SIEMENS

# NX Nastran 10.2 Release Guide

# Contents

# Proprietary & Restricted Rights Notice

is specifically optimized for your machine). If you want to conserve bandwidth and you want to install the required libraries yourself, download the lean distributions. The zip and tgz files are identical, except that on Linux, Unix, and MacOS, unpacking the tgz file ensures that the configure script is marked as executable (unpack with tar zxvpf), otherwise you will have to change its permissions manually.

# Chapter 1: Mechanical and Thermal Strain for Composites

## Mechanical and Thermal Strain for Composites

A shell composite is defined with a PCOMP or PCOMPG property entry which is referenced by a CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, or CTRIAR element definition.

Previously, when requesting output for a shell composite, there was no way to request only mechanical strain or only thermal strain. Only the combined mechanical and thermal strain could be requested using the STRAIN case control command.

Now in SOLs 101 and 106 for a shell composite, you can request mechanical strain using the updated ELSTRN case control command, thermal strain using the updated THSTRN case control command, and the combined mechanical and thermal strain using the existing STRAIN case control command.

NX Nastran automatically computes equivalent PSHELL and MAT2 entries from the composite definition. The equivalent PSHELL and MAT2 entries represent a smeared representation of the composite. The parameter NOCOMPS determines if the strain requested with any of the ELSTRN, THSTRN, or STRAIN commands is computed on the composite ply layers, on the equivalent PSHELL, or on both. The following table summarizes the NOCOMP options.

| Value of the parameter NOCOMPS | Is strain computed for the composite? | Is strain computed for the equivalent element? |
|---|---|---|
| 1 (default) | YES | NO |
| 0 | YES | YES |
| -1 | NO | YES |

For example, if the THSTRN case control command, the STRAIN case control command, and NOCOMPS=0 are defined in SOL 101, the thermal strain and the combined strain will be computed for both the composite, and for the equlivalent shell element.

See the updated *Classical Lamination Theory for Shell Composites (PCOMP,PCOMPG)*.

See the ELSTRN, THSTRN, and STRAIN case control commands.

## ELSTRN

**Requests elastic strain output at grid points on elements for SOL 401. Also requests elastic strain output at grid points on composite shell elements for SOLs 101 and 106.**

Requests elastic strain at grid points on elements.

**FORMAT:**

$$\text{ELSTRN}\left[\left(\begin{bmatrix}\text{STRCUR}\\ \text{FIBER}\end{bmatrix}, \begin{bmatrix}\text{CENTER}\\ \text{CORNER or BILIN}\\ \text{SGAGE}\\ \text{CUBIC}\end{bmatrix}, \text{PRINT, PUNCH, PLOT}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

**EXAMPLES:**

```
ELSTRN=ALL
ELSTRN(PRINT,PUNCH)=17
The following example is only supported in SOLs 101 and 106:
ELSTRN(STRCUR,CORNER,PRINT) = ALL
```

**DESCRIBERS FOR SOL 401:**

| Describer | Meaning |
|---|---|
| PRINT | Compute and write output to the print file (f06). (Default) |
| PUNCH | Compute and write output to the punch file (pch). |
| PLOT | Compute output. |
| ALL | Requests output for all grid points. |
| n | Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0) |
| NONE | Output is not computed. |

**DESCRIBERS FOR SOLS 101 AND 106:**

| Describer | Meaning |
|---|---|
| PRINT | Compute and write output to the print file (f06). (Default) |
| PUNCH | Compute and write output to the punch file (pch). |
| PLOT | Compute output. |
| ALL | Requests output for all grid points. |
| n | Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0) |
| NONE | Output is not computed. |
| STRCUR | Elastic strain at the reference plane and curvatures is output for plate elements. |
| FIBER | Elastic strain at locations Z1, Z2 is computed for plate elements. |
| CENTER | Outputs elastic strains at the center of the element only. See Remark 2 for SOLs 101 and 106. |
| CORNER or BILIN | Outputs elastic strains at the center and corner grid points only using extrapolation. See Remark 3 for SOLs 101 and 106. |
| SGAGE | Outputs elastic strains at the center and corner grid points using strain gage approach. See Remark 4 for SOLs 101 and 106. |
| CUBIC | Outputs elastic strains at the center and corner grid points using cubic bending correction. See Remark 5 for SOLs 101 and 106. |

**REMARKS FOR SOL 401:**

1. Only supported in a static subcase for SOL 401.
2. Both PRINT and PUNCH may be requested.
3. OP2 file output requires PARAM,POST,-1 or PARAM,POST,-2.

**REMARKS FOR SOLS 101 AND 106:**

1. The STRCUR option is default for shell elements that reference a PCOMP or PCOMPG entry. Any FIBER option on STRAIN or ELSTRN or THSTRN will override STRCUR option for all STAIN and ELSTRN and THSTRN.

2. For the CENTER option, elastic strains are output at the center of the element for CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry. For CQUAD4, CQUADR, CTRIA3, CTRIAR, CTRIA6, and CQUAD8 elements that reference a PCOMP or PCOMPG entry, ply strains are always reported at the center of the element.

3. For the CORNER (or BILIN) option, elastic strains are output at the center and grid points for CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry.

4. For the SGAGE option, elastic strains are output at the center and grid points for CQUAD4 elements that reference a PCOMP or PCOMPG entry.

5. For the CUBIC option, elastic strains are output at the center and grid points for CQUAD4 and CQUADR elements that reference a PCOMP or PCOMPG entry.

6. The CENTER/CORNER(BILIN)/SGAGE/CUBIC option on STRAIN card overrides the corresponding option on ELSTRN and THSTRN card.

## THSTRN

**Requests thermal strain output at grid points on elements for SOL 401. Also requests thermal strain output at grid points on composite shell elements for SOLs 101 and 106.**

Requests thermal strain at grid points on elements.

