control

1.13 Computational Memory and Processing Power Demand

If the stiffness matrix is stored and processed as if it were a *full* matrix, the storage and processing time resources rapidly becomes prohibitive as the number of DOFs N increases. With regards to memory needs, a full square matrix stored without taking advantage of symmetry, requires storage for N^2 entries. If each entry is an 8-byte (32-bit) double precision floating-point number (note that single-precision floating-point numbers require 4-bytes or 16-bits), the required storage is $8N^2$ bytes. Thus, a matrix of order $N = 10^4$ would require 8×10^8 bytes or 800 MegaBytes (MB) for storage. For large N the linear static solution is dominated by the factorization of **K**. This operation requires approximately $N^3/6$ floating point operation units. Fortunately a very high percentage of the entries of the master stiffness matrix **K** are zero. Such matrices are called *sparse*. There are clever programming techniques that take advantage of sparsity that fit certain patterns. These sparse matrix algorithms do not ignore non-zero terms that are far away from the diagonal. A common sparse matrix algorithm is the *skyline* storage method, which requires $8N^{3/2}$ bytes storage for two-dimensional problems. The factorization of a *skymatrix* requires approximately $0.5N^2$ floating-point operation units for two-dimensional problems.

2 LS-DYNA Cards

The input keyword file (.key) is order independent, case insensitive and multiple blocks of the same keyword can be defined if desired. A keyword deck is started with an asterisk (‘*’) in column one. A dollar sign (‘$’) in column one precedes a comment and causes the input line to be ignored. Each card is 80 character strings long; hence a 10-field card will have 8-character string fields whilst an 8-field card will have 10-character string fields.

2.1 Keyword Format of Input Deck

To flag input as a keyword file
*KEYWORD

To denote end of input file
*END

To specify title of analysis
*TITLE

To control CPU time for the termination of the analysis
*CONTROL_CPU

CPUTIME							

2.2 Output Cards

2.2.1 ASCII Output Files

Default ASCII files are

log file	gives dynamic relaxation convergence factor information; added mass in mass scaling
otf file	gives smallest time steps and corresponding governing element modes; mass of model
message file	gives total CPU time of analysis after completion

The most useful ASCII file is the glstat (cheap in storage), which gives the global data of the analysis. It includes the time step, the kinetic energy, the internal energy, the stonewall energy, the spring and damper energy, hourglass energy, system damping energy, sliding interface energy, external work and total energy. ASCII files such as BNDOUT and NCFORC are in general very, very, very expensive in storage.
*DATABASE_GLSTAT

DT = 0.01							

DT = time interval between outputs

The energy output control card is often required
*CONTROL_ENERGY

Hourglass Energy	**Stonewall Energy**	**Interface Energy**	**Rayleigh Damping Energy**				

Note that KE + IE + Sliding Interface Energy + Hourglass Energy + Stonewall Energy + Damping Energy = Initial KE + External Work; If LHS > RHS, energy introduced artificially for example from numerical instability or sudden detection of artificial penetration through a contact surface. If RHS > LHS, energy absorbed artificially.

To plot material energies
*DATABASE_MATSUM

2.2.2 BINARY Output Files

Dynamic relaxation complete state low frequency graphical processing files (.rlf) (Cheap in storage)
*DATABASE_BINARY_D3DRLF

CYCL = 1							

CYCL = output at every n^{th} convergence check

Transient complete state low frequency graphical processing files (.ptf) (Expensive in storage)
*DATABASE_BINARY_D3PLOT

DT = 0.1	LCDT	BEAM	NPLTC				

Time interval DT = Total Analysis Time / Number of States Desired

Contact force low frequency graphical processing files (.ctf) (Expensive in storage)
*DATABASE_BINARY_INTFOR

DT = 0.1	LCDT						

Time interval = Total Analysis Time / Number of States Required

Time history high frequency signal processing files (.thf) for nodes, beams, shells & solids (Very cheap in storage)
*DATABASE_BINARY_D3THDT

DT = 0.01	LCDT						

*DATABASE_HISTORY_<NODE,BEAM,SHELL,TSHELL,SOLID> (HM: outputblock)

Extra time history high frequency signal processing files (.xtf) (Very cheap in storage)
*DATABASE_BINARY_XTFILE

*DATABASE_EXTENT_BINARY

Independent restart dump files (.dpf) written separately every specified number of cycles (V. expensive in storage)
*DATABASE_BINARY_D3DUMP

CYCL = 99999							

Auxiliary dump files (.adf) written over every specified number of cycles
*DATABASE_BINARY_RUNRSF

