Advanced Linear Analysis using MSC Nastran

NAS101B Course Notes

March 2012

MSC Software®
Simulating Reality, Delivering Certainty ™

Legal Information

MSC.Software Corporation reserves the right to make changes in specifications and other information contained in this document without prior notice. The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design. MSC.Software Corporation assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

CONTENTS

CONTENTS

Appendix

A **Documentation** ·

MSC Software®

SECTION 1

INTRODUCTION

INTRODUCTION

COURSE OBJECTIVES

- **In this seminar, you will learn about**
  - Theory of buckling analysis and how to perform a buckling analysis
  - Rigid elements – MPC, RBAR,RBE2, and RBE3
  - Modeling with interface element CINTC and connectors
  - Composites
  - Linear contact and permanent glued contact
  - Different model checkout tools
  - Modeling tips and tricks

MSC Software

BUILDING MSC NASTRAN MODELS IN THIS COURSE

- **In this course, all finite element models will be created and edited using a text editor, not a graphical pre-processor.**

- **Being able to understand and edit MSC Nastran input files using a text editor is a critical skill for the following reasons:**

  - Reviewing the MSC Nastran input file allows the user to check out the model to make sure what looked OK on the graphical screen has been translated correctly into the right types of elements, correctly-specified properties, correctly-defined loads and boundary conditions, etc. In other words, it allows you to verify that you got what you really intended to create.

  - Reviewing the input file is an excellent way to debug a model. By developing the skill to quickly scan the text input file, you can spot problems quickly and discover problems that you may not be able to detect through a graphical interface.

  - Text editing is often the most efficient way to make minor modifications to the model.

  - Not all MSC Nastran capabilities are accessible through the graphical pre-processor.

  - Important information, such as modeling assumptions and model change log, are often documented as comment lines in a MSC Nastran input file.

  - For the reasons mentioned above, many customers choose to use the MSC Nastran input file instead of a graphical database as the primary means of storing MSC Nastran models and exchanging models with other customers.

COURSE WORKSHOPS

- The workshops that accompany this lecture material are presented in both British and metric units.

- In the exercise workbook, the metric units workshops are in the second half of the book, while the British (inch, pound, second) unit versions are in the first half of the book.

- When you perform a workshop, follow the steps for the system of units that you prefer.

COMPANY OVERVIEW

- **MSC.Software (formerly known as MacNeal-Schwendler Corporation) has been supplying sophisticated computer-aided engineering tools since 1963.**

- **MSC.Software is the developer, distributor, and supporter of the most complete and widely-used structural analysis program in the world, MSC Nastran.**

- **NASTRAN* development was initiated in 1966 under the sponsorship of the National Aeronautics and Space Administration, based on the known requirements of the aerospace industry for structural analysis**

    * NASTRAN is a registered trademark of the National Aeronautics and Space Administration.

COMPANY OVERVIEW (Cont.)

- **MSC.Software has been involved in NASTRAN since its inception and has marketed its own enhanced, proprietary version MSC Nastran since 1972.**

WHAT IS MSC NASTRAN?

- **MSC Nastran is a general-purpose, finite element analysis program capable of solving a wide variety of engineering problems including:**
  - Linear static analysis
  - Static analysis with geometric and material nonlinearity
  - Transient analysis with geometric and material nonlinearity
  - Linear and nonlinear contact analysis
  - Normal modes analysis
  - Buckling analysis
  - Direct and modal complex eigenvalue analysis
  - Direct and modal frequency analysis
  - Direct and modal transient analysis

MSC Software

WHAT IS MSC NASTRAN? (Cont.)

- Linear cyclic symmetry (including static, normal modes, buckling, and direct frequency response)
- Linear and nonlinear steady state heat transfer
- Linear and nonlinear transient heat transfer
- Aeroelasticity
- Substructure analysis (superelements)
- Design sensitivity and optimization
- Acoustics
- Composite Material Analysis
- p-elements

WHAT IS MSC NASTRAN? (Cont.)

- **Extensively documented (SimCompanion)**

- **Extensively tested**

- **Continually enhanced with new capabilities**

- **Highly efficient in the use of modern numerical analysis techniques**

- **Mainly written in FORTRAN (with some C)**

- **Used extensively by aerospace, automobile, energy, biomedical, and other industries**

SOURCES OF INFORMATION

- **News file printed at the top of the .f06 file contains information on new capabilities and enhancements, and changes made with respect to previous versions.**

- **Version-dependent release guides provide in-depth discussions of new features.**

- **Current error list delivered with MSC Nastran contains known errors, general limitations, and in most cases, an avoidance for the error. You can obtain updates from your regional support engineer or previously mentioned website.**

- **Users' conferences proceedings, technical articles (both by MSC.Software and outside publications)**

MSC Software

SIMCOMPANION

- **One stop for full online support**

- **Find answers to your questions**

- **Search across ALL content**

- **Subscribe to email notification**

- **Single sign-on to ALL content**

- **Access to other support resources**
    - Case Management Portal
    - Discussion Forums
    - Training Information

http://simcompanion.mscsoftware.com

MSC Software

SIMCOMPANION

- **Personalized Support via the following channels**
  - Web
    - Submit a Case Online
    - Manage My Cases
  - Email
    - List of Addresses in Support Contact Information
  - Phone
    - List of Phone Numbers in Support Contact Information

Web:

http://support.mscsoftware.com/servicerequest

Email:

English

- MD Nastran or MSC Nastran	mscnastran.support@mscsoftware.com
- MD Adams or Adams	mscadams.support@mscsoftware.com
- Patran	mscpatran.support@mscsoftware.com
- Dytran	mscdytran.support@mscsoftware.com
- Easy5	easy5.support@mscsoftware.com
- Fatigue	mscfatigue.support@mscsoftware.com
- Marc	mscmarc.support@mscsoftware.com
- Mvision	mscmvision.support@mscsoftware.com
- Sinda	mscsinda.support@mscsoftware.com
- Sofy	mscsofy.support@mscsoftware.com
- SimDesigner	simdesigner.support@mscsoftware.com
- MSC SimManager	simmanager.support@mscsoftware.com
- MSC SimXpert	simxpert.support@mscsoftware.com
Chinese (Simplified)	support.cn@mscsoftware.com
Chinese (Traditional)	support.tw@mscsoftware.com
Dutch	support.be@mscsoftware.com
French	support.fr@mscsoftware.com
German	support.de@mscsoftware.com
Italian	support.it@mscsoftware.com
Japanese	support.jp@mscsoftware.com
Korean	support.kr@mscsoftware.com
Nordic	support.no@mscsoftware.com
Portuguese	suporte.mscbrasil@mscsoftware.com
Russian	support.ru@mscsoftware.com
Spanish	support.es@mscsoftware.com

Phone:

Belgium and Luxembourg	+31 182 536444
Brazil	0800-891-4346
China - Beijing	+86-10-82607000
China - Shanghai	+86-21-63326655
China - Chengdu	+86-28-86199275/6
China - Shenzhen	+86-755-23811895
Czech Republic	+420 54517 6106
DACH	+49 89 431 987 277
Denmark	+45 61 22 32 00
Finland	0800-9-14709
France	+33 5 34 60 44 80

MSC Software

SIMCOMPANION

- **Product Info and Docs**
  - Access to all Product Documentation

MSC.SOFTWARE TRAINING AND EDUCATION

- **MSC Nastran seminars are held worldwide.**

- **Locations, dates, and descriptions of all scheduled classes can be obtained from SimCompanion**

- **MSC.Software also conducts cost-effective private seminars at clients' facilities upon request**

- **Seminars are available for all the features available in MSC Nastran**
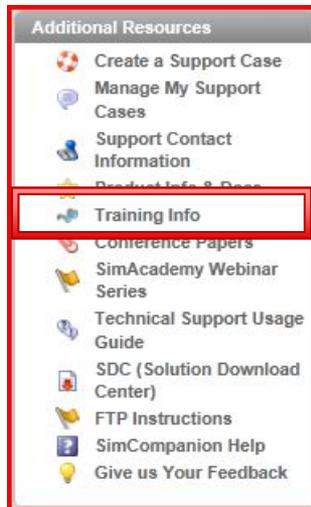
MSC.SOFTWARE TRAINING AND EDUCATION (Cont.)

- **NAS101-A** - *Linear Static & Normal Analysis using MSC Nastran*
- **NAS101-B** - *Advanced Linear Analysis using MSC Nastran*
- **NAS102-A** - *Dynamic Analysis using MSC NastranNAS103 –Nastran Nonlinear Analysis*
- **NAS102-B** - *Advanced Dynamic Analysis using MSC Nastran*
- **NAS103** - *MSC Nastran Nonlinear Analysis*
- **NAS104** - *MSC Nastran Thermal Analysis*
- **NAS106-A** - *Basic Substructure Analysis using MSC Nastran - Primary Superelements*
- **NAS106-B** - *Advanced Substructure Analysis using MSC Nastran - Secondary Superelements*
- **NAS107** - *Design Sensitivity and Optimization in MSC Nastran*
- **NAS110** - *DMAP and Database Application in MSC Nastran*
- **NAS111** - *MSC Nastran Aeroelastic Analysis*
- **NAS113** - *Analysis of Composite Materials with MSC Nastran*
- **NAS115** - *Fluid Structure Analysis in MSC Nastran*
- **NAS120** - *Linear Statics Normal Modes and Buckling Analysis MSC Nastran & Patran*
- **NAS122** – *Dynamic Analysis Using Patran and Nastran*
- **NAS123** –*Nastran Implicit Nonlinear (SOL600) Analysis*
- **NAS127** - *Rotordynamic Analysis using MSC Nastran*
- **NAS133** – *Nastran Advanced Nonlinear (SOL400)*

For more courses and registration, please follow the link.

http://store.mscsoftware.com/training/trainingevents.cfm?PROD=MSC.NASTRAN

MSC Software

SIMCOMPANION

- **Access to Communities**
  - VPD Community Discussion Forums
  - Subscribe to discussion communities of interest

SECTION 2

LINEAR BUCKLING ANALYSIS

CONTENTS

MSC Software®

THEORY OF BUCKLING

- **The equilibrium equations for a structure subjected to a constant force system take the following form**

$$[K] \{ u \} = \{ P \}$$

- **Under loading, the structure deforms and internal loads are developed within the structure. Write the equilibrium equations for this deformed state:**

$$([K] + [K_D])\{ u^* \} = \{ P \}$$

  - The matrix $[K_D]$ is the differential stiffness matrix(i.e., geometric stiffness matrix or the stress stiffness matrix).
  - The differential stiffness is the stiffness $[K_D]$ that results from including the higher-order terms of the strain-displacement relations. These relations are assumed to be independent of the displacements of the structure associated with an arbitrary intensity of load.

MSC Software

THEORY OF BUCKLING (Cont.)

- What is "**differential stiffness**"?

  - It's the phenomena of Stress Stiffening or Stress Softening in the structure from the applied load.

  - What happens when an Axial Load is acting on a beam? The Stiffness no longer will remain same, instead decreases. The reduced stiffness will reduce the natural frequency and period of elongation.

  - For example, Compressing an Euler Beam causes "stress softening" which leads to instability/buckling

- **Let λ be an arbitrary scalar multiplier for another "intensity" of load**

$$([\ K\ ] + \lambda[\ K_D\ ])\ \{\ u^*\ \} = \lambda\{\ P\ \}$$

- **By perturbing the structure slightly at a variety of load intensities, the load intensities that possess unstable equilibrium positions can be found. This leads to the associated eigenvalue problem for buckling.**

$$([\ K\ ] + \lambda[\ K_D\ ])\ \{\ \delta u^*\ \} = 0$$

MSC Software

SOLUTION OF THE EIGENVALUE PROBLEM

$$[\ K - \lambda K_D\ ]\ \{\phi\} = 0 \ldots\ldots\ldots\ldots(Eq.1)$$

- **The solution is nontrivial (different from zero) only for specific values of $\lambda = \lambda_i$ for i = 1, 2, 3,…, n**

- **For a non-trivial solution, $|\ K - \lambda K_D\ | = 0$**

- **This is a classical eigenvalue problem.** $([K]-\lambda[I])\{x\}=\{0\}.$

- **We are searching for the eigenvalues (λ) of the stiffness matrix [K]. These eigenvalues cause the stiffness matrix to become singular**

  – Singular stiffness matrix means it has a zero value, i.e., the determinant of the matrix is equal to zero.

  – So, buckling is an "eigenvalue problem" that is a function of the material & geometric stiffness matrices. Consequently, there will be a number of buckling modes and corresponding mode shapes.

SOLUTION OF THE EIGENVALUE PROBLEM (Cont.)