**FORMAT:**

$$\text{THSTRN}\left[\left(\begin{bmatrix}\text{STRCUR}\\ \text{FIBER}\end{bmatrix}, \begin{bmatrix}\text{CENTER}\\ \text{CORNER or BILIN}\\ \text{SGAGE}\\ \text{CUBIC}\end{bmatrix}, \text{PRINT}, \text{PUNCH}, \text{PLOT}\right)\right] = \begin{Bmatrix}\text{ALL}\\ \text{n}\\ \text{NONE}\end{Bmatrix}$$

**EXAMPLES:**

```
THSTRN=ALL
THSTRN(PRINT,PUNCH)=17
The following example is only supported in SOLs 101 and 106:
THSTRN(STRCUR,CORNER,PRINT) = ALL
```

**DESCRIBERS FOR SOL 401:**

| Describer | Meaning |
|---|---|
| PRINT | Compute and write output to the print file (f06). (Default) |
| PUNCH | Compute and write output to the punch file (pch). |
| PLOT | Compute output. |
| ALL | Requests output for all grid points. |
| n | Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0) |
| NONE | Output is not computed. |

**DESCRIBERS FOR SOLS 101 AND 106:**

| Describer | Meaning |
|---|---|
| PRINT | Compute and write output to the print file (f06). (Default) |
| PUNCH | Compute and write output to the punch file (pch). |
| PLOT | Compute output. |
| ALL | Requests output for all grid points. |
| n | Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the output. (Integer>0) |
| NONE | Output is not computed. |
| STRCUR | Elastic strain at the reference plane and curvatures is output for plate elements. |
| FIBER | Elastic strain at locations Z1, Z2 is computed for plate elements. |
| CENTER | Outputs elastic strains at the center of the element only. See Remark 2 for SOLs 101 and 106. |
| CORNER or BILIN | Outputs elastic strains at the center and corner grid points only using extrapolation. See Remark 3 for SOLs 101 and 106. |
| SGAGE | Outputs elastic strains at the center and corner grid points using strain gage approach. See Remark 4 for SOLs 101 and 106. |
| CUBIC | Outputs elastic strains at the center and corner grid points using cubic bending correction. See Remark 5 for SOLs 101 and 106. |

**REMARKS FOR SOL 401:**

1. Only supported in a static subcase for SOL 401.
2. Both PRINT and PUNCH may be requested.
3. OP2 file output requires PARAM,POST,-1 or PARAM,POST,-2.

**REMARKS FOR SOLS 101 AND 106:**

1. The STRCUR option is default for shell elements that reference a PCOMP or PCOMPG entry. Any FIBER option on STRAIN or ELSTRN or THSTRN will override STRCUR option for all STAIN and ELSTRN and THSTRN.

2. For the CENTER option, thermal strains are output at the center of the element for CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry. For CQUAD4, CQUADR, CTRIA3, CTRIAR, CTRIA6, and CQUAD8 elements that reference a PCOMP or PCOMPG entry, ply strains are always reported at the center of the element.

3. For the CORNER (or BILIN) option, thermal strains are output at the center and grid points for CQUAD4, CQUADR, and CTRIAR elements that reference a PCOMP or PCOMPG entry.

4. For the SGAGE option, thermal strains are output at the center and grid points for CQUAD4 elements that reference a PCOMP or PCOMPG entry.

5. For the CUBIC option, thermal strains are output at the center and grid points for CQUAD4 and CQUADR elements that reference a PCOMP or PCOMPG entry.

6. The CENTER/CORNER(BILIN)/SGAGE/CUBIC option on STRAIN card overrides the corresponding option on ELSTRN and THSTRN card.

## STRAIN

### Element Strain Output Request

Requests the form and type of strain output.

**FORMAT:**

$$\text{STRAIN}\left[\left(\begin{bmatrix}\text{SORT1}\\ \text{SORT2}\end{bmatrix}, \begin{bmatrix}\text{PRINT, PUNCH}\\ \text{PLOT}\end{bmatrix}, \begin{bmatrix}\text{REAL or IMAG}\\ \text{PHASE}\end{bmatrix}, \begin{bmatrix}\text{VON MISES}\\ \text{MAXS or SHEAR}\end{bmatrix},\right.\right.$$

$$\begin{bmatrix}\text{STRCUR}\\ \text{FIBER}\end{bmatrix}, \begin{bmatrix}\text{CENTER}\\ \text{CORNER or BILIN}\\ \text{SGAGE}\\ \text{CUBIC}\end{bmatrix}, \begin{bmatrix}\text{PSDF}\\ \text{ATOC}\\ \text{CRMS}\\ \text{RMS}\\ \text{RALL}\end{bmatrix}, \begin{bmatrix}\text{RPRINT}\\ \text{NORPRINT}\end{bmatrix}, [\text{RPUNCH}],$$

$$\left.\left.\begin{bmatrix}\text{CPLYMID}\\ \text{CPLYBT}\\ \text{CPLYBMT}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{ALL}\\ n\\ \text{NONE}\end{Bmatrix}$$

**EXAMPLES:**

```
STRAIN=5
STRAIN(CORNER)=ALL
STRAIN(PRINT,PHASE)=15
STRAIN(PLOT)=ALL
```

**DESCRIBERS:**

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

| Describer | Meaning |
|---|---|
| SORT2 | Output will be presented as a tabular listing of frequency or time for each element. |
| PRINT | The printer will be the output medium. |
| PUNCH | The punch file will be the output medium. |
| PLOT | Generates strain for the requested set but no printer output. |
| REAL or IMAG | Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. |
| PHASE | Requests polar format (magnitude and phase) of complex output. Phase output is in degrees. |
| PSDF | Requests the power spectral density function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 10. |
| ATOC | Requests the autocorrelation function be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 10. |
| CRMS | Requests the cumulative root mean square function be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 10. |
| RMS | Requests the root mean square and zero crossing functions be calculated for random analysis post-processing. Request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 10. |
| RALL | Requests all of PSDF, ATOC, RMS, and CRMS be calculated for random analysis post-processing. The request must be made above the subcase level and RANDOM must be selected in the case control. See Remark 10. |
| RPRINT | Writes random analysis results to the print file. (Default) See Remark 10. |

| Describer | Meaning |
|---|---|
| NORPRINT | Disables the writing of random analysis results to the print file. See Remark 10. |
| RPUNCH | Writes random analysis results to the punch file. See Remark 10. |
| VONMISES | von Mises strain is output. |
| MAXS or SHEAR | Maximum shear strains are output. |
| STRCUR | Strain at the reference plane and curvatures is output for plate elements. |
| FIBER | Strain at locations Z1, Z2 is computed for plate elements. |
| CENTER | Outputs strains at the center of the element only. See Remark 16. |
| CORNER or BILIN | Outputs strains at the center and corner grid points using extrapolation. See Remark 17. |
| SGAGE | Outputs strains at the center and corner grid points using strain gage approach. See Remark 18. |
| CUBIC | Outputs strains at the center and corner grid points using cubic bending correction. See Remark 19. |
| CPLYMID | Requests element strains at the middle of each ply for elements referencing PCOMPS property entries. See Remark 11, Remark 12, Remark 13, and Remark 15. |
| CPLYBT | Requests element strains at the bottom and top of the ply for elements referencing PCOMPS property entries. See Remark 11, Remark 12, Remark 13, and Remark 15. |
| CPLYBMT | Requests element strains at the bottom, middle, and top of each ply for elements referencing PCOMPS property entries. See Remark 11, Remark 12, Remark 13, and Remark 15 |
| ALL | Output strain for all elements. |

| Describer | Meaning |
| --- | --- |
| n | Set identification of a previously appearing SET command. Only strain for elements with identification numbers that appear on this SET command will be output. (Integer>0) |
| NONE | No element strain will be output. |

**REMARKS:**

1. In SOLs 106 and 129, nonlinear strains for nonlinear elements are requested by the STRESS/NLSTRESS commands and appear in the nonlinear stress output. The STRAIN command will generate additional output for total strain except for hyperelastic elements. The additional STRAIN output request will also be ignored for nonlinear material elements when the parameter LGDISP is -1, which is the default (strains will appear in the nonlinear stress output).