CYCL = 99999							

This is good to keep just in case we may need to increase the analysis time

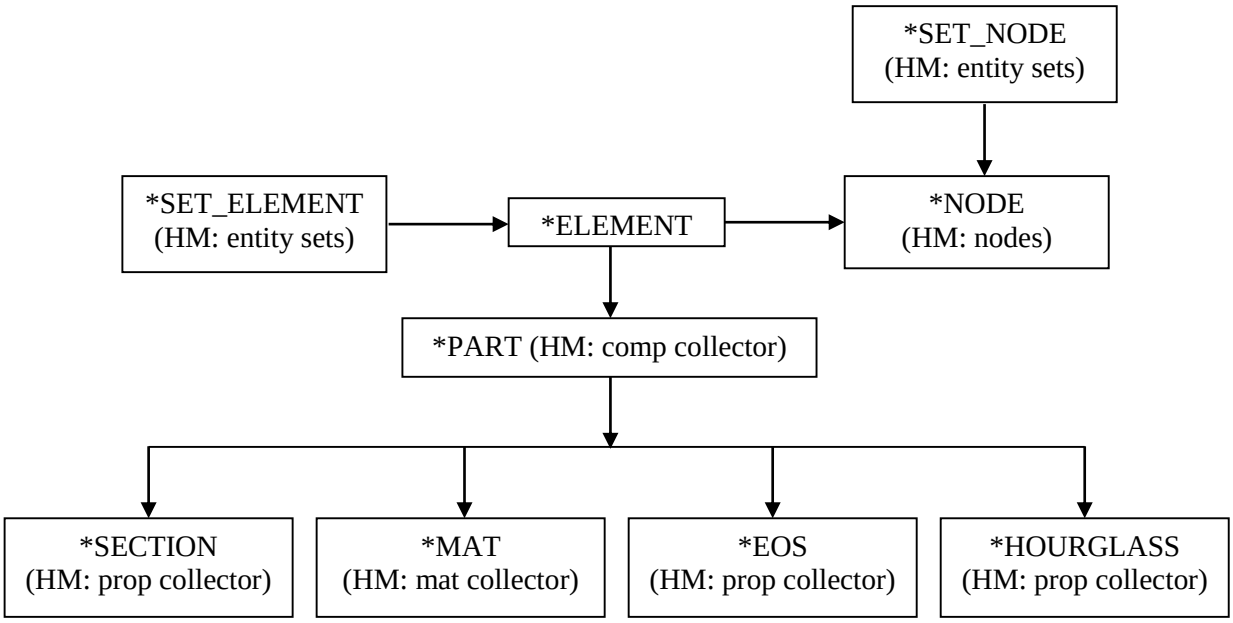
2.3 Node Cards

*NODE (HM: nodes)

NID	**X**	**Y**	**Z**	TC	RC	

A node without any element will be attached a small mass and inertia. Generally, these mass-less nodes do not cause any stability problems except when they interact with the structure.

2.4 Stiffness Element Cards



Basic LS-DYNA element and property card structure

Beam, shell and solid elements in LS-DYNA that use uniformly reduced numerical integration can develop zero energy deformation modes called hourglass modes, having a typically zig-zag appearance to a mesh. These modes are controlled numerically by either an artificial stiffness or viscosity, which resists the formation of these undesirable modes. A brick solid element requires 3 DOFs x 8 Nodes = 24 independent deformation modes. However, with a single integration point, the element provides 3 direct stresses + 3 shear stresses + 6 rigid body modes = 12 independent deformation modes. Hence, 12 deformation modes are not stiff and have to be suppressed. As the mesh gets distorted, the element accuracy is reduced and the timestep required for stability becomes smaller. The analysis may crash. To check the prominence of hourglassing, compute the hourglass energy and ensure that it is less than 10% of the total energy. Usually, it is the solid elements that produce the greatest hourglassing.

To avoid hourglassing:-

(i) avoid subjecting single solid elements to bending exciting hourglass modes

(ii) try to use solid elements with good aspect ratios

(iii) avoid point loads, use pressure loads instead

(iv) avoid single node or line contact

(v) use the viscous or stiffness method to suppress hourglassing; use *CONTROL_HOURGLASS to define the global method of hourglass suppression and/or *HOURGLASS to defined hourglass suppression cards referred to by parts *PART

*CONTROL_HOURGLASS

IHQ	**Hourglass Coefficient**						

*HOURGLASS

HGID	**IHQ**	**QM**	IBQ	Q2	Q1	QB	QW

IHQ=1 Standard LS-DYNA viscous hourglass suppression
IHQ=3 Exact volume solid integration viscous suppression for high velocity models such as seismic analyses
IHQ=5 Exact volume solid integration stiffness suppression for low velocity and small displacement models

(vi) employ fully integrated elements (which are stiff in all the modes of deformation associated with its DOFs) especially where distortions are large, but this very significantly increases computational demand

Single Integration Point Elements	**Fully Integration Elements**
Constant stress through element	Stress varies across element
Less accurate	More accurate
Cheap, as a single 8 integration points element is more expensive than 8 single integration point elements	More expensive
Susceptible to hourglassing as the element is not stiff in all deformation modes, hence not suitable for large distortions	Not susceptible to hourglassing

2.4.1 Spring Element Cards

Discrete spring elements have 1 DOF at each of their 2 nodes. To define a spring element
*ELEMENT_DISCRETE (HM: springs)

EID	**PID**	**N1**	**N2**	**VID**	Scale Factor	PF	Offset Prestress

VID = 0 (local axes) spring acts along axis from N1 to N2
VID ≠ 0 (defined axes) and define an orientation vector *DEFINE_SD_ORIENTATION (HM: vectors) either fixed in space or based on the motion of two nodes

*PART

PID	**SECID**	**MID**	EOSID	HGID	GRAV	ADPOPT	TMID

*SECTION_DISCRETE

SECID	**DRO**	KD	V0	CL	FD	CDL	TDL

DRO = 0 translational spring in which case material cards define force-displacement curve
DRO = 1 torsional spring in which case material cards define moment rotation curve

To define **elastic linear** spring material
*MAT_SPRING_ELASTIC

MID	**Elastic Stiffness**						

To define **elastoplastic** spring material with isotropic hardening
*MAT_SPRING_ELASTOPLASTIC

MID	**Elastic Stiffness**	**Tangent Stiffness**	**Yield**				

To define **elastic nonlinear** spring material with an arbitrary load curve with strain effects if necessary
*MAT_SPRING_NONLINEAR_ELASTIC

MID	**LCID**	LCR					

To define a **inelastic nonlinear** spring material
*MAT_SPRING_INELASTIC

MID	**LCID**	**Unloading Stiffness**	**CTF**				

To define **nonlinear inelastic** spring material with arbitrary load curves for loading and unloading with strain softening, kinematic hardening or isotropic hardening
*MAT_SPRING_GENERAL_NONLINEAR