- **For each eigenvalue λ_i, there is a corresponding distinct eigenvector $\{\phi_i\}$ which represents the buckled shape.**

- **$\{\phi_i\}$ can be scaled by any constant multiplier and still be a solution to Equation 1.**

- **The components of $\{\phi_i\}$ are real numbers.**

- **The critical buckling loads for the structure are computed as**
$$\{P\}_{cr_i} = \lambda_i \{P\} \qquad \text{(smallest value of Pcr will govern!)}$$

- **Usually only the lowest eigenvalue λ_1 is of interest because it is associated with the lowest buckling load for the structure.**

- **The eigenvalue, λ, is also called the buckling load factor (BLF).**
  - A structure has buckled if the buckling analysis indicates that BLF ≤ 1.0

SOLUTION SEQUENCES FOR BUCKLING & STABILITY PROBLEM

- **SOL 105   Linear buckling**

- **SOL 106   (with PARAM,BUCKLE) Nonlinear buckling**

- **SOL 400   Nonlinear buckling, modern**

- **SOL 600   Nonlinear buckling, MARC**

- <u>**Limitations of SOL 105**</u>
  - In prebuckled configuration:
    - Deflections must be small
    - Stresses must be elastic (and linearly related to strain)

MSC Software

- **Note:**
  - SOL 105 may be applicable for structures with slight material imperfections or slightly eccentric loadings (i.e. load does not align with centroid producing a small degree of bending). Must use engineering judgment.
  - Same arguments hold for plate structures.

EXAMPLES OF NONLINEAR BUCKLING

- **<u>Highly Eccentrically Loaded Column</u>**

Large bending stresses as well as compressive axial
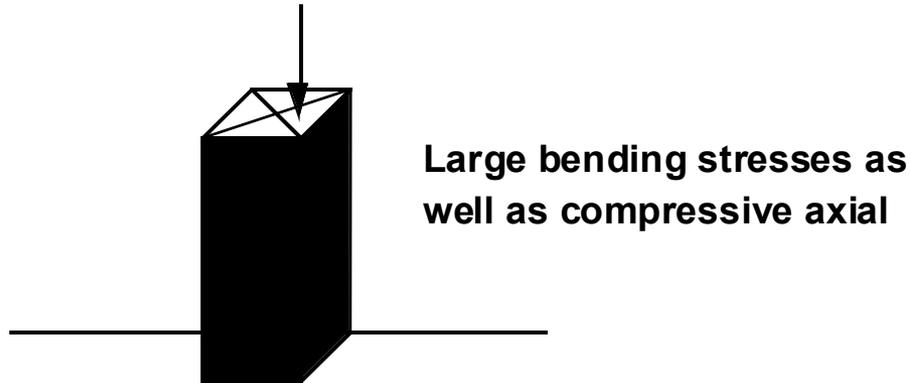
- **<u>Snap-Through of Thin Shell (like the Bottom of an Oil Can)</u>**

Large prebuckled deflection and possible inelastic prebuckled
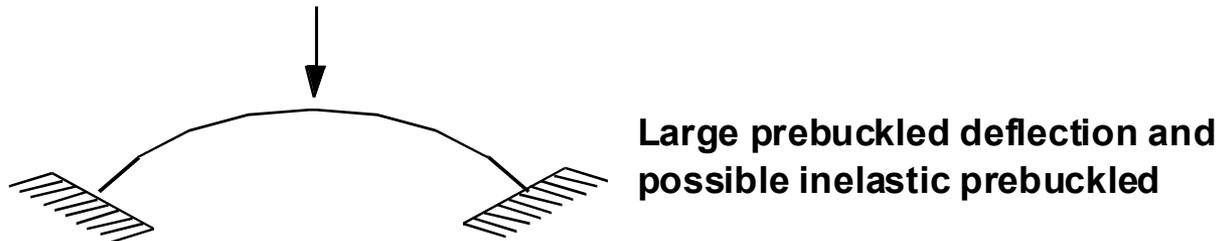
RULES FOR SOL 105 BUCKLING ANALYSIS

- **(For reference, see section 13 of the MSC Nastran Linear Static Analysis Users Guide)**
  - The Case Control must contain at least two subcases.
  - Normally, the first subcase is the static solution under loading.
  - METHOD must appear in a separate subcase to select an EIGB or EIGRL entry from the Bulk Data for the buckling solution.
  - If you have multiple static solutions, then use the STATSUB command to select the static subcase for the buckling solution.

MSC Software

RULES FOR SOL 105 BUCKLING ANALYSIS (Cont.)

- **If desired, different SPC sets may be applied in the static subcase and the buckling subcase.**

- **Output requests may be placed in any selected subcase.**

- **Output requests that apply to both, the static solution and the buckling modes, may be placed above the subcase level.**

MSC Software®

DATA ENTRIES FOR LINEAR BUCKLING

- **<u>Executive Control Section</u>**

  SOL 105

- **<u>Case Control Section</u>**

SUBCASE 1
  LOAD = M   **Defines static loading condition (LOAD, TEMP, DEFORM)**

SUBCASE 2
  METHOD = N   **Selects eigenvalue extraction method**
  STATSUB = i   **Selects static subcase to use for buckling solution (defaults to first subcase)**

DATA ENTRIES FOR LINEAR BUCKLING (Cont.)

- **The Case Control must contain at least two subcases.**

- **Bulk Data Section**

  Static loading condition required

EIGB	**Eigenvalue extraction data entry**
or	
EIGRL	**Eigenvalue extraction data for Lanczos method**

- **In buckling analysis, only Lanczos, INV and SINV are available**

EIGRL ENTRY

- **EIGRL Entry - <u>recommended eigenvalue solution method</u>**

  Defines data needed to perform real eigenvalue or buckling analysis with the Lanczos Method.

1	2	3	4	5	6	7	8	9	10
EIGRL	SID	V1	V2	ND	MSGLVL	MAXSET	SHFSCL	NORM	
EIGRL	1	0.1	3.2	10					

MSC Software

EIGRL ENTRY (Cont.)

Field	Contents
SID	Set identification number (unique integer > 0)
V1, V2	Vibration analysis: Frequency range of interest
	Buckling analysis: range of interest (V1 < V2, real). If all modes below a frequency are desired , set V2 to the desired frequency and leave V1 blank. It is not recommended to put 0.0 for V1, it is more efficient to use a small negative number or to leave it blank.
ND	Number of roots desired (integer > 0 or blank)
MSGLVL	Diagnostic level (integer 0 through 3 or blank)
MAXSET	Number of vectors in block (integer 1 through 15 or blank)

MSC Software

EXAMPLE - SIMPLE EULER COLUMN

- **Problem**

Find the critical load and corresponding first buckling mode shape of a solid circular rod.

P_{cr}

Free

L

Fixed
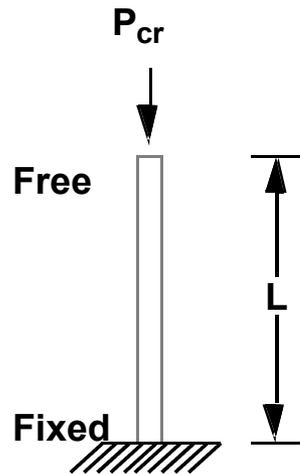
Solid Circular Cross Section

diameter	=	0.25 inches
E	=	30×10^6 psi
ν	=	0.33
I	=	1.917E-4 in^4
A	=	4.909E-2 in^2
L	=	21 in

102 STRUCTURAL STEEL DESIGN

Table 5-1   Effective Lengths for Main Members Only

	(a)	(b)	(c)	(d)	(e)	(f)
Buckled shape of column is shown by dashed line						
Theoretical K value	0.5	0.7	1.0	1.0	2.0	2.0
Recommended design value when ideal conditions are approximated	0.65	0.80	1.2	1.0	2.10	2.0

End condition code

Rotation fixed and translation fixed
Rotation free and translation fixed
Rotation fixed and translation free
Rotation free and translation free

MSC Software

EXAMPLE - SIMPLE EULER COLUMN (Cont.)

- **<u>Theoretical Solution</u>**

  where L_{eff} = Effective column length

  = 2L for free-fixed column

$$P_{cr} = \frac{\pi^2 EI}{L_{eff}^2}$$

– With

L_{eff} = Effective Length

= 2L = 2 X 21 = 42

E = 30.E+06

I = 1.917E-04
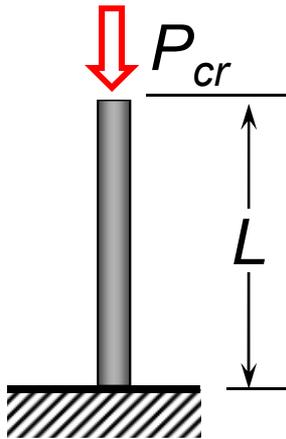
$$P_{cr} = \frac{\pi^2 (30.E+6)(1.917E-4)}{(2 \times 21)^2}$$

$$= 32.18$$

MSC Software

EXAMPLE - SIMPLE EULER COLUMN (Cont.)

- **MSC Nastran Model**
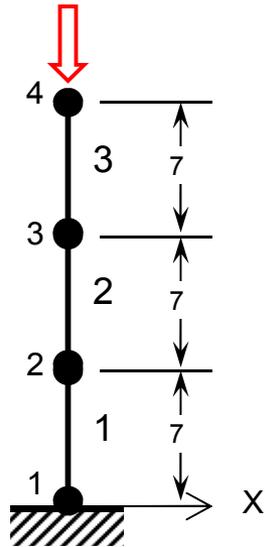
MSC/NASTRAN Solution

Load Value on Force Entry

$P_{cr} = 32.18 \times 1.0 = \underline{32.18 \text{ lbs}}$

Eigenvalue

EXAMPLE - SIMPLE EULER COLUMN – INPUT FILE

```
SOL 105
CEND
TITLE = BUCKLING OF FIXED-FREE BEAM
DISP=ALL
SPC = 10
ECHO = punch
SUBCASE 1
    SUBTITLE=STATIC SUBCASE
    LOAD = 5
SUBCASE 2
    SUBTITLE=BUCKLING SUBCASE
    METHOD = 100
BEGIN BULK
CBEAM      1         1         1         2         0.        0.        1.
CBEAM      2         1         2         3         0.        0.        1.
CBEAM      3         1         3         4         0.        0.        1.
FORCE      5         4         0         1.        -1.       0.        0.
EIGRL      100                           1         0
GRID       1                   0.        0.        0.                  345
GRID       2                   7.        0.        0.                  345
GRID       3                   14.       0.        0.                  345
GRID       4                   21.       0.        0.                  345
SPC1       10        123456    1
MAT1       2         3.+7                .33
PBEAM      1         2         .04909    1.917-4 1.917-4             3.835-4
ENDDATA
```

MSC Software

EXAMPLE - SIMPLE EULER COLUMN – OUTPUT FILE

First eigenvalue: $P_{cr} = \lambda_1 \times 1 \text{ lbs} = 32.18 \text{ lbs}$

First eigenvector (buckled shape)

MSC Software

MULTIPLE BUCKLING ANALYSES IN A SINGLE RUN

- Multiple buckling solution and SUBCASEs set up

  - All static subcases must appear first

  - The buckling subcases follow the last static subcase

  - A METHOD entry must appear in each of the buckling subcases

  - Each buckling subcase must contain a STATSUB command that references the appropriate subcase ID of the static subcase

- Use STATSUB in buckling SUBCASE to point to desired load for $[K_d]$

```
SUBCASE 100
    LOAD = 1
SUBCASE 200
    LOAD = 2
SUBCASE 300
    LOAD = 3
$ -------------
SUBCASE 1200
   STATSUB = 200
    METHOD =    5
$
SUBCASE 1300
   STATSUB = 300
    METHOD =    5
```

MSC Software

MULTIPLE BUCKLING ANALYSES IN A SINGLE RUN (contd.)