2. Both PRINT and PUNCH may be requested.

3. STRAIN=NONE overrides an overall output request.

4. The PLOT option is used when strains are requested for postprocessing but no printed output is desired.

5. Definitions of stress, strain, curvature, and output locations are given in the *NX Nastran Element Library Reference Manual*.

6. If the STRCUR option is selected, the values of Z1 will be set to 0.0 and Z2 will be set to -1.0 on the output.

7. The VONMISES, MAXS, and SHEAR describers are ignored in the complex eigenvalue and frequency response solution sequences. Although, von Mises stress and strain are computed by default in the frequency response solutions 108 and 111 when stress and strain results are requested. The system cell setting NASTRAN SYSTEM(579)=1 can be defined to disable the von Mises stress and strain request in SOLs 108 and 111.

8. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. (In superelement analysis, the first subcase refers to the first subcase of each superelement. Therefore, it is recommended that these options be specified above all subcases.) Consequently, options specified in subcases other than the first subcase will be ignored. See also the FORCE command for further discussion. These options

are discussed in “Understanding Plate and Shell Element Output” in the *NX Nastran Element Library Reference*.

9. The defaults for SORT1 and SORT2 depend on the type of analysis:

    - SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis. If SORT2 is selected in a frequency response solution for one or more of the commands ACCE, DISP, FORC, GPFO, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT2 format.

    - SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, GPFO, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO then the remaining commands will also be output in SORT1 format.

    - XY plot requests will force SORT2 format thus overriding SORT1 format requests.

10. The following applies to random solutions:

    - By default, frequency response results are not output. If in addition to random output, frequency response output is desired, specify SYSTEM(524)=1 or RANFRF=1 in the input file. The PRINT, PUNCH, PLOT describers control the frequency response output. The RPRINT, NORPRINT, RPUNCH describers control the random output.

    - The SORT1 and SORT2 describers only control the output format for the frequency response output. The output format for random results is controlled using the RPOSTS1 describer on the RANDOM case control command or the parameter RPOSTS1, except for RMS results, which are only available in SORT1 format.

    - Any combination of the PSDF, ATOC, RMS, and CRMS describers can be selected. The RALL describer selects all four.

11. If some combination of the CPLYMID, CPLYBT, or CPLYBMT describers are specified, the describer producing the most output data is used.

12. If different CPLYMID, CPLYBT, or CPLYBMT describers are specified on STRESS and STRAIN case control commands in the same input file, the describer specified on the STRESS case control command takes precedence.

13. Failure indices and strength ratios for in-plane ply failure are output for the locations corresponding to the CPLYMID/CPLYBT/CPLYBMT specification and the CENTER/CORNER (or BILIN)/SGAGE/CUBIC specification. Failure indices

and strength ratios for inter-laminar failure are always output at the top and bottom of each ply at the locations corresponding to the CENTER/CORNER (or BILIN)/SGAGE/CUBIC specification.

14. A PSHELL entry with a MID1 or MID2 greater than or equal to $10^8$ requires the parameter NOCOMPS to be 0 or -1 for any stress or strain recovery. See the parameter NOCOMPS for more information.

15. For elements referencing PCOMPS property entries, the REAL, IMAG, PHASE, VONMISES, MAXS, SHEAR, STRCUR, FIBER, SGAGE, CUBIC, PSDF, ATOC, CRMS, RALL, RPRINT, NOPRINT, and RPUNCH describers are not supported.

16. For the CENTER option, strains are output at the center of the element for CQUAD4, CQUADR, and CTRIAR elements that reference a PSHELL entry. For CQUAD4, CQUADR, CTRIA3, CTRIAR, CTRIA6, and CQUAD8 elements that reference a PCOMP or PCOMPG entry, ply strains are always reported at the center of the element. For CHEXA and CPENTA elements that reference a PCOMPS entry, the ply strains are output at the center of the element for each ply.

17. For the CORNER (or BILIN) option, strains are output at the center and grid points for CQUAD4, CQUADR, and CTRIAR elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the ply strains are output at the center and corner grid locations for each ply.

18. For the SGAGE option, strains are output at the center and grid points for CQUAD4 elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the output is the same as that obtained by specifying CORNER or BILIN.

19. For the CUBIC option, strains are output at the center and grid points for CQUAD4 and CQUADR elements that reference a PSHELL entry. For CHEXA and CPENTA elements that reference a PCOMPS entry, the output is the same as that obtained by specifying CORNER or BILIN.

20. Shear strain is output as engineering shear strain, which is twice the tensor shear strain.

## Classical Lamination Theory for Shell Composites (PCOMP,PCOMPG)

Classical lamination theory makes the following assumptions regarding the behavior of the laminae:

- The laminate consists of perfectly bonded laminae.
- The bonds are infinitesimally thin and nonshear-deformable; i.e., displacements are continuous across laminae boundaries so that no lamina can slip relative to another.

- There is linear variation of strain through the laminate thickness.

Deformation in the X-Y plan of the plate at any point C at a distance z in the normal direction to plate middle surface is

$$U = U_0 + z\theta_y$$

**Equation 1-1.**

$$V = V_0 + z\theta_x$$

**Equation 1-2.**

where *U*, *V*, and *W* are the displacements along the X, Y, and Z directions in the element coordinate system, and $\theta_x$, $\theta_y$ are the rotations.

The strain-displacement-middle surface strain and curvatures relationship is given by:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \dfrac{\partial U_0}{\partial x} \\ \dfrac{\partial V_0}{\partial y} \\ \dfrac{\partial U_0}{\partial y} + \dfrac{\partial V_0}{\partial x} \end{Bmatrix} + z \begin{Bmatrix} \dfrac{\partial \theta_y}{\partial x} \\ -\dfrac{\partial \theta_x}{\partial y} \\ \dfrac{\partial \theta_y}{\partial y} - \dfrac{\partial \theta_x}{\partial x} \end{Bmatrix} = \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{Bmatrix} - z \begin{Bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{Bmatrix}$$

**Equation 1-3.**

where the $^0$'s and $\chi$'s are the middle surface strains and curvatures, respectively.