MID	**LCID Loading**	**LCID Unloading**	**BETA**	**Initial Yield in Tension**	**Initial Yield in Compression**		

2.4.2 Beam Element Cards

Beam elements have 6 DOFs at each of its 2 nodes. The Hughes-Liu formulation has a single integration point along the length, hence models constant bending. The Belytschko-Schwer formulation on the other hand models linear variations in bending moment exactly. The latter beam hence models nodal forces and moments without any internal beam forces or moments exactly. Hence, to model a beam with a point load in the middle exactly, subdivide the beam into two finite beam elements with the common node at the position of the point load. A constant magnitude distributed load on a beam would have a parabolic variation of bending moment. This cannot be modeled exactly, but can be modeled approximately by a number of linear bending moment Belytschko-Schwer beam finite elements. The number of section integration points depends on the quadrature rule QR specified. To model pins at the ends of the beam, we could enable the pin flags within ELEMENT_BEAM.

*ELEMENT_BEAM (HM: bar2)

EID	**PID**	**N1**	**N2**	**N3**	**RT1**	**RR1**	**RT2**	**RR2**	**LOCAL**

A plane through N1, N2 and N3 defines the principal plane r-s of the beam, RT1, RT2, RR1 and RR2 define end release conditions and LOCAL defines whether the coordinate system is global or local.

*PART

PID	**SECID**	**MID**	EOSID	HGID	GRAV	ADPOPT	TMID

*SECTION_BEAM

SECID	**ELFORM**	**Shear Factor**	**QR/IRID**	**Cross Section**	SCOOR		
Area	**Major Iss**	**Minor Itt**	**Polar Irr**	**Shear Area**			

ELFORM = 1 Hughes-Liu (Single integration point along length, i.e. constant bending moment)
= 2 Belytschko-Schwer resultant beam (Linear bending moment distribution)
= 3 Truss
= 4 Belytschko-Schwer full cross section integration
= 6 Discrete beam/cable

QR/IRID = 1.0 One integration point across section
= 2.0 2 x 2 Gauss quadrature rule across section
= 3.0 3 x 3 Gauss quadrature rule across section
= 4.0 3 x 3 Lobatto quadrature rule across section
= 5.0 4 x 4 Gauss quadrature rule across section
= IRID User defined integration rule quadrature rule as specified in *INTEGRATION_BEAM

To set miscellaneous output parameters
*CONTROL_OUTPUT

		Update Beam Reference Node					

Materials to model beams without yield
*MAT_ELASTIC
*MAT_LINEAR_DISCRETE_BEAM

Materials to model beams with lumped plasticity at the ends
*MAT_SEISMIC_BEAM
*MAT_NONLINEAR_DISCRETE_BEAM
*MAT_FORCE_LIMITED

The beam is elastic until the bending moment at the ends reaches the plastic capacity M_P when plastic hinges form. The post-full yield behavior is governed by the moment-rotation curves and axial load-strain curves. Since, the axial collapse load is usually affected by the bending moment in the beam, a different axial collapse load curve needs to be specified for different values of end moments.

Material to model concrete beam yielding
*MAT_CONCRETE_BEAM

Material to model tall shear walls such as chimneys
*MAT_SEISMIC_BEAM
*MAT_CONCRETE_BEAM

To define user-defined integration rules QR/IRID
*INTEGRATION_BEAM

2.4.3 Shell (QUAD4, QUAD8, TRIA3, TRIA6) Element Cards

Shell elements have 5 DOFs at each node (no in-plane rotational DOF). By default it has a single integration point in plane and 2 integration points through the thickness. A fully integrated shell element will have 4 in plane integration points. Shells can be used either as shells (with bending and in-plane stiffness) or just as membranes (only in-plane stiffness). Thick shell elements have 8 or 6 nodes with nodes on top and bottom of the surface like a brick element. They have 3 translational DOFs at each node (no rotational DOFs) again like a brick element. By default it has a single integration point in plane and 3 integration points through the thickness.

To provide controls for computing shell response
*CONTROL_SHELL

Warpage Angle Limit	Sorting of Degenerate Quads	Shell Normal Update	Shell Thickness Change Enabling	Default Shell Formulation Theory	Warping Stiffness Enabling	Plane Stress Plasticity	Projection Method For Warping

To define QUAD4s or TRIA3s,

*ELEMENT_SHELL_<BLANK, THICKNESS, BETA> (HM: tria3, quad4)

EID	**PID**	**N1**	**N2**	**N3**	**N4**		

The THICKNESS option overwrites that specified in *SECTION_SHELL.

*PART

PID	**SECID**	**MID**	EOSID	HGID	GRAV	ADAPTIVE	TMID

*SECTION_SHELL_<BLANK, ALE> (HM: prop collector)

SECID	**ELFORM**	**Shear Factor**	**Number of Through Thickness Integration Points**	Printout Option	Integration Rule ID QR/IRID	ICOMP	SETYP
T1	**T2**	**T3**	**T4**	NLOC			

Shell elements in ascending form of accuracy and cost

	Shell Element Formulation	**ELFORM**	**Accuracy**	**Cost**
One Point Quadrature	Belytschko-Tsay (BT)	2	Flat geometry hence no warpage stiffness	1
	Belytschko-Tsay with warping stiffness (BTW)			1.07
	Belytschko-Leviathan (BL)			1.25
	Belytschko, Wong, Chiang (BWC)	10		1.28
	Co-rotational Hughes-Liu (CHL)			1.49
	Hughes-Liu (HL)			2.45
Fully Integrated Elements	Fully Integrated Belytschko-Tsay (FBT)			2.80
	Co-rotational Fully Integrated Hughes-Liu (CFHL)			8.84
	Fully Integrated Hughes-Liu (FHL)			20.01