- **The third subcase (Subcase 11) is for a lateral buckling analysis. The Case Control command (STATSUB = 2 in Subcase 11) tells MSC Nastran that you want to generate the differential stiffness matrix from the first static subcase (Subcase 2).**

- **The fourth subcase (Subcase 21) is for a Euler beam buckling analysis. The Case Control command (STATSUB = 5 in Subcase 21) tells MSC Nastran that you want to generate the differential stiffness matrix from the second static subcase (Subcase 5)**

```
$
SUBCASE 2
LABEL = CANTILEVER BEAM
LOAD = 10
SPC = 10
$
SUBCASE 5
LABEL = SIMPLY SUPPORTED BEAM
LOAD = 20
SPC = 20
$
SUBCASE 11
LABEL = LATERAL BUCKLING OF CANTILEVER BEAM
METHOD = 10
SPC = 10
STATSUB = 2
$
SUBCASE 21
LABEL = EULER BUCKLING OF SIMPLY SUPPORTED BEAM
METHOD = 10
SPC = 20
STATSUB = 5
$
$BEGIN BULK
$
```

MSC Software

BUCKLING OF PRELOADED STRUCTURE

- **MSC Nastran allows option of having preload in addition to buckling analysis**

- **For preload, the preload keyword is added to the STATSUB command**
  - STATSUB(PRELOAD) = x

- **For buckling, the buckle keyword is added to the STATSUB command**
  - STATSUB(BUCKLE) = y

- **The default keyword is buckle for the STATSUB case control command**

- **The example to the right shows the setup to include both preload and buckling calculations. The following case control commands are needed**
  - STATSUB (PRELOAD) = x
  - STATSUB (BUCKLE) = y

```
SOL 105
CEND
TITLE = Buckling of Preloaded Structure
SPC = 2
SUBCASE 1
    TITLE=PRELOAD
    LOAD = 2
SUBCASE 2
    TITLE=BUCKLING LOAD
    LOAD = 3
SUBCASE 3
    SUBTITLE=Buckling + preload
    STATSUB(PRELOAD) = 1
    STATSUB(BUCKLE) = 2
    METHOD = 1
    VECTOR(SORT1,REAL)=ALL
BEGIN BULK
PARAM   COUPMASS 1
eigrl,1,,,10,,,,MAX
CBEAM   1       1       1       2       0.      1.      0.
. . . . .
. . . . .
LOAD    2       1.      1.      1
SPC1    1       123456  1
FORCE   1       11      0       0.1     0.      1.      0.
LOAD    3       1.      1.      1
ENDDATA
```

MSC Software

BUCKLED SHAPE NORMALIZATION

- **Open the MSC Nastran *.f06 file with DISP=ALL output request and look for the number 1.000000.**

- **Is this a rotation or a translation? It must be a translation. If it is a rotation, the mesh may be too coarse and you need to refine the local mesh.**

- **When looking at the buckled shape one has to always look at translation value.**

- **In the example of Euler Buckling, Grid 4 has T2=1.000000E+00.**

- **In buckling solutions, Maximum Displacement is normalized to unity.**

MSC Software

REFERENCES FOR BUCKLING AND STABILITY ANALYSIS

- **MSC Nastran Linear Static Analysis Users Guide, Section 13.**

WORKSHOP

- Learn linear Buckling Analysis by performing Workshop 1

MSC Software

LAGRANGE RIGID ELEMENTS

- **For rigid element, the default linear formulation does not support differential stiffness or buckling analysis.**

- **The Lagrange rigid element supports**
  - differential stiffness
  - buckling

MSC Software

RIGID ELEMENT ENHANCEMENTS

- **<u>Example</u>**

  - Buckling analysis of the following model using 2 different approaches
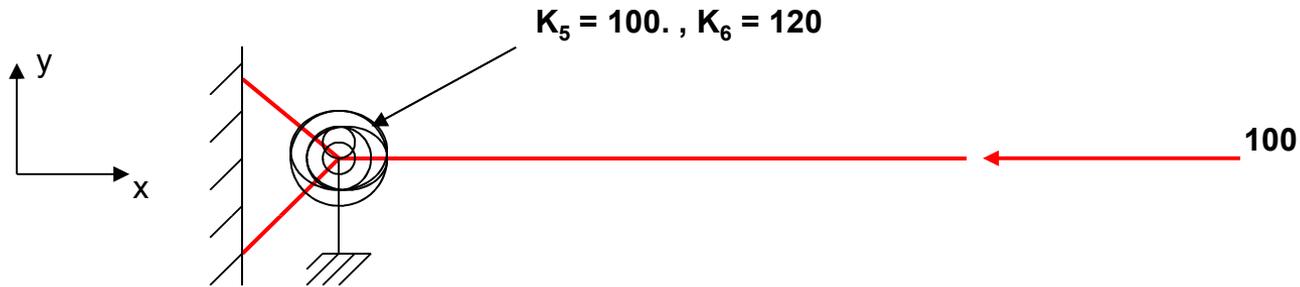
  1. Lagrange Rigid Element
  2. Stiff beam

RIGID ELEMENT ENHANCEMENTS (Cont.)

INPUT FILE USING STIFF BEAM

```
$
$   lagrange2.dat
$
SOL 105
CEND
TITLE = BUCKLING ANALYSIS - USE STIFF BEAM
DISP = ALL
SPC  = 10
SUBCASE 1
LABEL=STATIC PRELOAD CASE
  LOAD = 100
SUBCASE 2
LABEL = BUCKLING CASE
  METHOD = 10
BEGIN BULK
EIGRL,10,,,10
FORCE,100,4 ,0,-100.0,1.0,0.0,0.0
CELAS2,101,100.0,3,5
CELAS2,102,120.0,3,6
CBEAM,100,100,3,4,0.,0.,1.
PBEAM,100,1,100.,100.,100.,,100.
MAT1,1,1.E7,,.32
GRID, 3 ,,2.0,0.0,0.0
GRID, 4 ,,4.0,0.0,0.0
SPC1,10,1234,3
ENDDATA
```

INPUT FILE USING LAGRANGE RIGID ELEMENT

```
$
$   lagrange1.dat
$
SOL 105
CEND
TITLE = BUCKLING ANALYSIS - RBAR
SUBTI = LAGRANGE ELIMINATION METHOD
DISP = ALL
SPC  = 10
RIGID = LGELIM
SUBCASE 1
LABEL=STATIC PRELOAD CASE
  LOAD = 100
SUBCASE 2
LABEL = BUCKLING CASE
  METHOD = 10
BEGIN BULK
EIGRL,10,,,10
FORCE,100,4 ,0,-100.0,1.0,0.0,0.0
CELAS2,101,100.0,3,5
CELAS2,102,120.0,3,6
GRID, 3 ,,2.0,0.0,0.0
GRID, 4 ,,4.0,0.0,0.0
RBAR, 3,3,4,123456, , ,123456
SPC1,10,1234,3
ENDDATA
```

RIGID ELEMENT ENHANCEMENTS (Cont.)

- **<u>Output Using Lagrange Rigid Element</u>**

```
0    STATIC PRELOAD CASE                                                        SUBCASE 1
                          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       0.0           0.0           0.0           0.0           0.0           0.0
         4        G       0.0           0.0           0.0           0.0           0.0           0.0
0    BUCKLING CASE                                                              SUBCASE 2

                          R E A L   E I G E N V A L U E S
     MODE   EXTRACTION      EIGENVALUE         RADIANS          CYCLES         GENERALIZED      GENERALIZED
     NO.      ORDER                                                              MASS           STIFFNESS
       1        1         5.000000E-01       7.071068E-01     1.125395E-01     2.000000E+02     1.000000E+02
       2        2         6.000000E-01       7.745967E-01     1.232809E-01     2.000000E+02     1.200000E+02
1    BUCKLING ANALYSIS - RBAR                                    MARCH   2, 2012  MSC.NASTRAN 11/25/11    PAGE    12
     LAGRANGE ELIMINATION METHOD
0    BUCKLING CASE                                                              SUBCASE 2
     EIGENVALUE =  5.000000E-01
                          R E A L   E I G E N V E C T O R   N O.        1

     POINT ID.   TYPE      T1            T2            T3            R1            R2            R3
         3        G       0.0           0.0           0.0           0.0         1.000000E+00   1.109505E-16
         4        G       0.0         2.219010E-16  -2.000000E+00   0.0         1.000000E+00   1.109505E-16
1    BUCKLING ANALYSIS - RBAR                                    MARCH   2, 2012  MSC.NASTRAN 11/25/11    PAGE    13
     LAGRANGE ELIMINATION METHOD
```

- **<u>Output Using stiff beam</u>**

```
0    BUCKLING CASE                                                              SUBCASE 2
                          R E A L   E I G E N V A L U E S
     MODE   EXTRACTION      EIGENVALUE         RADIANS          CYCLES         GENERALIZED      GENERALIZED
     NO.      ORDER                                                              MASS           STIFFNESS
       1        1         4.999978E-01       7.071052E-01     1.125393E-01     5.000000E+01     2.499989E+01
       2        2         5.999833E-01       7.745859E-01     1.232792E-01     5.000000E+01     2.999917E+01
       3        3         1.893939E+06       1.376205E+03     2.190298E+02     1.000000E+02     1.893939E+08
       4        4         1.681615E+07       4.100750E+03     6.526545E+02     3.999999E+01     6.726456E+08
       5        5         1.681615E+07       4.100750E+03     6.526545E+02     3.999998E+01     6.726456E+08
       6        6         3.000000E+07       5.477226E+03     8.717275E+02     6.666664E+01     1.999999E+09
       7        7         3.000000E+07       5.477226E+03     8.717275E+02     6.666663E+01     1.999999E+09
1    BUCKLING ANALYSIS - USE STIFF BEAM                          MARCH   2, 2012  MSC.NASTRAN 11/25/11    PAGE    12

0    BUCKLING CASE                                                              SUBCASE 2
     EIGENVALUE =  4.999978E-01
                          R E A L   E I G E N V E C T O R   N O.        1

     POINT ID.   TYPE      T1            T2            T3            R1            R2            R3
         3        G       0.0           0.0           0.0           0.0        -5.000000E-01  -3.223676E-05
         4        G     -3.611736E-11 -6.468463E-05  1.000000E+00  2.344971E-11 -5.000001E-01  -3.223808E-05
1    BUCKLING ANALYSIS - USE STIFF BEAM                          MARCH   2, 2012  MSC.NASTRAN 11/25/11    PAGE    13
```

MSC Software

SECTION 3

UNDERSTANDING MPCs, CONNECTORs AND R-ELEMENTS

CONTENTS

MSC Software

CONTENTS

RBEs and MPCs

- **Not necessarily "rigid" elements**
  - Working Definition:

The motion of a DOF is dependent on the <u>motion</u> of at least one other DOF

MSC Software®

MOTION AT ONE GRID DRIVES ANOTHER

- **Simple Translation**
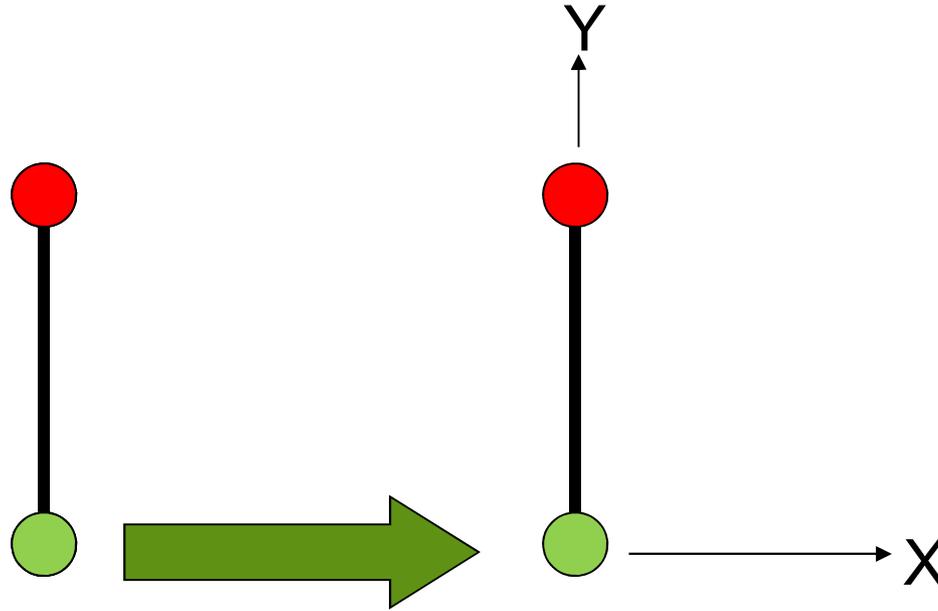
X motion of Green Grid drives X motion of Red Grid

MSC Software®

MOTION AT ONE GRID DRIVES ANOTHER (Cont.)