The stress resultants for an N-layer laminate are obtained by integration of the stresses in each lamina through the laminate thickness as:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \int_{-\frac{t}{2}}^{\frac{t}{2}} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} dz = \sum_{k=1}^{N} \int_{z_{k-1}}^{z_k} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix}_k dz$$

**Equation 1-4.**

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = -\int_{-\frac{t}{2}}^{\frac{t}{2}} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} z\,dz = -\sum_{k=1}^{N} \int_{z_{k-1}}^{z_k} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix}_k z\,dz$$

**Equation 1-5.**

The stress resultant to strain relationship is:

$$\begin{Bmatrix} N_x \\ N_y \\ N_{xy} \end{Bmatrix} = \sum_{k=1}^{N} [G]_k \left\{ \int_{z_{k-1}}^{z_k} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{Bmatrix}_k dz - \int_{z_{k-1}}^{z_k} \begin{Bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{Bmatrix}_k z\,dz \right\}$$

**Equation 1-6.**

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \sum_{k=1}^{N} [G]_k \left\{ -\int_{z_{k-1}}^{z_k} \begin{Bmatrix} \varepsilon_x^0 \\ \varepsilon_y^0 \\ \gamma_{xy}^0 \end{Bmatrix}_k z\,dz + \int_{z_{k-1}}^{z_k} \begin{Bmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{Bmatrix}_k z^2\,dz \right\}$$

**Equation 1-7.**

where $[G]_k$ is the material matrix transformed from the laminate coordinate system into the lamina coordinate system.

These relations can be written in the following form used to describe composite elements:

$$\begin{Bmatrix} F \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B & D \end{bmatrix} \begin{Bmatrix} \varepsilon^0 \\ \chi \end{Bmatrix}$$

**Equation 1-8.**

where:

| | | |
|---|---|---|
| [*A*] | = | $\sum_{k=1}^{N} [G]_k (z_k - z_{k-1})$ |
| [*B*] | = | $\frac{1}{2} \sum_{k=1}^{N} [G]_k (z_k^2 - z_{k-1}^2)$ |
| [*D*] | = | $\frac{1}{3} \sum_{k=1}^{N} [G]_k (z_k^3 - z_{k-1}^3)$ |

are named in composite element literature as the membrane, membrane-coupling, and bending matrices, respectively.

In the shell element formulation in NX Nastran (CQUAD4, CQUAD8, CTRIA3, and CTRIA6), these relationships take the following form:

$$\begin{Bmatrix} N \\ M \\ Q \end{Bmatrix} = \begin{bmatrix} TG_1 & T^2 G_4 & 0 \\ T^2 G_4 & \frac{T^3}{12} G_2 & 0 \\ 0 & 0 & T_s G_3 \end{bmatrix} \begin{Bmatrix} \varepsilon^0 \\ \chi \\ \gamma \end{Bmatrix}$$

where:

**Equation 1-9.**

| | | |
|---|---|---|
| $[A]$ | = | $TG_1$ (3x3 matrix) |
| $[B]$ | = | $-T^2G_4$ (3x3 matrix) |
| $[D]$ | = | $\frac{T^3}{12}G_2$ (2x2 matrix) |
| $\{Q\}$ | = | $\begin{Bmatrix} Q_x \\ Q_y \end{Bmatrix}$ = transverse shear resultants |
| $\{\gamma\}$ | = | $\begin{Bmatrix} \gamma_x \\ \gamma_y \end{Bmatrix}$ = transverse shear strains |
| $T$ | | nominal plate thickness |
| $T_s$ | = | effective transverse shear material thickness |
| $[G_3]$ | = | effective transverse shear material matrix |

If you use the PSHELL entry, you can directly input $G_1$, $G_2$, $G_4$, $T$, $G_3$, and $T_s$ ($G_1$=MID1, $G_2$=MID2, $G_3$=MID3, $G_4$=MID4 on the PSHELL entry) If you use the PCOMP entry, you can have NX Nastran calculate the composite equivalent material matrices from the data you supply.

The thermal loads and moments are defined as:

$$\begin{Bmatrix} N^{th} \\ M^{th} \end{Bmatrix} = \begin{bmatrix} G\alpha_{\varepsilon_0} & G\alpha_{\chi\varepsilon_0} \\ G\alpha_{\chi\varepsilon_0} & G\alpha_{\chi} \end{bmatrix} \begin{Bmatrix} \Delta T_0 \\ \Delta T_{grad} \end{Bmatrix}$$

$$= \begin{bmatrix} t \cdot G_1 \cdot \alpha_{\varepsilon_0} & t^2 \cdot G_4 \cdot \alpha_{\chi\varepsilon_0} \\ t^2 \cdot G_4 \cdot \alpha_{\chi\varepsilon_0} & \frac{t^3}{12} \cdot G_2 \cdot \alpha_{\chi} \end{bmatrix} \begin{Bmatrix} \Delta T_0 \\ \Delta T_{grad} \end{Bmatrix}$$

$$= \begin{bmatrix} A \cdot \alpha_{\varepsilon_0} & B \cdot \alpha_{\chi\varepsilon_0} \\ B \cdot \alpha_{\chi\varepsilon_0} & D \cdot \alpha_{\chi} \end{bmatrix} \begin{Bmatrix} \Delta T_0 \\ \Delta T_{grad} \end{Bmatrix}$$

The equivalent thermal properties are computed as follows:

MID1 (membrane) is computed as:

$$\{\alpha_{\varepsilon_0}\} = [A]^{-1}\{G\alpha_{\varepsilon_0}\}$$

MID2 (bending) is computed as:

$$\left\{\alpha_{\chi}\right\} = \left[D\right]^{-1}\left\{G\alpha_{\chi}\right\}$$

MID4 (membrane bending coupling) is computed as:

$$\left\{\alpha_{\chi\varepsilon_0}\right\}\Big|_{UseInCodes} = \left[B\right]\left\{\alpha_{\chi\varepsilon_0}\right\} = \left\{G\alpha_{\chi\varepsilon_0}\right\}$$