ELFORM element formulation theory

1: Hughes-Liu (Accurate, expensive, non-planar geometry thus warpage modeled)
2: Belytschko-Tsay (Default, less accurate, cheap)
4: C_0 triangular shell
5: Belytschko-Tsay membrane
9: Fully integrated Belytschko-Tsay membrane
10: Belytschko, Wong, Chiang (Inexpensive warping stiffness for Belytschko-Tsay)
12: Plane stress
13: Plane strain
16: Fully integrated version of Belytschko, Wong, Chiang

To define user-defined integration rules QR/IRID, use *INTEGRATION_SHELL and to define viscous or stiffness hourglass control for under-integrated elements, use *HOURGLASS.

To model non-yielding **elastic, linear, isotropic** materials
*MAT_ELASTIC

The **inelastic, nonlinear, isotropic** material model capable of defining either isotropic or kinematic hardening with **strain-rate** effects (for impulsive **blast** or **projectile crash** analysis) with the **Cowper-Symonds** rules
*MAT_PLASTIC_KINEMATIC
*MAT_PIECEWISE_LINEAR_PLASTICITY
*MAT_PLASTICITY_WITH_DAMAGE
*MAT_STRAIN_RATE_DEPENDENT_PLASTICITY

To model reinforced concrete slabs yielding
*MAT_RC_SHEARWALL

To model reinforced concrete squat shear walls
*MAT_RC_SHEARWALL
*MAT_SPRING_SQUAT_SHEARWALL

To define an eight node thick shell element
*ELEMENT_TSHELL
*SECTION_TSHELL
*INTEGRATION_SHELL
*HOURGLASS

2.4.4 Solid (Brick, Wedge, Tetrahedral) Element Cards

Solid elements have 3 translational DOFs at each node (no rotational DOFs). By default it has a single integration point, hence no bending resistance. A fully integrated HEX8 solid will have 8 integration points. The basic solid finite element is the brick. The wedge and tetrahedral elements are degenerate brick elements, i.e. the wedge is a brick element with one of the faces collapsed into an edge, and the tetrahedral element is a wedge with the edge collapsed into a point. The more degenerate an element, the less well it is likely to perform. Tetrahedral elements should therefore always be avoided.

*ELEMENT_SOLID_<BLANK, ORTHO> (HM: tetra4, penta6, hex8)

EID	**PID**	**N1**	**N2**	**N3**	**N4**	**N5**	**N6**	**N7**	**N8**

*PART

PID	**SECID**	**MID**	EOSID	HGID	GRAV	ADAPTIVE	TMID

*SECTION_SOLID_<BLANK, ALE>

SECID	**ELFORM**	AET					

ELFORM element formulation theory
- 1: Constant stress solid element
- 2: Fully integrated S/R solid

To model non-yielding **elastic, linear, isotropic** materials
*MAT_ELASTIC

To model linear orthotropic elastic materials
*MAT_ORTHOTROPIC_ELASTIC
*MAT_ORTHOTROPIC_THERMAL
*MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC

The **inelastic, nonlinear, isotropic** basic bilinear elastoplastic material model with only isotropic hardening
*MAT_ISOTROPIC_ELASTIC_PLASTIC

The **inelastic, nonlinear, isotropic** material model capable of defining either isotropic or kinematic hardening with **strain-rate** effects (for impulsive **blast** or **projectile crash** analysis) with the **Cowper-Symonds** rules
*MAT_PLASTIC_KINEMATIC
*MAT_PIECEWISE_LINEAR_PLASTICITY
*MAT_PLASTICITY_WITH_DAMAGE
*MAT_STRAIN_RATE_DEPENDENT_PLASTICITY

The hardening rule depends of β which the user can select as $\beta = 0$ for kinematic hardening where the diameter of the yield surface remains unchanged but the center can move and $\beta = 0$ for isotropic hardening where the center of the yield surface remains fixed and the diameter increases. A β value between these give a combination of the kinematic and isotropic hardening rules. This value of β affects the yield stress when the loading is reversed, e.g. what is the yield stress for compressive loading following yield in tension. Be sure to convert to true stress and true strain. Beyond the Ultimate Tensile Strength, the experimental curve is hard to interpret because all the deformation is concentrated in the neck region; it is recommended that the input curve be made flatter beyond UTS than the experimental curve to allow for the uncertainties. The failure strain should generally be set based on a reduction in cross-sectional area when the tensile test specimen fails. The unloading curve (hence inelastic material model) has the same stiffness as the initial elastic stiffness.

Variants of the bilinear theme include
*MAT_ELASTIC_PLASTIC_THERMAL
which allows properties to vary with temperature

*MAT_ORIENTED_CRACK
which includes fracture at predefined tensile stress

*MAT_POWER_LAW_PLASTICITY
where hardening parameters may exist in terms of an 'n' value

To model concrete with both volumetric crushing and strain softening in shear (to simulate cracking)
*MAT_SOIL_CONCRETE

To model plastics
*MAT_VISCOELASTIC

To model rubber
*MAT_MOONEY_RIVLIN_RUBBER
Mooney-Rivlin rubber constants for rubber with Shore Hardness of 60 (ihrd 60) are presented.