- **Simple Rotation**
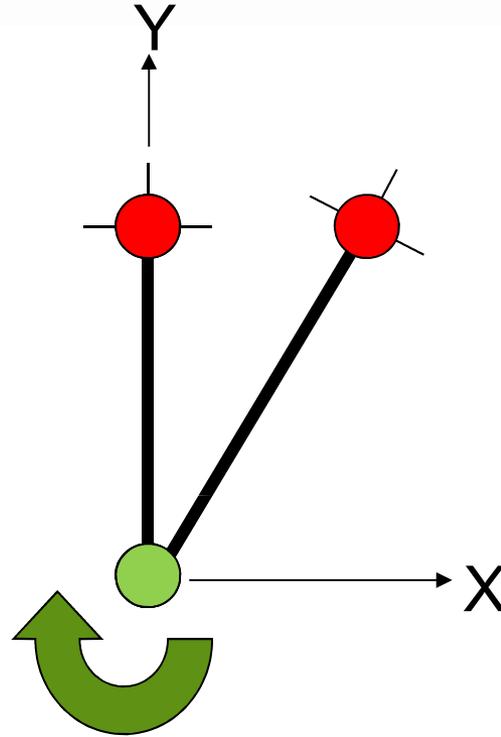
Rotation of Green Grid drives X translation and Z rotation
    of Red Grid

RBEs and MPCs

> *The motion of a DOF is dependent on the <u>motion</u> of at least one other DOF*

- **Displacement, not elastic relationship**
- **<u>Not</u> dictated by stiffness, mass, or force**
- **Linear relationship**
- **Small displacement theory**
- **Dependent v. Independent DOFs**
- **Stiffness/mass/loads at dependent DOF transferred to independent DOF(s)**

SMALL DISPLACEMENT THEORY & ROTATIONS

- **Small displacement theory:**

$$\sin(\Phi) = \tan(\Phi) = \Phi$$
$$\cos(\Phi) = 1$$

- **For Rz @ A**

$$Rz_B = Rz_A = \Phi$$
$$Tx_B = (-\Phi)*L_{AB}$$
$$Ty_B = 0$$

MSC Software®
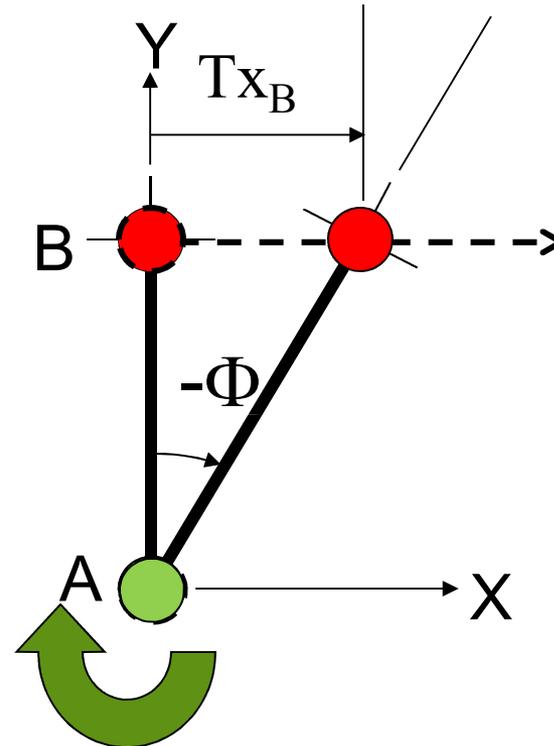
TYPICAL "RIGID" ELEMENTS IN MSC NASTRAN

- **Geometry-based**
  - RBAR
  - RBE2 } *Really*-rigid "rigid" elements

- **Geometry- & User-input based**
  - RBE3 → Interpolation Element

- **User-input based**
  - MPC

COMMON GEOMETRY-BASED RIGID ELEMENTS

- **RBAR**
  - Rigid Bar with six DOF at each end
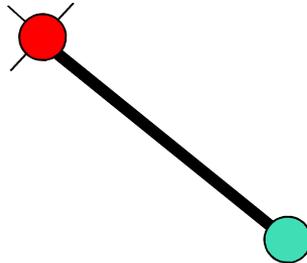
- **RBE2**
  - Rigid body with independent DOF at one GRID, and dependent DOF at an arbitrary number of GRIDs.
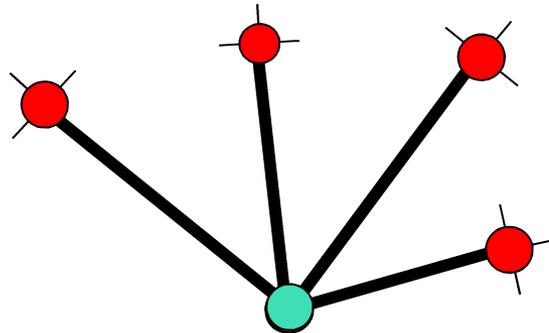
The RBAR

- **The RBAR is a rigid link between two GRID points**
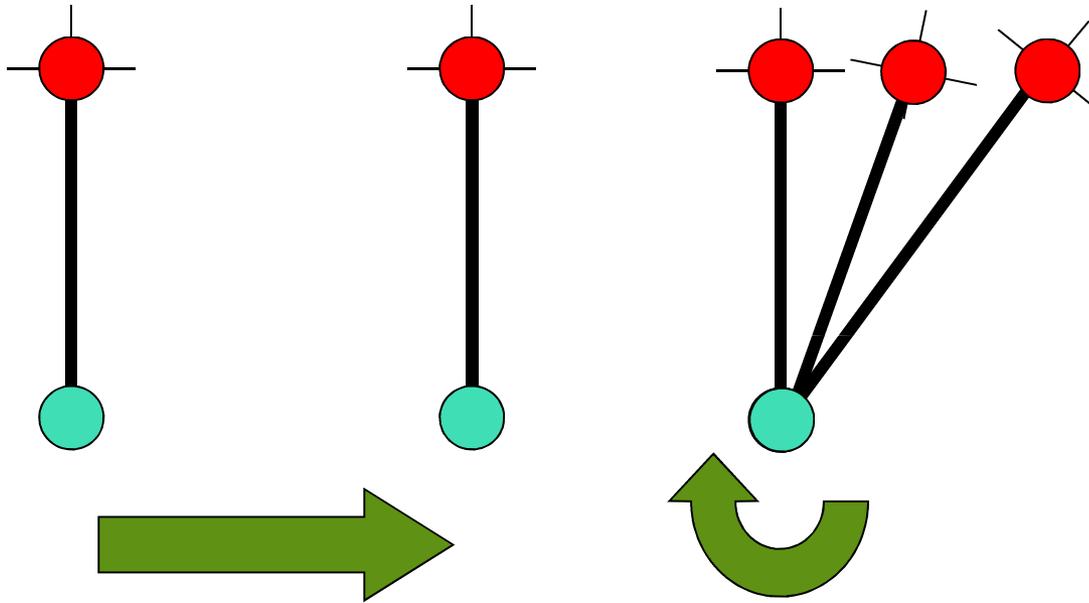
The RBAR (Cont.)

- Most common to have all the <u>dependent</u> DOFs at one GRID, and all the <u>independent</u> DOFs at the other
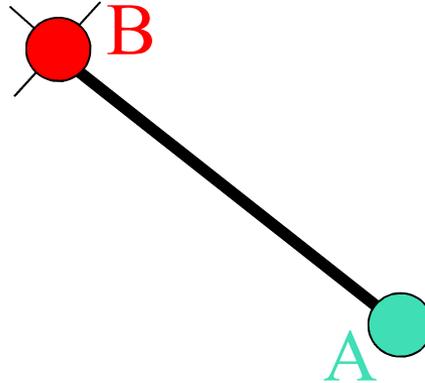
- **Can mix/match dependent DOF between the GRIDs, but this is rare**

- **The independent DOFs <u>must</u> be capable of describing the rigid body motion of the element**

1	2	3	4	5	6	7	8	9	10
RBAR	EID	GA	GB	CAN	CNB	CMA	CMB		
RBAR	535	1	2	123456			123456		

MSC Software®

RBAR EXAMPLE: FASTENER

- **Use of RBAR to "weld" two parts of a model together:**

1	2	3	4	5	6	7	8	9	10
RBAR	EID	GA	GB	CAN	CNB	CMA	CMB		
RBAR	535	1	2	123456			123456		
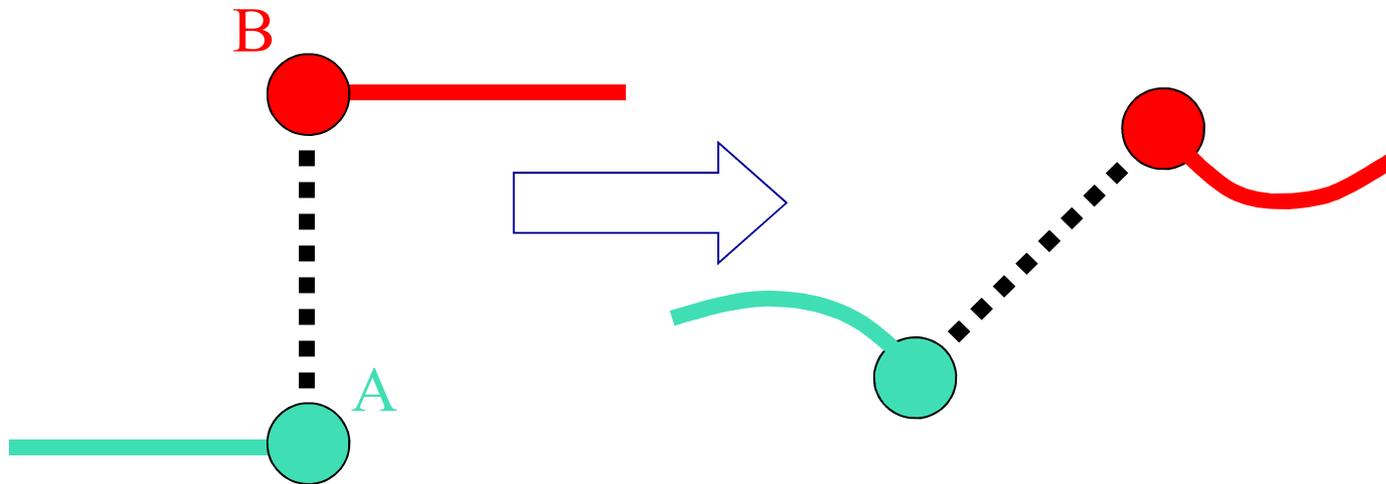
RBAR EXAMPLE: PIN-JOINT

- **Use of RBAR to form pin-jointed attachment**

1	2	3	4	5	6	7	8	9	10
RBAR	EID	GA	GB	CAN	CNB	CMA	CMB		
RBAR	535	1	2	123456			123		

MSC Software®
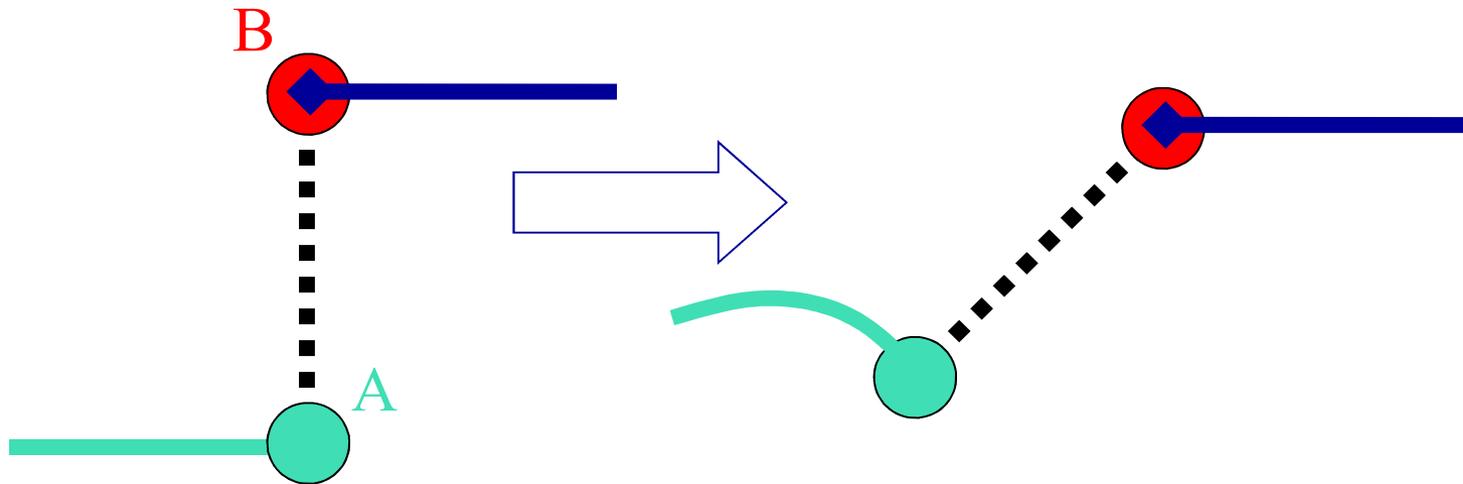
MULTI-POINT CONSTRAINTS (MPC)

MSC Software

MULTIPOINT CONSTRAINTS (MPC)

- **Each MPC entry is used to specify one displacement (U_m) as a linear combination of one or more other displacements(U_n).**

- **MSC Nastran divides the G-set into 2 sets.**

  M = dependent DOFs,

  N = independent DOFs

- **Then performs the reduction from the G to N set.**

MULTIPOINT CONSTRAINTS (MPC) (Cont.)