The thermal loads and moments from the homogeneous thermal membrane strain and curvature is computed as:

$$\begin{Bmatrix} N^{th} \\ M^{th} \end{Bmatrix} = \begin{bmatrix} A & B \\ B & D \end{bmatrix} \begin{Bmatrix} \varepsilon_0^{th} \\ \chi^{th} \end{Bmatrix}$$

$$\begin{Bmatrix} \varepsilon_0^{th} \\ \chi^{th} \end{Bmatrix} = \begin{bmatrix} A & B \\ B & D \end{bmatrix}^{-1} \begin{Bmatrix} N^{th} \\ M^{th} \end{Bmatrix}$$

$$\begin{bmatrix} G\alpha_{\varepsilon_0} & G\alpha_{\chi\varepsilon_0} \\ G\alpha_{\chi\varepsilon_0} & G\alpha_{\chi} \end{bmatrix} =$$

$$= \begin{bmatrix} \sum\limits_{k=1}^{N} \begin{bmatrix} \bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{16} \\ \bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{26} \\ \bar{Q}_{16} & \bar{Q}_{26} & \bar{Q}_{66} \end{bmatrix}_k \begin{Bmatrix} \alpha_x \\ \alpha_y \\ \alpha_{xy} \end{Bmatrix}_k \left(z_k - z_{k-1}\right) & \frac{1}{2}\sum\limits_{k=1}^{N} \begin{bmatrix} \bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{16} \\ \bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{26} \\ \bar{Q}_{16} & \bar{Q}_{26} & \bar{Q}_{66} \end{bmatrix}_k \begin{Bmatrix} \alpha_x \\ \alpha_y \\ \alpha_{xy} \end{Bmatrix}_k \left(z_k^2 - z_{k-1}^2\right) \\ \frac{1}{2}\sum\limits_{k=1}^{N} \begin{bmatrix} \bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{16} \\ \bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{26} \\ \bar{Q}_{16} & \bar{Q}_{26} & \bar{Q}_{66} \end{bmatrix}_k \begin{Bmatrix} \alpha_x \\ \alpha_y \\ \alpha_{xy} \end{Bmatrix}_k \left(z_k^2 - z_{k-1}^2\right) & \frac{1}{3}\sum\limits_{k=1}^{N} \begin{bmatrix} \bar{Q}_{11} & \bar{Q}_{12} & \bar{Q}_{16} \\ \bar{Q}_{12} & \bar{Q}_{22} & \bar{Q}_{26} \\ \bar{Q}_{16} & \bar{Q}_{26} & \bar{Q}_{66} \end{bmatrix}_k \begin{Bmatrix} \alpha_x \\ \alpha_y \\ \alpha_{xy} \end{Bmatrix}_k \left(z_k^3 - z_{k-1}^3\right) \end{bmatrix}$$

The following coefficients are used to determine equivalent thermal properties:

$$\left\{G\alpha_{\varepsilon_0}\right\} = \sum_{k=1}^{N} [G]_k \{\alpha\}_k (z_k - z_{k-1})$$

$$\{G\alpha_{\chi}\} = \frac{1}{3}\sum_{k=1}^{N}[G]_k\{\alpha\}_k(z_k^3 - z_{k-1}^3)$$

$$\left\{G\alpha_{\chi\varepsilon_0}\right\} = \frac{1}{2}\sum_{k=1}^{N}[G]_k\{\alpha\}_k(z_k^2 - z_{k-1}^2)$$

These coefficients are used to calculate the equivalent thermal properties as follows:

$$\left\{\alpha_{\varepsilon_0}\right\} = [A]^{-1}\left\{G\alpha_{\varepsilon_0}\right\}$$

and

$$\{\alpha_x\} = [D]^{-1}\{G\alpha_x\}$$

where $\{\alpha_{\varepsilon 0}\}$, $\{\alpha_{\chi}\}$, and $\{\alpha_{\chi\varepsilon 0}\}$ are the equivalent thermal properties for membrane, bending, and membrane/bending coupling, respectively. Note that $\{\alpha_0\}$ is not directly calculated, but is determined from $\{G\alpha_0\}$ when the PCOMP input is used when the MID4 field on the PSHELL is > 400,000,000. Note that $\{G_{\chi\xi 0}\}$ cannot be input directly using PSHELL and $\{\alpha_0\}$ can be input only if [*B*] is invertible (which is generally not true).

The thermal expansion relationships in the shell element formulation take the following form:

$$\alpha_{\varepsilon_0} = T\alpha_1$$

$$\alpha_{\chi} = \frac{T^3}{12}\alpha_2$$

$$\alpha_{\chi\varepsilon_0} = T^2\alpha_3$$

where $\alpha_1$, $\alpha_2$, and $\alpha_3$ are the thermal expansion inputs on the materials referenced by the MID1, MID2, and MID4 fields on the PSHELL entry. If you use PCOMP, NX Nastran automatically calculates these relationships.

The terms $G_1$, $G_2$, and $G_4$ are defined by the following integrals:

$$G_1 = \frac{1}{T}\int [G_e]dz$$

$$G_2 = \frac{1}{I}\int z^2 [G_e]dz$$

$$G_4 = \frac{1}{T^2}\int (-z)[G_e]dz$$

**Equation 1-10.**

The limits on the integration are from the bottom surface to the top surface of the laminated composite. The matrix of material moduli, $[G_e]$, has the following form for isotropic materials:

$$[G_e]_l = \begin{bmatrix} \frac{E}{1-\nu^2} & \frac{\nu E}{1-\nu^2} & 0 \\ \frac{\nu E}{1-\nu^2} & \frac{E}{1-\nu^2} & 0 \\ 0 & 0 & \frac{E}{2(1+\nu)} \end{bmatrix}$$

**Equation 1-11.**

For orthotropic materials, the matrix, $[G_e]$, is written as follows:

$$[G_e]_0 = \begin{bmatrix} \frac{E_1}{1 - \nu_1\nu_2} & \frac{\nu_1 E_2}{1 - \nu_1\nu_2} & 0 \\ \frac{\nu_2 E_1}{1 - \nu_1\nu_2} & \frac{E_2}{1 - \nu_1\nu_2} & 0 \\ 0 & 0 & G_{12} \end{bmatrix}$$

**Equation 1-12.**

Here, $\nu_1 E_2 = \nu_2 E_1$ to satisfy the requirement that the matrix of elastic moduli be symmetric. In general, the analyst may supply element properties with respect to a particular orientation which does not necessarily correspond to the principal material axes. In this case, the analyst must also supply the value of the angle, θ or material coordinate system that orients the element material axis relative to the side G1-G2 of the element. The material elastic modulus matrix is then transformed by the program into the element modulus matrix through the relation

$$[G_e] = [U]^t [G_m][U]$$

**Equation 1-13.**

where:

$$[U] = \begin{bmatrix} \cos^2\theta & \sin^2\theta & \cos\theta\sin\theta \\ \sin^2\theta & \cos^2\theta & -\cos\theta\sin\theta \\ -2\cos\theta\sin\theta & 2\cos\theta\sin\theta & \cos^2\theta - \sin^2\theta \end{bmatrix}$$

**Equation 1-14.**

The finite element model for a structure composed of composite materials requires the evaluation of the matrix of elastic moduli for each plate element of the model. The characteristics of the composite media are totally contained in these matrices.