Material Parameter	**Property (mm, tonnes, N, sec)**	**Property (m, kg, N, sec)**
Density	0.9e-09 tonnes/mm³	900 kg/m³
Poisson's Ratio	0.49	0.49
Parameter A	0.474 N/mm²	0.474e+06 N/m²
Parameter B	0.118 N/mm²	0.118e+06 N/m²

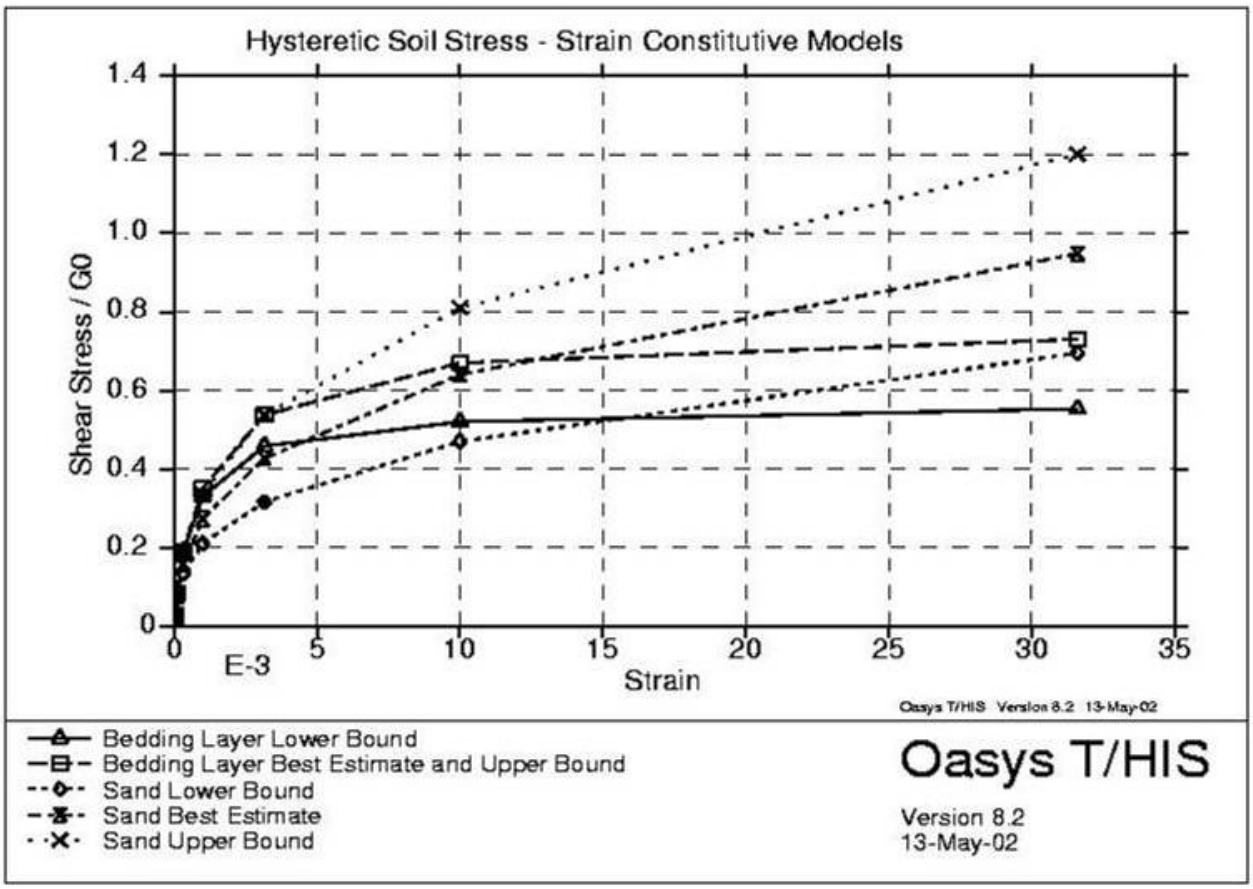
To model fabrics
*MAT_FABRIC

To model glass
*MAT_ELASTIC_WITH_VISCOSITY
*MAT_LAMINATED_GLASS

To model **isotropic, inelastic (hence exhibiting hysteretic damping)** and **non-linear stiffness degrading (with strain)** soil elements under cyclic loading such as during earthquakes
*MAT_HYSTERETIC_SOIL
*DEFINE_CURVE

This is simply a **non-linear** material models where the material stress-strain curve is dependent upon the strain. In a **linear** material model, $G = E/[2(1+\nu)]$. But in a non-linear material model, G is dependent upon the level of strain. The material model itself is of course still **isotropic**, i.e. same property in all directions, but non-linear. The non-linear soil material model simulates directly the non-linear shear stress-shear strain behaviour of soil loaded by strong cyclic seismic shear waves to large shear strain amplitudes. The shear stress - shear strain curves defining the soil elements' constitutive models are derived from the G/G_0 degradation curves. Let us look at typical examples on a project. We shall investigate the Sakhalin Tank Loading Unit (TLU), an offshore gravity base structure near Sakhalin Island, Russia weighing 82.7MN with fundamental frequency of about 1.0Hz and associated modal mass of about 9.7MN. At the site of the TLU, the sandy soil overlying the bedrock is approximately 2 m in depth. On the seabed surface, a leveling course (bedding layer) of gravel of 0.5 m thick is to be placed. At 2.5 m below the top surface of the soil and further downwards, the measured shear wave velocity is approximately 700.0 m/sec., indicating that the material below this level is sufficiently stiff to be considered as bedrock. Values of the small strain shear modulus, G_0, for the five layers of soil elements are listed and the figure shows the normalised (to G_0) shear stress - shear strain curves of the gravel bedding layer and the sand.