- **General form for MPC equations:**

$$A_{M_i} U_{M_i} + \sum_j A_{N_j} U_{N_j} = 0$$

where

$$A_{M_i} = \quad \text{Scaling coefficient for the dependent DOFs}$$

$$A_{N_j} = \quad \text{Scaling coefficient for the independent DOFs}$$

$$U_{M_i} = \quad \text{Displacement of dependent DOFs}$$

$$U_{N_j} = \quad \text{Displacement of independent DOFs}$$

- **The equations for all MPCs and R-type elements are assembled to form the constraint equations:**

$$R_M U_M + R_N U_N = 0$$

MSC Software®

MULTIPOINT CONSTRAINTS (MPC) (Cont.)

- – This can be written as

$$U_M = \underbrace{- R_M^{-1} R_N} U_N = G_M U_N$$

Therefore, R_M must not be singular

- **The G-set matrices are rewritten as follows:**

$$\{U_G\} = \left\{ \begin{matrix} U_M \\ U_N \end{matrix} \right\} = \left[\begin{matrix} G_{MN} \\ I_{NN} \end{matrix} \right] U_N$$

- **As explained in The MSC NASTRAN REFERENCE GUIDE, Section 9.4.3, the equation**

$$\lfloor \mathrm{K}_{gg} \rfloor \lfloor \mathrm{U}_g \rfloor = \lfloor \mathrm{P}_g \rfloor$$

$$= \left[\begin{matrix} \overline{k}_{NN} & k_{NM} \\ k_{NM}^T & k_{MM} \end{matrix} \right] \left\{ \begin{matrix} U_N \\ U_M \end{matrix} \right\} = \left\{ \begin{matrix} \overline{P}_N \\ P_M \end{matrix} \right\}$$

MULTIPOINT CONSTRAINTS (MPC) (Cont.)

Becomes

$$[K_{NN}][U_N] = [P_N]$$

where

$$[k_{NN}] = [\bar{k}_{NN} + k_{NM}G_M + G_M^T k_{NM}^T + G_M^T k_{MM}G_M]$$
$$\{P_N\} = \{\bar{P}_N + G_M^T P_M\}$$

MSC Software®

MPC – BULK DATA ENTRY

Defines a multipoint constraint equation of the form

$$\sum_{j} A_j\, u_j = 0$$

where u_j represents degree of freedom Cj at grid scalar point Gj

1	2	3	4	5	6	7	8	9	10
MPC	SID	G1	C1	A1	G2	C2	A2		MPC
		G3	C3	A3	-etc.-				
MPC	3	28	3	6.2	2	4.29			
		1	4	-2.91					

Field	Contents
SID	Set identification number: (Integer > 0)
Gj	Identification number of grid or scalar point: (Integer > 0)
Cj	Component number. (Any one of the integer 1 through 6 for grid points; blank or zero for scalar point).
Aj	Coefficient (Real; Default = 0.0 except A1 must be nonzero).

MPC – BULK DATA ENTRY (Cont.)

- **<u>Remarks:</u>**

  1. Multipoint constraint sets must be selected with the Case Control command MPC = SID.

  2. The first degree of freedom (G1, C1) in the sequence is defined to be the dependent degree of freedom assigned by one MPC entry cannot be assigned dependent by another MPC entry or by a rigid element.

  3. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.

  4. The m-set degrees of freedom specified on this entry may not be specified on other entries that define mutually exclusive sets. See refer the *MSC NASTRAN Quick Reference Guide*, for a list of these entries.

  5. By default, the grid point connectivity created by the MPC, MPCADD, and MPCAX entries is not considered during resequencing, (see the PARAM,OLDSEQ description in *MSC NASTRAN Quick Reference Guide*. In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,3.

MULTIPOINT CONSTRAINT EXAMPLE

1. Thick plate with bars attached.
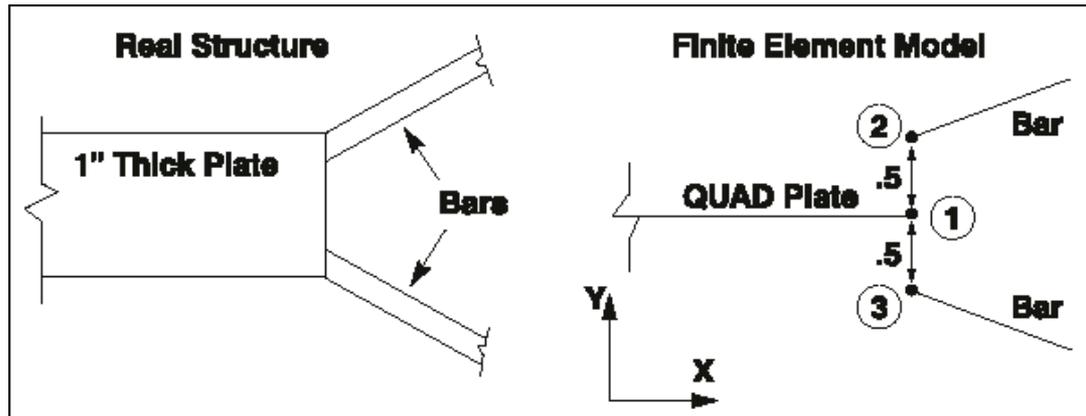
Using plate theory assumption ("plane sections remain plane"), we can write the equations for the in-plane motion of Grid Points 2 and 3 as function of the motion of Grid point 1.

$$[Note:\ u_{1_2} =\ disp.\ comp.\ 1\ at\ grid\ 2]$$

$$u_{1_2} = u_{1_1} - .5 * u_{6_1} \qquad u_{2_2} = u_{2_1}\ ;\ u_{6_2} = u_{6_1}$$

$$u_{1_3} = u_{1_1} + .5 * u_{6_1} \qquad u_{2_3} = u_{2_1}\ ;\ u_{6_3} = u_{6_1}$$

MULTIPOINT CONSTRAINT EXAMPLE (Cont.)

MPC entries are:

```
MPC, 1,      2, 1, 1.,   1, 1,  -1.,  ,+MPC1A
+MPC1A, ,  1, 6,  .5
MPC, 1,     3, 1, 1.,   1, 1, -1.,  ,+MPC1B
+MPC1B, ,  1, 6, -.5
MPC, 1,     2, 2, 1.,   1, 2, -1.
MPC, 1,     3, 2, 1.,   1, 2, -1.
MPC, 1,     2, 6, 1.,   1, 6, -1.
MPC, 1,     3, 6, 1.,   1, 6, -1.
```

Notes:

- Select MPC = 1 in the Case Control to use these entries.

- The MPC equations are written using the displacement coordinate system of the GRID points. If GRID points involved in MPC have different coordinate system, be careful, to avoid "grounding" the structure.

MSC Software®

MULTIPOINT CONSTRAINT EXAMPLE (Cont.)

- **1a. If GRID 1 has the following displacement coordinate system (CID =1), and GRID 2 and 3 have the basic displacement coordinate system, the MPC equations would look like:**

```
CORD2R, 1,  , 0., 0., 0.,  0., 0., 1.,
 , 0., 1., 0.
GRID,   1,  , 4., .5, 0.,  1  $ CD = 1
MPC, 1,   2, 1,  1.,    1, 2, 1., ,+MPC1A
+MPC1A, , 1, 6,   0.5
MPC, 1,   3, 1,  1.,    1, 2, 1., ,+MPC1B
+MPC1B, , 1, 6,  -0.5
$
MPC, 1,   2,  2,  1., 1,  1,  -1.
MPC, 1,   3,  2,  1., 1,  1,  -1.
$
MPC, 1,   2,  6,  1., 1,  6,  -1.
MPC, 1,   3,  6,  1., 1,  6,  -1.
```

MSC Software®
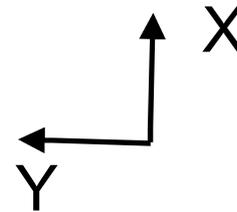
MULTIPOINT CONSTRAINT EXAMPLE (Cont.)

2. Calculate the distance between two points:

Example:

There is a tolerance requirement on a structure.   We want to know the clear distance between two specified points.
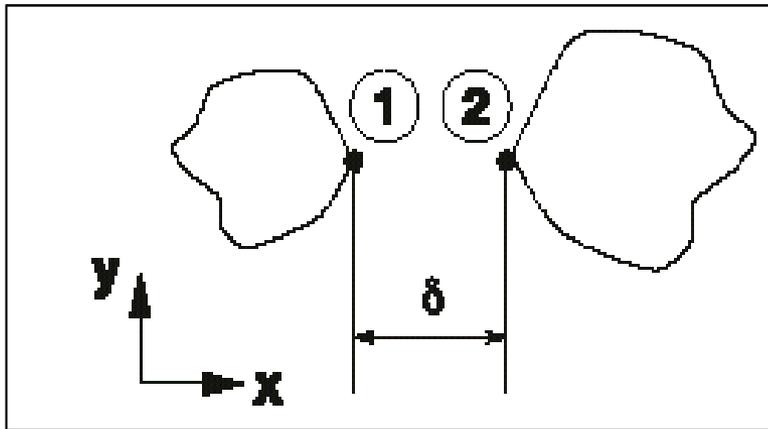
$\delta_o = 10.0 = $ Initial clearance

MSC Software®

MULTIPOINT CONSTRAINT EXAMPLE (Cont.)

Total distance $\delta = \delta_o + u_{1_2} - u_{11}$

Use a scalar point to represent the distance δ.

```
$ Component 1 of node 1001 is SPC'ed to 10.0 (the
$ original distance between node 1 and node 2)
GRID, 1001
SPC,    1,   1001, 1, 10.0
$ Create a scalar variable to hold (final) distance
$ between node 1 and node 2
SPOINT, 1000
$ Set MPC:  Ux1000 = Ux1001 + Ux2 - Ux1
MPC, 10,  1000, 0, 1.,   1001, 1, -1.,   ,+MPC10A
+MPC10A, , 2, 1, -1.,  1, 1, 1.
```

*   Call out MPC = 10   in the Case Control Section

** Call out SPC = 1   in the Case Control Section

MULTIPOINT CONSTRAINT EXAMPLE (Cont.)

3. Average displacement

$$\Delta = \frac{u_{1_1} + u_{1_2}}{2}$$

To get the average displacement of Grid Points 1 and 2 of the previous example, add the following:

```
$ Create a scalar variable (1002) to hold the average
$ distance between node 1 and node 2
SPOINT, 1002
$
MPC, 10,  1002, 0, 1.,  1, 1, -0.5,  ,+MPC10B
+MPC10B, , 2, 1, -0.5
```

MULTIPOINT CONSTRAINT EXAMPLE (Cont.)

4 **Enforce a relative gap.**

- **In order to do this, we wish to constrain the SPOINT to have the desired gap. Therefore, the SPOINT must be independent on the MPC and we need to re-write the MPC entries.**

```
$ Create a scalar variable (1000) to hold the final
$ distance (clearance) between node 1 and node 2
SPOINT, 1000
$   Relative gap (final) =  (Ux2 - Ux1) + gap (initial)
$   MPC Eq.:   Ux1000 =  Ux2 - Ux1 + Ux1001  input as:
$  -Ux1 +Ux2 +Ux1001 -Ux1000 =  0.
MPC, 10,  1, 1, -1.,            2, 1,  1.,  ,+MPC10A
+MPC10A, , 1001, 1, 1.,   1000, 0, -1.
$
SPC,    1,   1001, 1, 0.02  $Initial gap
SPC,    1,   1000, 0, 0.001 $Final gap to be allowed
```

MULTIPOINT CONSTRAINTS EXAMPLE (Cont.)

- **MPC at Selective Mesh Refinement**
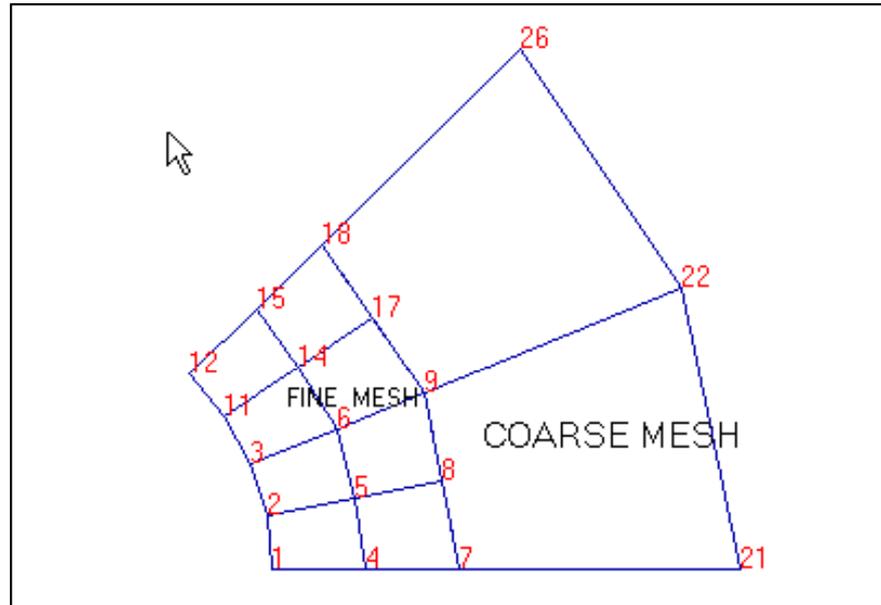
- $U_8 = 0.5*(U_7 + U_9)$; $U_{17} = 0.5*(U_9 + U_{18})$

MSC Software

"RTYPE" ELEMENTS

THE "RTYPE" ELEMENTS

- Generate internal MPC equations (for linear R-type element) that eliminate dependent degrees of freedom. (The user selects the dependent points.)