To illustrate evaluation of these matrices, consider the cross-ply laminates shown in Figure 24-2. Here, all three configurations are represented by a single quadrilateral plate element. The coordinate axes are coincident with the element coordinate axes. Then, if we assume that each lamina of the n-ply laminates is of thickness *T*/*n*, where *T* is the total thickness of each of three configurations, the matrices of elastic moduli may be evaluated from the following relations:

$$[G_1] = \frac{1}{T}\left\{ \int_{-\frac{T}{2}}^{-\frac{T}{2}+\frac{T}{n}} [G_e]_1 z^2 dz + \int_{-\frac{T}{2}+\frac{T}{n}}^{-\frac{T}{2}+\frac{2T}{n}} [G_e]_2 dz + \ldots + \int_{-\frac{T}{2}+\frac{(n-1)T}{n}}^{\frac{T}{2}} [G_e]_n dz \right\}$$

**Equation 1-15.**

$$[G_2] = \frac{1}{T}\left\{ \int_{-\frac{T}{2}}^{-\frac{T}{2}+\frac{T}{n}} [G_e]_1 dz + \int_{-\frac{T}{2}+\frac{T}{n}}^{-\frac{T}{2}+\frac{2T}{n}} [G_e]_2 z^2 dz + \ldots + \int_{-\frac{T}{2}+\frac{(n-1)T}{n}}^{\frac{T}{2}} [G_e]_n z^2 dz \right\}$$

**Equation 1-16.**

$$[G_4] = \frac{1}{T^2}\left\{\int_{-\frac{T}{2}}^{-\frac{T}{2}+\frac{T}{n}} [G_e]_1(-z)dz + \int_{-\frac{T}{2}+\frac{T}{n}}^{-\frac{T}{2}+\frac{2T}{n}} [G_e]_2(-z)dz + \ldots + \int_{\frac{T}{2}+\frac{(n-1)T}{n}}^{\frac{T}{2}} [G_e]_n(-z)dz\right\}$$

**Equation 1-17.**

These relations reflect the assumption that the xy-plane of the element coordinate system is coincident with the geometric middle plane of the laminate. The xy-plane of the element coordinate system is defined in the mean plane of the element so that any offset between the mean plane of the connected grid points and the geometric middle plane of the laminate would be reflected in the integration limits of the preceding relations.

Note that $I = T^3/12$ in the evaluation of $[G_2]$, i.e., the value $(12I)/\Gamma^2$, will be assigned the default value of 1.0 on the plate element property entry.

The matrix of elastic moduli for transverse shear, $[G_3]_m$ is defined as a two-by-two matrix of the form

$$[G_3]_m = \begin{bmatrix} G_{11} & G_{12} \\ G_{21} & G_{22} \end{bmatrix}$$

**Equation 1-18.**

and the corresponding matrix transformed into an element coordinate system is given by

$$[G_3]_e = [W]^T[G_3]_m[W]$$

**Equation 1-19.**

where $[W] = \begin{bmatrix} \cos\theta & \sin\theta \\ -\sin\theta & \cos\theta \end{bmatrix}$

The mean value of the transverse shear modulus $\bar{G}$ for the laminated composite is defined in terms of the transverse shear strain energy, *U*, through the depth

$$U = \frac{1}{2}\frac{V^2}{\bar{G}T} = \frac{1}{2}\int\frac{(\tau(z))^2}{G(z)}dz$$

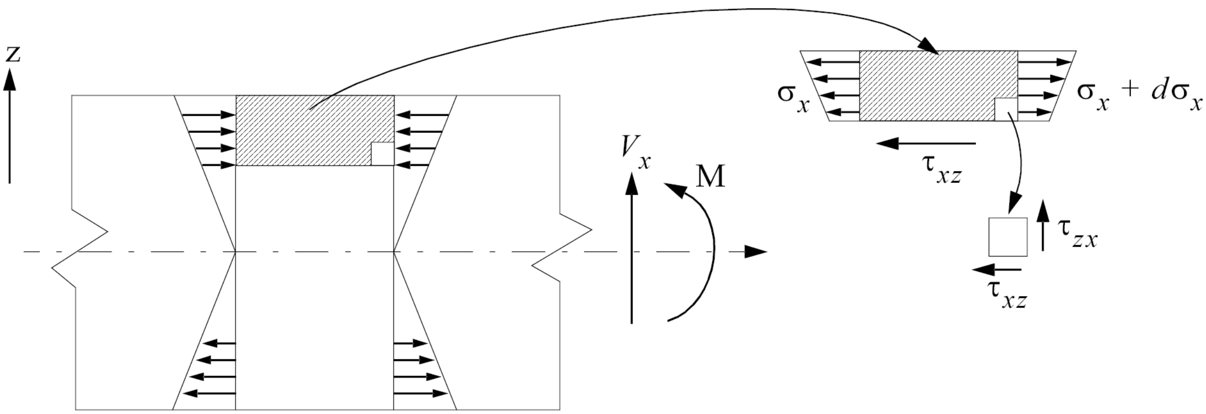
**Equation 1-20.**

A unique mean value of the transverse shear strain is assumed to exist for both the x and y components of the element coordinate system, but for ease of discussion, only the evaluation of an uncoupled x component of the shear moduli will be illustrated here. From Eq. 1-20 the mean value of transverse shear modulus may be written in the following form

$$\frac{1}{G_x} = \frac{T}{V_x^2}\sum_{i=1}^{N}\int_{z_{i-1}}^{z_i}\frac{(\tau_{zx}(z))^2}{(G_x)_i}dz$$

**Equation 1-21.**

where *G* is an “average” transverse shear coefficient used by the element code and $(G_x)_i$ is the local shear coefficient for layer . To evaluate Eq. 1-12, it is necessary to obtain an expression for $(\tau_{zx}(z))$. This can be accomplished by assuming that the x- and y-components of stress are decoupled from one another. This assumption allows the desired equation to be deduced through an examination of a beam unit cross-sectional width.