Soil Layer	Depth (m)	Soil Type	Density (kg/m^3)	G_0 (MN/m^2)		
				Lower Bound	Best Estimate	Upper Bound
1	0.5 to 0	Gravel Bedding Layer	1800	80.8	82.9	85.2
2	0 to –0.5	Sand	1800	97.0	99.0	102.0
3	-0.5 to -1.0	Sand	1800	93.1	98.2	105.3
4	-1.0 to -1.5	Sand	1800	89.2	96.0	105.4
5	-1.5 to -2.0	Sand	1800	86.0	94.0	105.5



While the horizontal response of the site soil to vertically propagating seismic waves depends on the non-linear deviatoric (shear) behaviour of the soil, the vertical response is determined by the volumetric behaviour, quantified by the bulk modulus of the soil material, which has to be entered into the material model. This is related to the shear modulus by

$$G = \frac{E}{2(1+\nu)}$$

Damping in the form of **hysteretic damping** within the model is explicitly accounted for due to the **inelastic** nature of the material model.

To define **viscous or stiffness hourglass control** for under-integrated elements, use *HOURGLASS.

2.5 Mass and Inertia Element Cards

Lumped mass elements have no DOF but add user specified masses to their single nodes.
*ELEMENT_MASS (HM: masses)

EID	**NID**	**MASS VALUE**						

To define a lumped inertia element assigned to a node
*ELEMENT_INERTIA

EID	**NID**	**CSID**							
IXX	**IXY**	**IXZ**	**IYY**	**IYZ**	**IZZ**				

CSID = 0 (global axes)
CSID = <integer> (defined axes) and declare a *DEFINE_COORDINATE_NODES (from three defined nodes), *DEFINE_COORDINATE_SYSTEM (from three points) or *DEFINE_COORDINATE_VECTOR (from two vectors)

2.6 Damping Cards

To define mass weighted nodal damping that applies globally to the nodes of deformable bodies and to the mass center of the rigid bodies
*DAMPING_GLOBAL

To define mass weighted damping by parts of deformable or rigid elements
*DAMPING_PART_MASS
Higher frequency motions are under-damped and lower frequency motions are over-damped.

To define Rayleigh stiffness damping coefficient by parts
*DAMPING_PART_STIFFNESS
Higher frequency motions are over-damped and lower frequency motions are under-damped.

Other damping cards are
*DAMPING_FREQUENCY_RANGE
*DAMPING_MODAL
*MAT_SEISMIC_ISOLATOR

To define a energy dissipating viscoelastic damper with damping proportional to the velocity
*ELEMENT_DISCRETE (HM: springs)
*SECTION_DISCRETE
*PART
*DEFINE_SD_ORIENTATION (HM: vectors)
*MAT_DAMPER_VISCOUS
*MAT_DAMPER_NONLINEAR_VISCOUS

To define a tuned-mass dampers
*ELEMENT_MASS
*ELEMENT_DISCRETE (HM: springs)
*SECTION_DISCRETE
*PART
*DEFINE_SD_ORIENTATION (HM: vectors)
*MAT_DAMPER_VISCOUS
*MAT_SPRING_ELASTIC

To model hysteretic dampers
*MAT_SPRING_ELASTOPLASTIC
or
*MAT_SEISMIC BEAM

To model viscoelastic bearing isolation devices
*ELEMENT_DISCRETE
*MAT_SPRING_ELASTIC
*MAT_SPRING_ELASTOPLASTIC

To model friction pendulum isolation devices
*ELEMENT_DISCRETE
*MAT_SPRING_ELASTIC
*MAT_SPRING_ELASTOPLASTIC

2.7 Rigid Element Cards and Structural Internal Constraints

Only one amongst the cards of restraints, rigid bodies, prescribed motions, nodal constraints, stonewall, tied interface and spotwelds should apply to any one DOF in the structure.

2.7.1 Rigid Body

2.7.1.1 Formulation of Rigid Body Joint

To switch the explicit rigid body joint treatment to an implicit formulation, which uses Lagrange multipliers to impose prescribed kinematic boundary conditions and joint constraints
*CONTROL_RIGID

Lagrange Multiplier Flag	Joint Stiffness Formulation						

2.7.1.2 Rigid Body With Mass And Inertia Properties From A Set Of Nodes

*CONSTRAINED_NODAL_RIGID_BODY_<BLANK, INERTIA> (HM: rigids → RgdBody)

NSID	**CID**						

To define a node set, use *SET_NODE. If the coordinate system ID, CSID ≠ 0, declare a *DEFINE_COORDINATE_NODES (from three defined nodes), *DEFINE_COORDINATE_SYSTEM (from three points) or *DEFINE_COORDINATE_VECTOR (from two vectors) to define a local coordinate system. Use INERTIA option to in define inertia properties instead of calculating them.

2.7.1.3 Rigid Body With Mass And Inertia Properties From An Element

*ELEMENT_BEAM
*ELEMENT_SHELL_<BLANK, THICKNESS, BETA> (HM: tria3, quad4)
*ELEMENT_SOLID_<BLANK, ORTHO> (HM: tetra4, penta6, hex8)

*PART_<BLANK, INERTIA>

PID	**SECID**	**MID**	EOSID	HGID	GRAV	ADAPTIVE	TMID

Use INERTIA option to in define inertia properties instead of calculating them.

*SECTION_BEAM
*SECTION_SHELL
*SECTION_SOLID

*MAT_RIGID

MID	**Mass Density**	**Young's Modulus**	**Poisson's Ratio**	N	COUPLE	M	ALIAS
Constraints Direction CMO	**Translational Constraints**	**Rotational Constraints**					

CMO = 1.0 (global coordinate system)
CMO = 0.0 (no constraints)
CMO = -1.0, and declare a *DEFINE_COORDINATE_NODES (from three defined nodes), *DEFINE_COORDINATE_SYSTEM (from three points) or *DEFINE_COORDINATE_VECTOR (from two vectors) to define a local coordinate system.