- Are automatically included in the solution (the "MPC =" command does not effect R–elements)

- "R" elements do not account for nonlinear and mass calculations.

- Internal forces and grid point forces are calculated for output.

(MPCFORCE case control request – output is by GRID point, not by R–element. The output at each GRID point is the summation from all R–elements and MPC's connected to it. The MPC Forces should sum to zero, if no structural element is attached at the grid.)

SAMPLE USES OF "RTYPE" ELEMENTS

- **When very stiff structure sections are inconvenient to model or numerically troublesome**

- **When different pieces of the model are mismatched and the grid points cannot be connected with conventional elements**

- **If connecting joints are free to slide and/or rotate in specific directions**

- **When elements are offset from grid points**

- **To distribute input loading or enforced motions**

- **To connect incompatible elements**

COMMONLY USED "RTYPE" ELEMENTS

- **RBAR "Rigid" bar connecting two grid points with 6 independent and 1-6 dependent DOFs**

- **RBE2 "Rigid element with six independent DOFs at one grid point and any number of dependent DOFs**

- **RBE3 "Interpolation" element with 1 – 6 <u>dependent</u> DOFs and any number of independent DOFs. Used for distributing loads or obtaining average displacement. No stiffness is introduced by RBE3.**

- **RSPLINE "Interpolation" element with any number of independent and dependent DOFs. Uses the displacement pattern of a beam element based on the independent DOFs to obtain displacements of the dependents DOFs.**

- **RSSCON "Interpolation" element used to connect shell elements to solid elements**

MSC Software

RBAR

- **RBAR connects to only two grid points with a total of six independent degrees of freedom.**

- **The independent DOF are selected by you, but must be able to define any motion is space.**

- **Simpler version of RBE1**

RBAR – BULK DATA ENTRY

- **Defines a rigid bar with six degrees of freedom at each end**

1	2	3	4	5	6	7	8	9	10
RBAR	EID	GA	GB	CNA	CNB	CMA	CMB	ALPHA	
RBAR	5	1	2	234	123			6.5-6	

Field	**Contents**
EID	Element identification number.
GA, GB	Grid point identification number of connection points. (Integer > 0).
CNA, CNB	Component numbers of independent degrees of freedom in the global coordinate system for the element at grid points GA and GB. See Remark 1. See Remarks 2 and 3. (Integer 1 through 6 with no embedded blanks, or zero or blank).
CMA, CMB	Component numbers of dependent degrees of freedom in the global coordinate system assigned by the element at grid points GA and GB. See Remarks 2 and 3. (Integers 1 through 6 with no embedded blanks, or zero or blank).
ALPHA	Thermal expansion coefficient. (Real > 0.0 or blank)

MSC Software®

RBAR – BULK DATA ENTRY (Cont.)

Remarks:

1. The total number of components in CNA and CNB must equal six; for example, CNA = 1236, CNB = 34. Furthermore, they must jointly be capable of representing any general rigid body motion of the element.

2. If both CMA and CMB are zero or blank, all of the degrees of freedom not in CNA and CNB will be made dependent; i.e., they will be made members of the m-set.

3. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. Se the *MSC NASTRAN Quick Reference Guide*, for a list of these entries.

4. Element identification numbers must be unique.

5. Rigid elements, unlike MPC's, are not selected through the Case Control Section.

6. Forces of multipoint constraint may be recovered in the linear structured solution sequences (101 – 200), with the MPCFORCE Case Control command.

7. For the Lagrange method, the thermal expansion effect will be computed for the rigid bar element if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as the average temperature of the two connected grid points GA and GB.

RBE2 – BULK DATA ENTRY

- **Defines a rigid body whose independent degrees of freedom are specified at a single grid point and whose dependent degrees of freedom are specified at an arbitrary number of grid points.**

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4	GM5	
RBE2	9	8	12	10	12	14	15	16	
	GM6	GM7	GM8	-etc.-	ALPHA				
	20	6.5-6							

Field	Contents
EID	Element identification number.
GN	Identification number of grid point to which all six independent degrees of freedom for the element are assigned. (Integer > 0)
CM	Component numbers of the dependent degrees of freedom in the global coordinates system at grid points Gmi.(Integers 1 through 6 with no embedded blanks).
Gmi	Grid point identification numbers at which dependent degrees of freedom are assigned. (Integer > 0).
ALPHA	Thermal expansion coefficient. (Real > 0.0 or blank)

MSC Software

RBE2 – BULK DATA ENTRY (Cont.)

Remarks:

1. Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.

2. For the Lagrange method, MSC Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees of freedom which is obtained by CM multiplied with the number of dependent grid points.

3. For the linear method, the dependent degrees-of-freedom indicated by CM will be made members of the m-set at all grid points. For the Lagrange method, they may or may not be members of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding the m-set described below apply to both types of methods.

4. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.

5. Element identification numbers should be unique with respect to all other element identification numbers.

6. Rigid elements, unlike MPCs, are not selected through the Case Control Section.

MSC Software

RBE2 – BULK DATA ENTRY (Cont.)

Remarks (Cont.):

7. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.

8. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation.

9. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See Degree-of-Freedom Sets, 1013 for a list of these entries.

10. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as follows: the temperature of the bar connecting the grid point GN and any dependent grid point are taken as the average temperature of the two connected grid points.

THE RBE2

- **One <u>independent</u> GRID (all 6 DOF)**

- **Multiple <u>dependent</u> GRID/DOFs**

Note: Small Rotations assumed

MSC Software
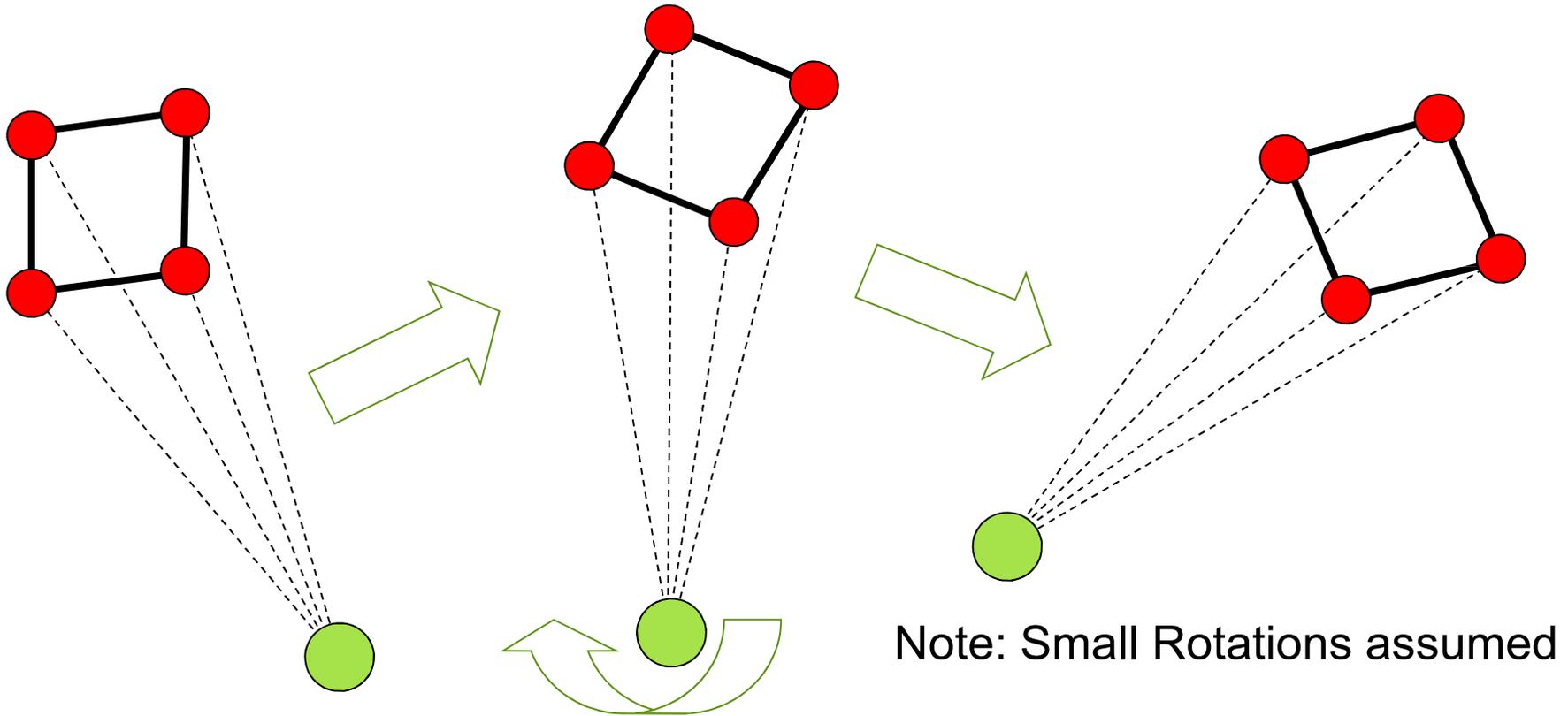
RBE2 EXAMPLE

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4	GM5	
RBE2	99	101	123456	1	2	3	4		

- **Note: <u>No</u> relative motion between GRIDs 1-4 !**
  - No deformation of element(s) between these GRIDs

MSC Software
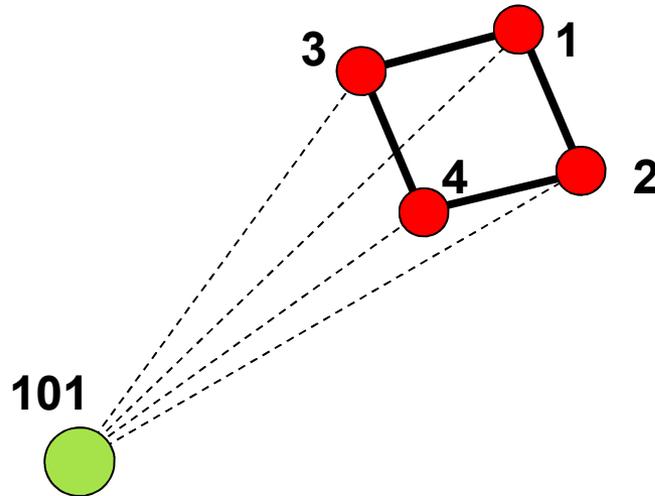
COMMON RBE2 USES

- **RBE2 between 2 GRIDs**
  - "Weld" two different parts together
    - 6 DOF Connection
  - "Ball Joint" two different parts together
    - 3 DOF connection

- **RBE2**
  - "Spider" or "wagon wheel" connections
  - Large mass/base-drive connection

RBE3 – THE "WIFFLETREE"

- **The RBE3 is an "interpolation" element. Example:**
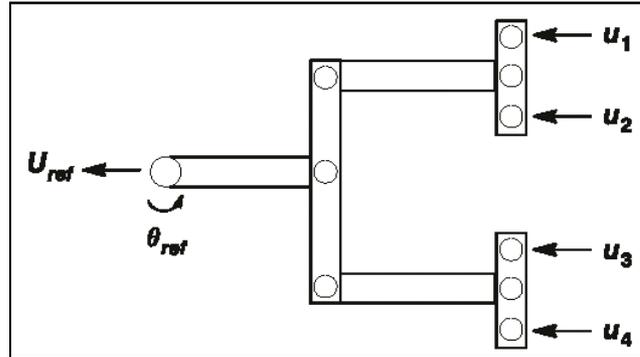
- **Basic equations**

$$U_{ref} = \sum WT_i U_i$$

$$\theta_{ref} = \sum WT_i (r_i \overset{>}{\times} U_i{}^{>})$$

$$+ \sum WT_i \theta_i$$

- **Select one to six dependents from any DOF.**

  Number of dependents = number of U_{ref} .

RBE3 – BULK DATA ENTRY

- **Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.**

1	2	3	4	5	6	7	8	9	10
RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
RBE3	14		100	1234	1.0	123	1	3	
	G1,3	WT2	C2	G2,1	G2,2	-etc.-	WT3	C3	
	5	4.7	1	2	4	6	5.2	2	
	G3,1	G3,2	-etc.-	WT3	C4	G4,1	G4,2	-etc.-	
	7	8	9	5.1	1	15	16		
	"UM"	GM1	CM1	GM2	CM2	GM3	CM3		
	UM	100	14	5	3	7	2		
		GM4	CM4	GM5	CM5	-etc.-			

Note: RBE3 degenerates into RBAR if there is only 1 Gij entry.