The equilibrium conditions in the horizontal direction and for total moment are

$$\frac{\partial \tau_{xz}}{\partial z} + \frac{\partial \sigma_x}{\partial x} = 0$$

**Equation 1-22.**

$$V_x + \frac{\partial M_x}{\partial_x} = 0$$

**Equation 1-23.**

Now, if the location of the neutral surface is denoted by $\bar{z}_x$ and ρ is the radius of curvature of the beam, the axial stress $E_x$ may be expressed in the form

$$\sigma_x + \frac{E_x(\dot{z}_x - z)}{(\overline{EI})_x} = 0$$

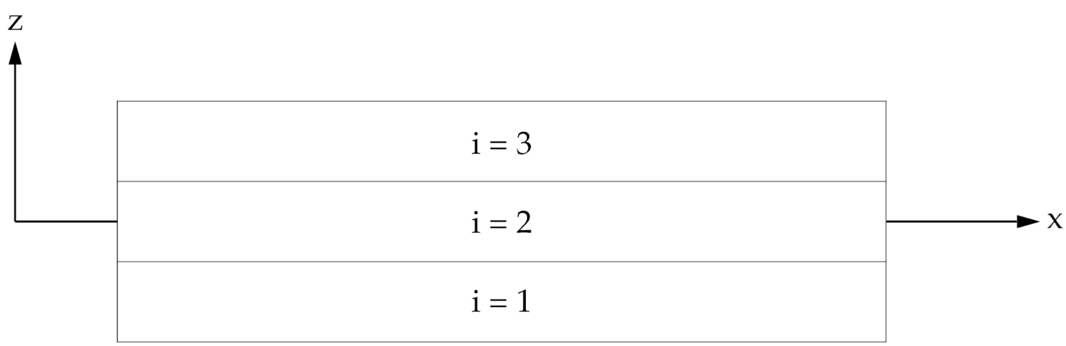
**Equation 1-24.**

Relation 20 may be differentiated with respect to x combined with Eq. 1-22 and Eq. 1-23. In a region of constant $E_x$ the result may be integrated to yield the following expression

$$\tau_{xz} = C_i + \frac{V_x}{(\overline{EI})_x}\left(\bar{z}_x z - \frac{z^2}{2}\right) E_{xi} \qquad z_{i-1} < z < z_i$$

**Equation 1-25.**

Eq. 1-29 is particularly convenient to use in the analysis of n-ply laminates because sufficient conditions exist to determine the constants Ci (i = 1,2,...,n) and the “directional bending center” $\bar{z}_x$. For example, consider the following laminated configuration




At the bottom surface ($i = 1$, $z = z_0$, and $\tau_{xz} = 0$)

$$C_1 = \frac{-V_x}{(\overline{EI})_x}\left(\bar{z}_x z_0 - \frac{z_0^2}{2}\right) E_{x1}$$

**Equation 1-26.**

and for the first ply at the interface between plies $i = 1$ and $i = 2$ ($z = z_1$)

$$(\tau_{xz})_1 = +\frac{V_x}{(\overline{EI})_x}\left[\bar{z}_x(z_1 - z_0)\frac{1}{2}[z_1^2 - z_0^2]\right] E_{x1}$$

**Equation 1-27.**

At this interface between plies $i = 1$ and $i = 2$,

$$(\tau_{xz})_2 = C_2 + \frac{V_x}{(\overline{EI})_x}\left(\dot{z}_x z_1 - \frac{z_1^{\;2}}{2}\right)E_{x2}$$

**Equation 1-28.**

and as $(\tau_{xz})_2 = (\tau_{xz})_1$ at $z = z_1$,

$$C_2 = (\tau_{xz})_1 - \frac{V_x E_{x2}}{(\overline{EI})_x}\left[\dot{z}_x z_1 - \frac{1}{2} z_1^{\;2}\right]$$

**Equation 1-29.**

Then, in the ply $z_1 < z < z_2$ the shear is

$$\tau_{xz}(z) = (\tau_{xz})_1 \frac{V_x E_{x2}}{(\overline{EI})_x}\left[\dot{z}_x (z - z_1) - \frac{1}{2}(z^2 - z_1^{\;2})\right]$$

**Equation 1-30.**

In general, for any ply, $Z_{i-1} < z < z_i$, the shear is

$$\tau_{xz}(z) = (\tau_{xz})_{i-1} \frac{V_x E_{xi}}{(\overline{EI})_x}\left[\dot{z}_x (z - z_{i-1}) - \frac{1}{2}(z^2 - (z_{i-1})^2)\right]$$

**Equation 1-31.**

At any ply interface, $z_i$, the shear is therefore

$$(\tau_{xz})_i = \frac{V_x}{(\overline{EI})_x} \sum_{j=1}^{i} E_{xj} T_j \left[\dot{z} - \frac{1}{2}(z_j + z_{j-1})\right]$$

**Equation 1-32.**

where $T_j = z_j - (z_j - 1)$.

Note that the shear at the top face, $(\tau_{xz})_n$, is zero and therefore

$$(\tau_{xz})_n = \frac{V_x}{(\overline{EI})}\left[\dot{z}_x \sum_{i=1}^{n} E_{xj}T_j - \sum_{i=1}^{n} E_{xj}T_j(z_j + z_{j-1})/2\right] = 0$$

**Equation 1-33.**

Eq. 1-33 proves that if $\bar{z}_x$ is the bending center, the shear at the top surface must be zero.

Eq. 1-31 could be substituted into Eq. 1-22 and integrated. A better form of Eq. 1-33, for this purpose is

$$(\tau_{xz}(z))_i = \frac{V_x E_{xi}}{(\overline{EI})_x}\left[f_{xi} + \dot{z}_x(z - z_{i-1})\frac{1}{2}(z^2 - (z_{i-1})^2)\right]$$

**Equation 1-34.**

where

$$f_{xi} = \frac{1}{E_{xi}}\sum_{j=1}^{i-1} E_{xj}T_j\left[\dot{z}_x - \frac{1}{2}(z_j + z_{j-1})\right]$$

**Equation 1-35.**

Substituting Eq. 1-34 into Eq. 1-25 and integrating the results, we obtain

$$\frac{1}{G_x} = \frac{T}{(\overline{EI})_x^2}\sum_{i=1}^{N}\frac{1}{G_{xi}}R_{xi}$$

**Equation 1-36.**

where

$$R_{xi} = (E_{xi})^2 T_i \left[ \left\{ f_{xi} + (\dot{z}_x - z_{i-1}) T_i - \frac{1}{3} T_i^2 \right\} f_{xi} + \left\{ \frac{1}{3}(\dot{z}_x - 2\, z_{i-1}) - \frac{1}{4} T_i \right\} \dot{z}_x T_i^2 + \left\{ \frac{1}{3} z_{i-1}^2 + \frac{1}{4} z_{i-1} T_i + \frac{1}{20} T_i^2 \right\} T_i^2 \right]$$

**Equation 1-37.**

This expression for the inverse shear modulus for the x-direction may be generalized to provide for the calculation of each term in the two-by-two matrix of shear moduli.

$$[\bar{G}_{kl}] = \left[ \frac{T}{(\overline{EI})_{kk}^2} \sum_{i=1}^{n} [G_{kl}^i]^{-1} R_{ki} \right]^{-1}$$

**Equation 1-38.**

where:

$k$ = 1,2

$l$ = 1,2

Note that if no shear is given, $[G^i]^{-1}$, and also that in Eq. 1-31

where $[G_2]^*$ is calculated in the same manner as $[G_2]$ except that Poisson's Ratio is set to zero.