The Young's Modulus and Poisson's Ratio is required for contact surface calculations. Note that different rigid bodies should reference different parts, as each part will have one mass and inertia property and will act as one rigid body even if all the elements of the part is not contiguous.

2.7.1.4 Extra Nodes on Rigid Bodies

Extra nodes are added onto a rigid body so that

(i) the response at a particular location can be recorded, or
(ii) a load can be applied at a particular location
(iii) joints can be attached at these nodes to join rigid bodies together
(iv) springs (stiffness) or lumped mass in a particular direction can be attached

The coordinates of the extra nodes are updated based on the motion of the rigid body.

*CONSTRAINED_EXTRA_NODES_<NODE, SET>

PID	**NID or NSID**						

PID defines the rigid body and NID or NSID defines the extra nodes to be added to the rigid body.

*SET_NODE_LIST

NSID	DA1	DA2	DA3	DA4			
NID	**NID**	**NID**	**NID**	**NID**	**NID**	**NID**	**NID**
NID	**NID**	**NID**	**NID**	…	**…**	**…**	**…**

2.7.1.5 Merging Rigid Bodies

To merge rigid bodies
*CONSTRAINED_RIGID_BODIES
*PART for master rigid body part
*PART for slave rigid body part

To define a joint between two rigid bodies (as tied nodes cannot be used on rigid bodies),
*CONSTRAINED_JOINT_<...OPTIONS...>_<LOCAL> (HM: joints)
*CONSTRAINED_JOINT_STIFFNESS_<GENERALIZED, FLEXION-TORSION>

The spherical joint is used to model pinned connections, the revolute a hinge connection and the cylindrical a hinge with sliding along its axis.

2.7.2 Rigidwall

To define a rigidwall
*RIGIDWALL_GEOMETRIC
*RIGIDWALL_PLANAR

This is a simplification of a contact surface which models a plane through which nodes cannot pass. This is used to model hard targets such as rigid walls defined by a vector normal to the plane of the rigid wall pointing out of the wall towards the model. Only nodes on the outside of the model need to be chosen as slave nodes.

2.7.3 Constraints Between DOFs

2.7.3.1 Interpolation Constraint

The motion of a single dependent node is interpolated from the motion of a set of independent nodes. This is useful for: -

(i) the load distribution of a translational load or moment, applied to the dependent node to the surrounding independent nodes
(ii) the modelling of shell-brick or beam-brick interfaces

*CONSTRAINED_INTERPOLATION

ID	**Dependent NID**	**Dependent DOF**					
Independent NID	**Independent DOF**	**X Weight Factor**	**Y Weight Factor**	**Z Weight Factor**	**XX Weight Factor**	**YY Weight Factor**	**ZZ Weight Factor**
Independent NID	**Independent DOF**	**X Weight Factor**	**Y Weight Factor**	**Z Weight Factor**	**XX Weight Factor**	**YY Weight Factor**	**ZZ Weight Factor**
…	**…**	**…**	**…**	**…**	**…**	**…**	**…**

To tie a beam to a solid with no common nodes for instance, the dependent node is a beam end node with dependent DOF 123456, and the independent nodes are a few nodes on the solid with DOF 123. If the beam node is shared, then the dependent DOF will be 123. To distribute a load,

To define nodal constraint sets for translational motion (no rotational coupling) in global coordinates
*CONSTRAINED_NODE_SET (HM: rigids → ConNode)
*SET_NODE

To define linear constraint equations between displacements / rotations in global coordinates
*CONSTRAINED_LINEAR (HM: equation)

To constrain particular DOFs in order to model pins (tied in translation), rigid planes (tied vertically) etc, we can use tied nodes
*CONTRAINED_TIED_NODES_FAILURE

2.7.4 Global Restraints

To define a global boundary constraint plane

*CONSTRAINED_GLOBAL

Translational Constraint	**Rotational Constraint**	**Direction of Normal**	**X Offset**	**Y Offset**	**Z Offset**		

2.7.5 Welds and Rivets

2.7.5.1 Generalized Spot Weld

*CONSTRAINED_GENERALIZED_WELD_<SPOT,FILLET,BUTT,CROSS_FILLET,COMBINED>
*SET_NODE
*DEFINE_COORDINATE_<NODES,SYSTEM,VECTOR> (HM: systems) if nodes coincident

The generalized spotweld option includes a ductile failure in addition to brittle failure of the simple spotweld and allows for more than 2 nodes in the spotweld and for the nodes to be coincident.

2.7.5.2 Generalized Fillet Weld

*CONSTRAINED_GENERALIZED_WELD_FILLET

2.7.5.3 Generalized Butt Weld

*CONSTRAINED_GENERALIZED_WELD_BUTT

2.7.5.4 Generalized Cross-Fillet Weld

*CONSTRAINED_GENERALIZED_WELD_CROSS_FILLET

2.7.5.5 Generalized Combined Weld

*CONSTRAINED_GENERALIZED_WELD_COMBINED

2.7.5.6 Mass-Less Spotweld

*CONSTRAINED_SPOTWELD
The spotweld is defined by 2 non-coincident nodes and a brittle failure load.