MSC Software

RBE3 – BULK DATA ENTRY (Cont.)

Field	Contents
EID	Element identification number. Unique with respect to other rigid elements. (Integer > 0).
REFGRID	Reference grid point identification number. (Integer > 0).
REFC	Component numbers at the reference grid point. (Any of the Integers 1 through 6 with no embedded blanks).
Wti	Weighted factor for components of motion on the following entry at grid points Gi, j. (Real).
Ci	Component numbers with weighted factor WTi at grid point Gi, j. (Any of the Integers 1 through 6 with no embedded blanks).
Gi, j	Grid points whose components Ci have weighted factor WTi in the averaging equations. (Integer > 0).
"UM"	Indicates the start of the degrees of freedom belonging to the m-set. The default action is to assign only the components in REFC to the m-set. (Character).
Gmi	Identification numbers of grid points with degrees of freedom in the m-set. (Integer > 0).
Cmi	Component numbers of GMi to be assigned to the m-set. (Any of the integers 1 through 6 with no embedded blanks).

RBE3 – BULK DATA ENTRY (Cont.)

Remarks:

1. It is recommended that for most applications only the translation components 123 be used for Ci. An exception is the case where the Gi,j are collinear. A rotation component may then be added to one grid point to stabilize its associated rigid body mode for the element.

2. Blank spaces may be left at the end of a Gi, j sequences.

3. For the Lagrange method, the default for "UM" must be used. For the linear method, the default

4. for "UM" should be used except in cases where the user wishes to include some or all REFC components in displacement sets exclusive from the m-set. If the default is not used for "UM":

   a. The total number of components in the m-set (i.e., the total number of dependent degrees-of-freedom defined by the element) must be equal to the number of components in REFC (four components in the example).

   b. The components specified after "UM" must be a subset of the component specified under REFC and (Gi, j, Ci).

   c. The coefficient matrix $[R_m]$ described in the *MSC NASTRAN Reference Manual*, Section must be nonsingular. PARAM, CHECKOUT in SOLs 101 – 200 may be used to check for this condition.

4. Dependent degrees of freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.

RBE3 – BULK DATA ENTRY (Cont.)

Remarks:

5. Rigid elements, unlike MPCs, are not selected through the Case Control section.

6. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.

7. Rigid elements are ignored in heat transfer problems.

8. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See the chapter on *Degree-of-Freedom Sets in the Quick Reference Guide,* for a list of these entries.

9. The formulation for the RBE3 element was changed in Version 70.7. This change allowed the element to give consistent answers that are not dependent upon the units of the model. Only models that connected rotation degrees-of-freedom for Ci were affected. Note that these models are ignoring the recommendation in Remark 5. The formulation prior to Version 70.7 may be obtained by setting SYSTEM(310)=1.

RBE3 DESCRIPTION

- **By default, the reference grid DOF will be the dependent DOF.**

- **Number of dependent DOF is equal to the number of DOF on the REFC field.**

- **Dependent DOF cannot be SPC'd, OMITted, SUPORTed or be dependent on other RBE/MPC elements.**

RBE3 IS NOT RIGID!

- ## RBE3 vs. RBE2

    - **RBE3 allows warping and 3D effects**

    - **In this example, RBE2 enforces beam theory (plane sections remain planar)**

RBE3

RBE2

MSC Software
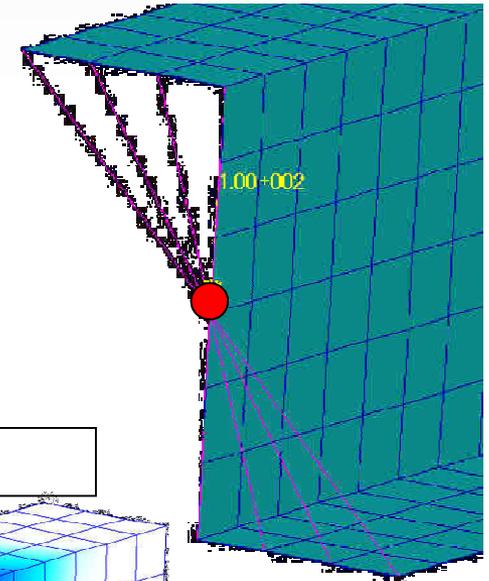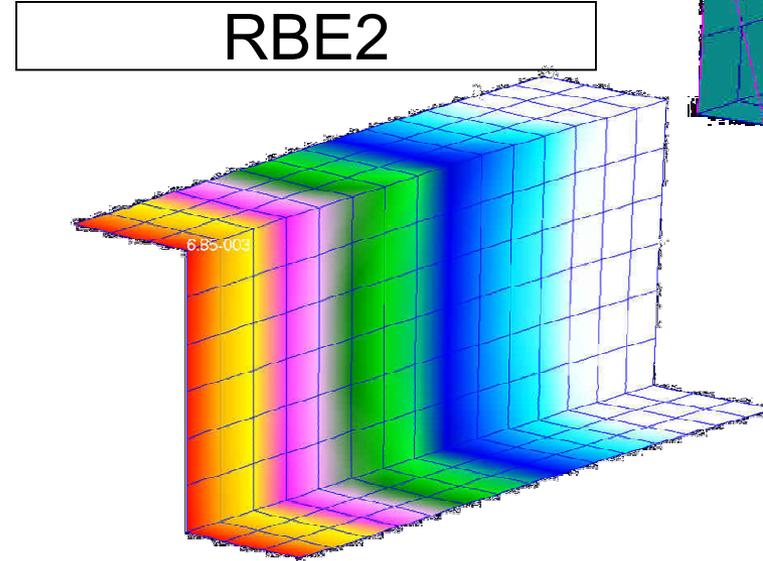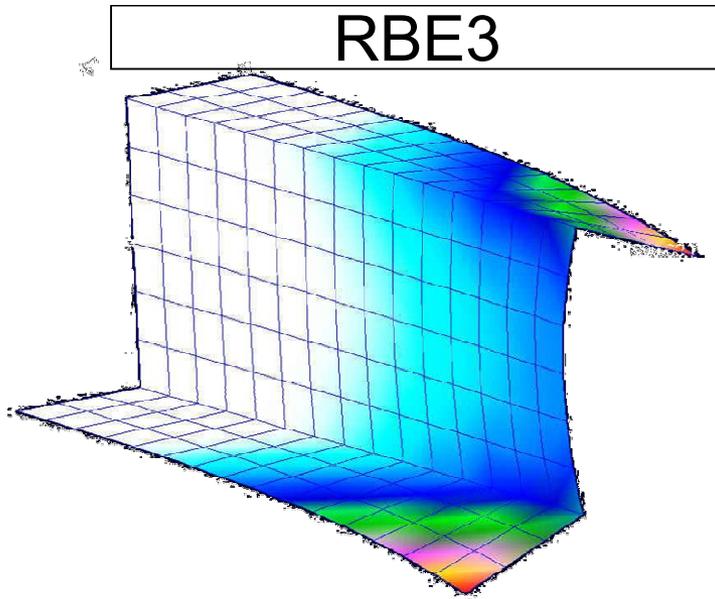
RBE3: HOW IT WORKS?

- **Forces/Moments applied at reference grid are distributed to the master grids in same manner as classical bolt pattern analysis.**
  - Step 1: Applied loads are transferred to the CG of the weighted grid group using an equivalent Force/Moment
  - Step 2: Applied loads at CG transferred to master grids according to each grid's weighting factor

  Note: If independent DOFs contain rotations, RBE3 does not work like classical bolt pattern analysis.

MSC Software

RBE3 HOW IT WORKS? (Cont.)

- **If independent DOFs include rotations, moments at CG are mapped as equivalent force couples, and concentrated moments.**

- **Step 1: Transform force/moment at reference grid to equivalent force/moment at weighted CG of master grids.**

$$F_{CG}=F_A$$

$$M_{CG}=M_A+F_A*e$$

MSC Software
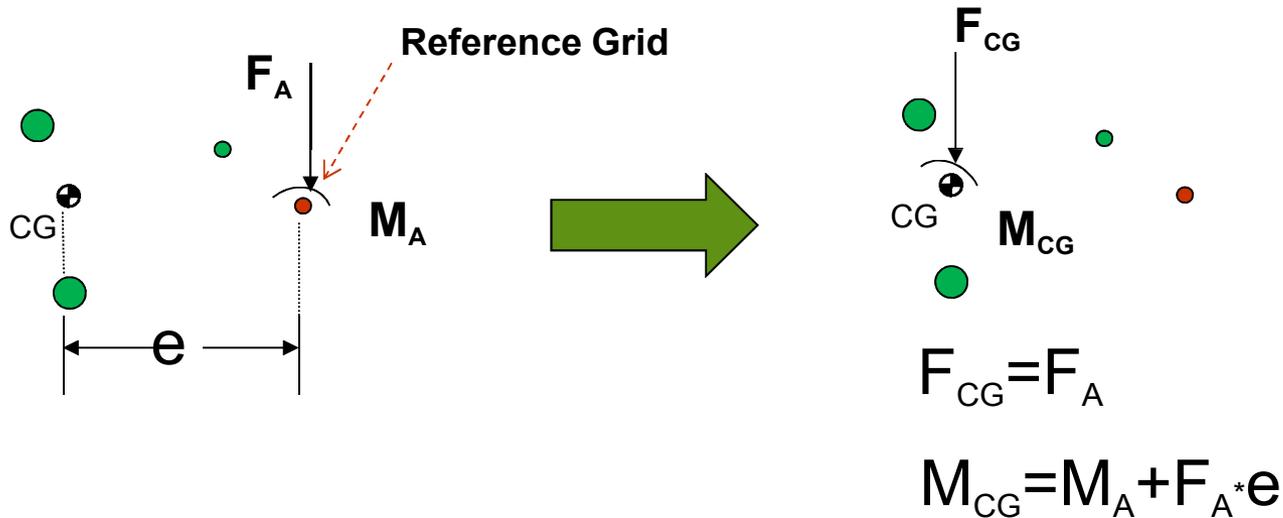
RBE3: HOW IT WORKS? (Cont.)

- **Step 2: Move loads at CG to master grids according to their weighting values.**
  - Force at CG divided amongst master grids according to weighting factors W_i
  - Moment at CG mapped as equivalent force couples on master grids according to weighting factors W_i

RBE3: HOW IT WORKS? (Cont.)

- **Step 2: Continued…**
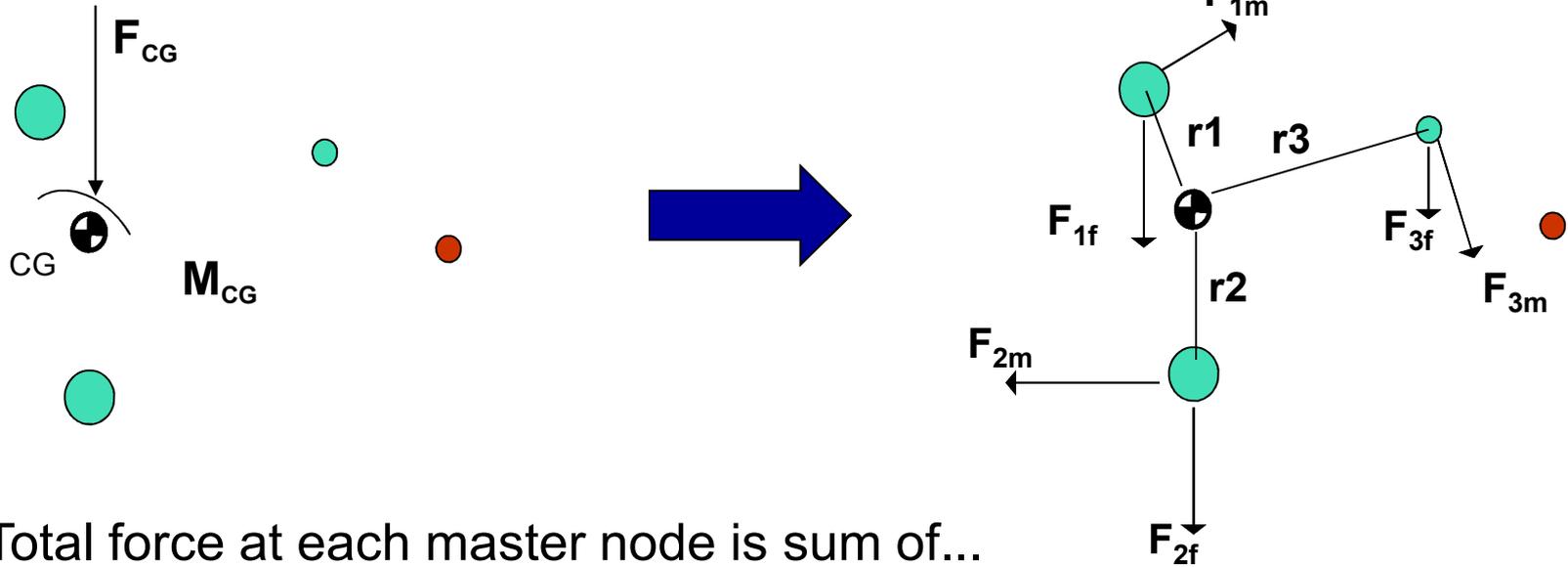
Total force at each master node is sum of...