The moduli for individual plies are provided through user input because, in general, $G_{12} \neq G_{21}$ will be used for the coupling terms. Finally,

$$[G_3] = \begin{bmatrix} \bar{G}_{11} & (\bar{G}_{12})_{avg} \\ (\bar{G}_{12})_{avg} & \bar{G}_{22} \end{bmatrix}$$

**Equation 1-39.**

As an example, consider a single layer element. For this case let $z_{i-1} = -T/2$, $\bar{z} = 0$, $f_o = 0$, and $EI = ET^3/12$. Evaluating Eq. 1-38 we obtain

$$R_i = E^2T^5\left[\frac{1}{12} - \frac{1}{8} + \frac{1}{20}\right] = \frac{E^2T^5}{120}$$

**Equation 1-40.**

and

$$\frac{1}{G} = \left(\frac{(12)^2T}{E^2T^6}\right) \cdot \left(\frac{E^2T^5}{120\ G_1}\right) = \frac{6}{5G_1}$$

**Equation 1-41.**

which provides the correct factor for a nonuniform shear distribution in a plate.

The coefficients of thermal expansion derived for membrane-bending coupling, which appear in the A1, A2, and A12 fields of MAT2 entry and correspond to the MID4 Field on the PSHELL, require special interpretation. They are given by:

$$\{\alpha_{MAT2}\} = [G_{ij_{MAT2}}]\{\alpha_{ACTUAL}\}$$

**Equation 1-42.**

To obtain the actual values of A1, A2, and A12 $\{\alpha_{MAT2}\}$, you must solve Eq. 1-42.

# Chapter 2:   Problem Report (PR) fixes

## Problem Report (PR) fixes

The NX Nastran 10.2 maintenance release includes the following fixes.

| PR# | Version Reported | Problem Description |
|---|---|---|
| 1987384 | V9.0 | Spelling mistake in message 4306. 'User Iformation' has been corrected to 'User Information'. |
| 7263723 | V9.1 | When creating an external superelement with the non-sparse MATOP4 option (negative unit number), the SEBULK card created by the software was incorrect. The superelement was incorrectly designated as "EXTERNAL" rather than "EXTOP4." The problem has been fixed. |
| 7270742 | V9.0 | Glue results requested with the BGRESULTS case control command had a sign convention which was opposite from other types of force output such as BCRESULTS and MPCFORCE. The sign convection of BGRESULTS has been reversed in 10.2. Setting system(624)=1 will revert the output back to the previous glue force convention. |
| 7270786 | V9.1 | A SOL 111 run which included external superelements and no output requested would run without progress repeating the same lines in the .f04 file. The problem has been fixed. |
| 7286371 | V10.0 | NX Nastran was not reading the RCF file in the installation location. The problem has been fixed. |
| 7291369 | V8.5 | Receiving the FATAL MESSAGE 3046 (SQFREQ) in a SOL 111 restart solution. Message stated that no excitations or loads existed. The problem has been fixed. |
| 7304082 | V10.0 | The DPD module was going into an infinite loop and causing disk issues. The problem has been fixed. |

| | | |
|---|---|---|
| 7323837 | V10.0 | Slow performance with ADD module. The problem has been fixed. |
| N/A | V3.0 | When multiple CTRIA6/CQUAD8 elements were defined for a SOL 200 analysis, there was a problem moving on to the next element causing a data corruption. The problem has been fixed. |
| 7266560 | V10.0 | A SOL 111 restart analysis which includes an external superelement failed to generate the expected results if the analysis included more than 1 subcase. The problem has been fixed. |
| 2248669 | V9.0 | Unable to load an external superelement .op2 file into an NX assembly. Invalid geometry data blocks in the OP2 file confused the NX post reader. The problem has been fixed. |
| N/A | N/A | In some cases, the header for total panel contributions was incorrect. The problem has been fixed. |
| N/A | N/A | For superelements, incorrect case control card checks for subcases were being applied resulting in no solution for the subcase. The problem has been fixed. |
| 7319762 | V10.0 | A model with a very large number of RBE2 elements is receiving FATAL MESSAGE 7572 (DFMSYN) when using the direct solver and BEND ordering (default). The problem has been fixed. |
| 7348203 | V10.1 | Bolt preload in SOL 401 was resulting in unexpected glue and contact forces. The problem was that forces on some CTETRA midside grids were not properly accounted. The problem has been fixed. |
| 7184760 | V9.0 | Using the same license server for FEMAP and NX Nastran, after completing a large NX Nastran solution, the NX Nastran license would sometimes remain in use.The workaround was to restart the license server. The problem has been fixed. |
| 7334910 | V10.0 | Incorrect results when bolt preload was included in a model running SMP. The workaround in versions prior to 10.2 is to include sys599=1 to turn off SMP processing in the internal force routines. The problem has been fixed. |
| 8252204 | V9.0 | A SOL 601,106 solution successfully converges at the final time step but fails during result recovery. The problem has been fixed. |

| | | |
|---|---|---|
| 2236324 | V8.5 | A SOL 601,106 solution failed to recover stress results for elements listed in a referenced SET command. The problem has been fixed. |
| 7235008 | V9.1 | A SOL 601,106 solution wrote the incorrect active nodal point for the stress results printed in the f06 output. The problem has been fixed. |
| 7235005 | V9.1 | The order in which SOL 601,106 writes the principal stresses to the f06 file is inconsistent with SOL 101. They are now written in the same order as SOL 101. |
| N/A | N/A | NX Nastran writes an approach code into the output datablocks to designate the analysis type. The approach code written to the nonlinear stress datablock from SOL 601,106 was modified in NX Nastran 10 from type=6 (transient) to type=10 (nonlinear statics). It has been requested to revert to the original type=6 approach code in order to continue post processing these results as time based results. Beginning in this release, SOL 601,106 is again outputting the nonlinear stress datablock with the type=6 approach code. |

**Siemens Industry Software**

**Headquarters**
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

**Americas**
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

**Europe**
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

**Asia-Pacific**
Suites 4301-4302, 43/F
AIA Kowloon Tower, Landmark East
100 How Ming Street
Kwun Tong, Kowloon
Hong Kong
+852 2230 3308

**About Siemens PLM Software**

Siemens PLM Software, a business unit of the Siemens Industry Automation Division, is a leading global provider of product lifecycle management (PLM) software and services with 7 million licensed seats and 71,000 customers worldwide. Headquartered in Plano, Texas, Siemens PLM Software works collaboratively with companies to deliver open solutions that help them turn more ideas into successful products. For more information on Siemens PLM Software products and services, visit www.siemens.com/plm.