2.7.5.7 Mass-Less Rivet

To define a mass less rivet
*CONSTRAINED_RIVET

2.7.6 Concrete Rebars

To model reinforcement bars in concrete
- *CONSTRAINED_LAGRANGE_IN_SOLID
- *PART or *SET_PART or *SET_SEGMENT for slave elements
- *PART or *SET_PART for master solid elements

2.8 Boundary Conditions Cards

2.8.1 Single Point Constraints SPCs

To apply essential boundary conditions, constraints of degrees of freedom can be applied on *NODE cards, but this not recommended and is only available for backward compatibility. Instead, it is better to explicitly define single point constraints

*BOUNDARY_SPC_<NODE, SET> (HM: constraints → BoundSPC)

NID or NSID	**CID**	**DOFX**	**DOFY**	**DOFZ**	**DOFRX**	**DOFRY**	**DOFRZ**
NID	**NID**	**NID**	**NID**	**NID**	**NID**	**NID**	**NID**
NID	**NID**	**NID**	**NID**	**NID**	**NID**	**NID**	**NID**
NID	**NID**	**NID**	**NID**	…	**…**	**…**	**…**

Rigid bodies should not be subjected to SPCs. Instead constraints on rigid bodies are applied on *MAT_RIGID.

*SET_NODE_LIST

NSID	DA1	DA2	DA3	DA4			
NID	**NID**	**NID**	**NID**	**NID**	**NID**	**NID**	**NID**
NID	**NID**	**NID**	**NID**	…	**…**	**…**	**…**

If the coordinate system ID is not global (CID ≠ 0) then it is necessary to define a local coordinate system i.e. declare a *DEFINE_COORDINATE_NODES (from three defined nodes), *DEFINE_COORDINATE_SYSTEM (from three points) or *DEFINE_COORDINATE_VECTOR (from two vectors) (HM: systems).

2.8.2 Non-Reflecting Boundaries

To define a non-reflecting boundary for infinite solid element modelling such as infinite soil

* BOUNDARY_NON_REFLECTING

Segment Set ID	**Dilatation Waves Activation**	**Shear Waves Activation**					

In seismic analyses, it is sufficient to activate the shear waves flag only. Segments are equivalent to triangular or quadrilateral element faces on the boundary defined by listing corner nodes clockwise or anticlockwise.

*SET_SEGMENT

ID	DA1	DA2	DA3	DA4			
NID	**NID**	**NID**	**NID**	**NID**	**NID**	**NID**	**NID**
NID	**NID**	**NID**	**NID**	**NID**	**NID**	**NID**	**NID**
NID	**NID**	**NID**	**NID**	**…**	**…**	**…**	**…**

2.8.3 Contact Cards

To control defaults for computation with contact surfaces
*CONTROL_CONTACT

The surface definition for contact is made up of segments on the shell or solid element surfaces.

2.8.3.1 Rigidwall Contact

If the node penetrates the rigidwall, the normal velocity and acceleration is set to zero.

2.8.3.2 Surface To Surface Contact

This is the most general contact definition, which includes penetration, separation and friction algorithms. It works by the penalty force method in which the penetration of one surface to another is resisted by a force. Contact is only between the defined slave and master surfaces. Make the smaller surface the slave and the larger surface the master.
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TITLE (HM: interface)
*CONTACT_CONSTRAINT_SURFACE_TO_SURFACE_TITLE

The AUTOMATIC method is a penalty method (see **Section 1.3.23**). The soft option flag can be set if penetration is observed. The CONSTRAINT method is a Lagrange method and should be used if the soft option in the usual AUTOMATIC method is still not good enough to prevent penetrations.

2.8.3.3 Single Surface Contact

This is used when a surface folds onto itself or where there is no separation into master and slave surfaces. It provides a generalized contact capability, which does not require identification of slave and master surfaces, i.e. any node on surface can contact any element. Hence the search logic is more complicated and more expensive than the surface-to-surface definition.
*CONTACT_AUTOMATIC_SINGLE_SURFACE_TITLE

2.8.3.4 Nodes to Surface Contact

This is a one-way contact surface treatment, the slave side is the nodes and the master side is the surface. It is used to model situations such as beam impacting a plate where the beam is not a ‘surface’. But the logic is similar to that for sliding with separation.
*CONTACT_AUTOMATIC_NODES_TO_SURFACE_TITLE
*CONTACT_CONSTRAINT_NODES_TO_SURFACE_TITLE

2.8.3.5 Eroding Contacts

This is used in situations where the elements on the contact surface fail and have to be deleted, hence modifying the contact surface.

2.8.3.6 Tied Interface

*CONTACT_TIED_SURFACE_TO_SURFACE
*CONTACT_TIED_NODES_TO_SURFACE
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE

This is used to glue different meshes together, hence providing a simple efficient way of mesh refinement.
However, this can produce internal reflection of stress waves (although so can conventional mesh refinement).

2.9 Restart Capabilities

*RESTART

To allow parts that is defined as deformable to rigid at the start on an analysis
*DEFORMABLE_TO_RIGID_<AUTOMATIC,INERTIA>
*RIGID_TO_DEFORMABLE

Computation cost can be significantly reduced by switching deformable parts that are not expected to deform to rigid parts. The analysis can be stopped and restarted with the parts switched back to deformable just when they are expected to deform.

*STRESS_INITIALIZATION

BIBLIOGRAPHY

1. NAFEMS. *A Finite Element Primer*. NAFEMS Ltd., Great Britain, 1992.
2. ADAMS, Vince. *Building Better Products With Finite Element Analysis*. High Mountain Press, 1999.
3. SPYRAKOS, Constantine. *Finite Element Modelling In Engineering Practice*. West Virginia University Press, United States of America, 1994.
4. HITCHINGS, Dennis. *A Finite Element Dynamics Primer*. NAFEMS Ltd., Great Britain, September 1991.
5. MACNEAL-SCHWENDLER CORP. *MSC.NASTRAN Practical Finite Element Modelling Techniques Seminar Notes*.