Forces derived from force at CG: $F_{if} = F_{CG}\{W_i/\Sigma W_i\}$

Plus Forces derived from moment at CG:

$F_{im} = \{M_{cg}W_ir_i/(W_1r_1^2+W_2r_2^2+W_3r_3^2)\}$

MSC Software

RBE3: HOW IT WORKS? (Cont.)

- **Masses on reference grid are smeared to the master grids similar to how forces are distributed**
  - Mass is distributed to the master grids according to their weighting factors
  - Motion of reference mass results in inertial force that gets transferred to master grids
  - Reference node inertial force is distributed in same manner as when static force is applied to the reference grid.

MSC Software

EXAMPLE 1: FORCE THROUGH CG

- **RBE3 distribution of loads when force at reference grid at CG passes through CG of master grids**
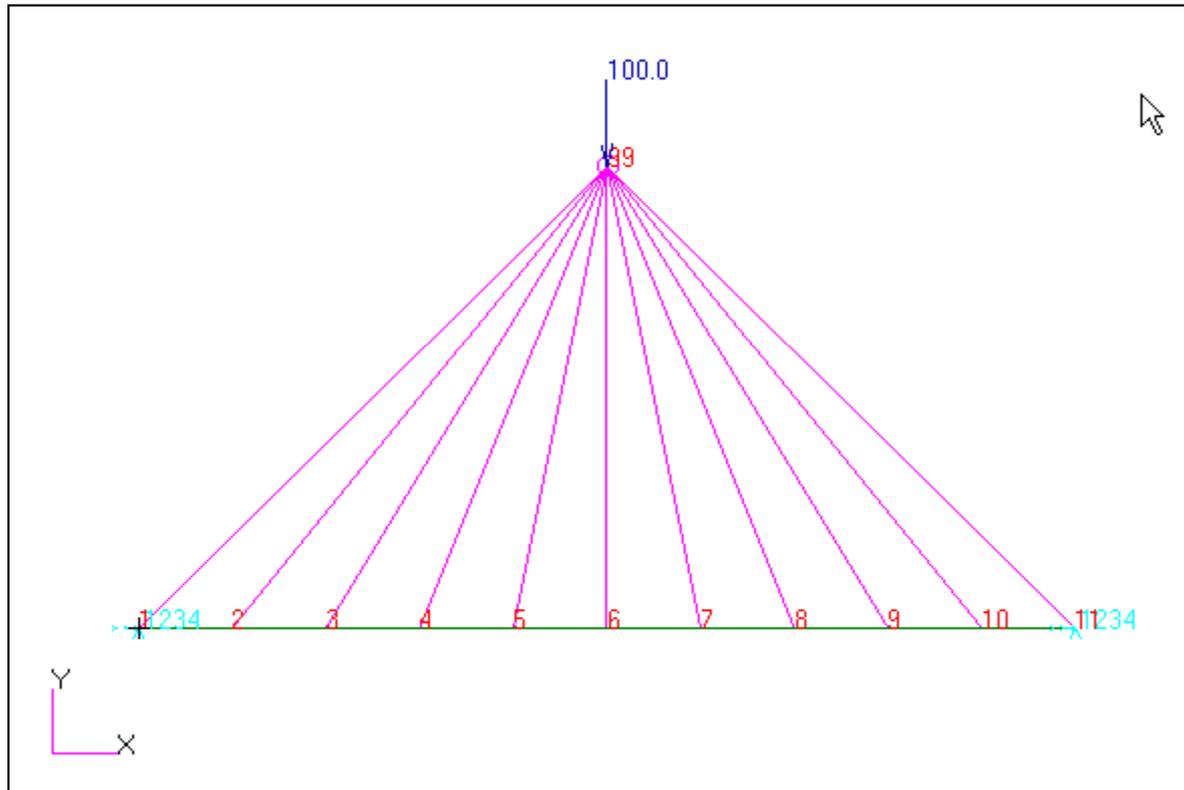
EXAMPLE 1: FORCE THROUGH CG (Cont.)

- **Simply supported beam**
  - 10 elements, 11 nodes numbered 1 through 11

- **100 lb Force in negative Y on reference grid 99**

1	2	3	4	5	6	7	8	9	10
RBE3	11	4	99	123456	1.	1234	1	2	
	3	4	5	6	7	8	9	10	
	11								

MSC Software

EXAMPLE 1: FORCE THROUGH CG (Cont.)

- **Load through CG with uniform weighting factors results in uniform load distribution**
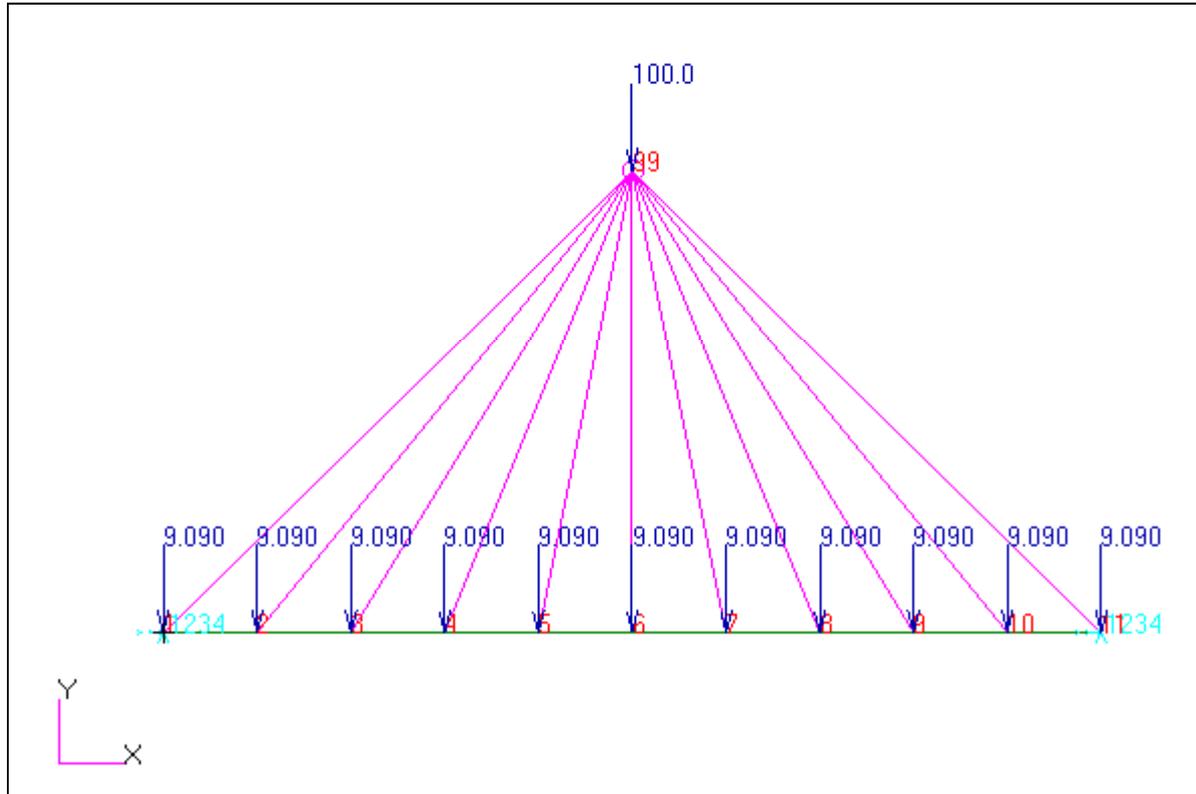
EXAMPLE 1: FORCE THROUGH CG (Cont.)

- **Comments…**
  - Since master grids are co-linear, the x rotation DOF is added so that master grids can determine all 6 rigid body motions, otherwise RBE3 would be singular

```
RBE3,  11,    ,99, 123456,  1.,  123,  1, 2
      ,   3,  4,  5,  6, 7,  8,  9, 10
      ,  11


*** USER FATAL MESSAGE 2038 (RBE3D)
    USER ACTION:  ADD MORE DOFS TO THE CONNECTED POINTS TO
                  INSURE THAT THEY CAN CONSTRAIN ALL 6 RIGID
                  BODY MODES OF THE ELEMENT.
```

  - Corrected RBE3 data (add DOF4 to one or more Master Grids):

```
RBE3,  11,    ,99, 123456,  1.,  1234,  1, 2
      ,   3,  4,  5,  6, 7,  8,  9, 10
      ,  11
~
```

EXAMPLE 2: LOAD NOT THROUGH CG

- **How does the RBE3 distribute loads when force on reference grid does not pass through CG of master grids?**
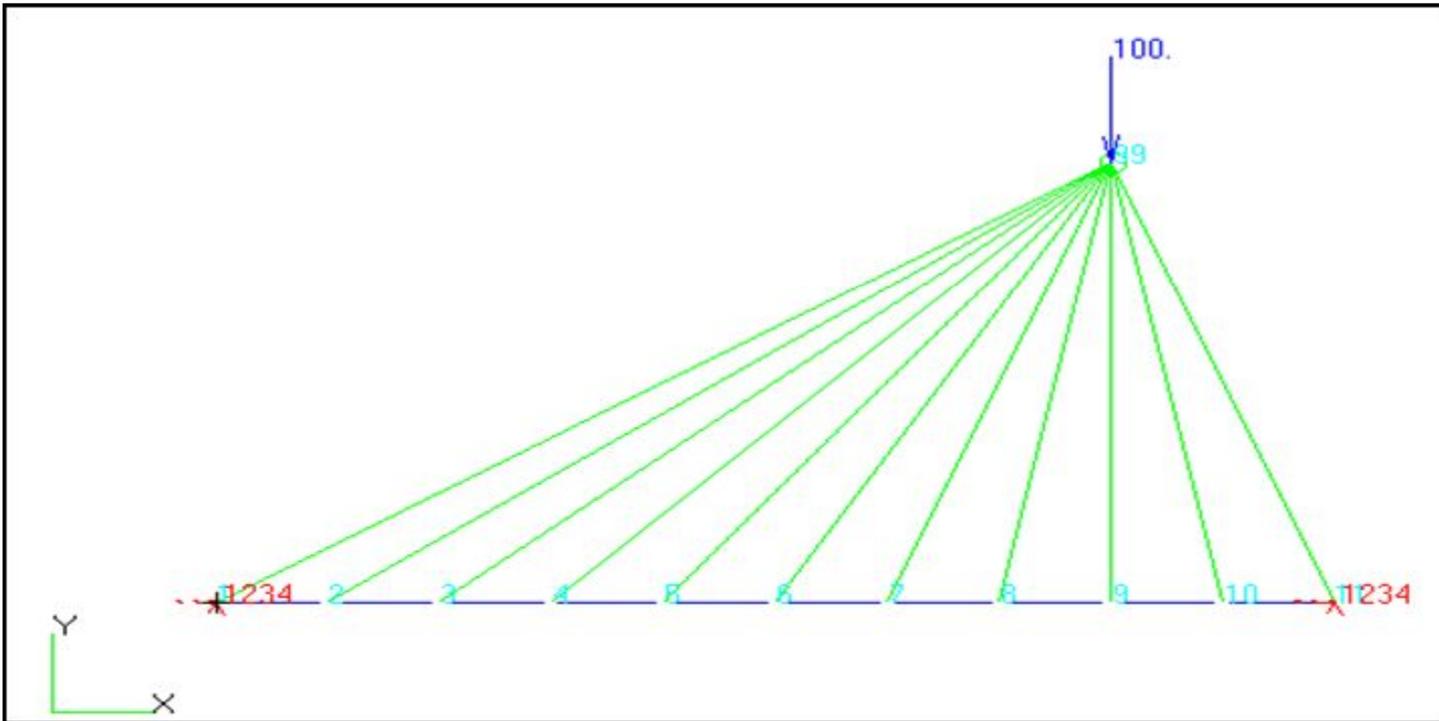
EXAMPLE 2: LOAD NOT THROUGH CG (Cont.)

- **The resulting force distribution is not intuitively obvious**
  - Note forces in the opposite direction on the left side of the beam.
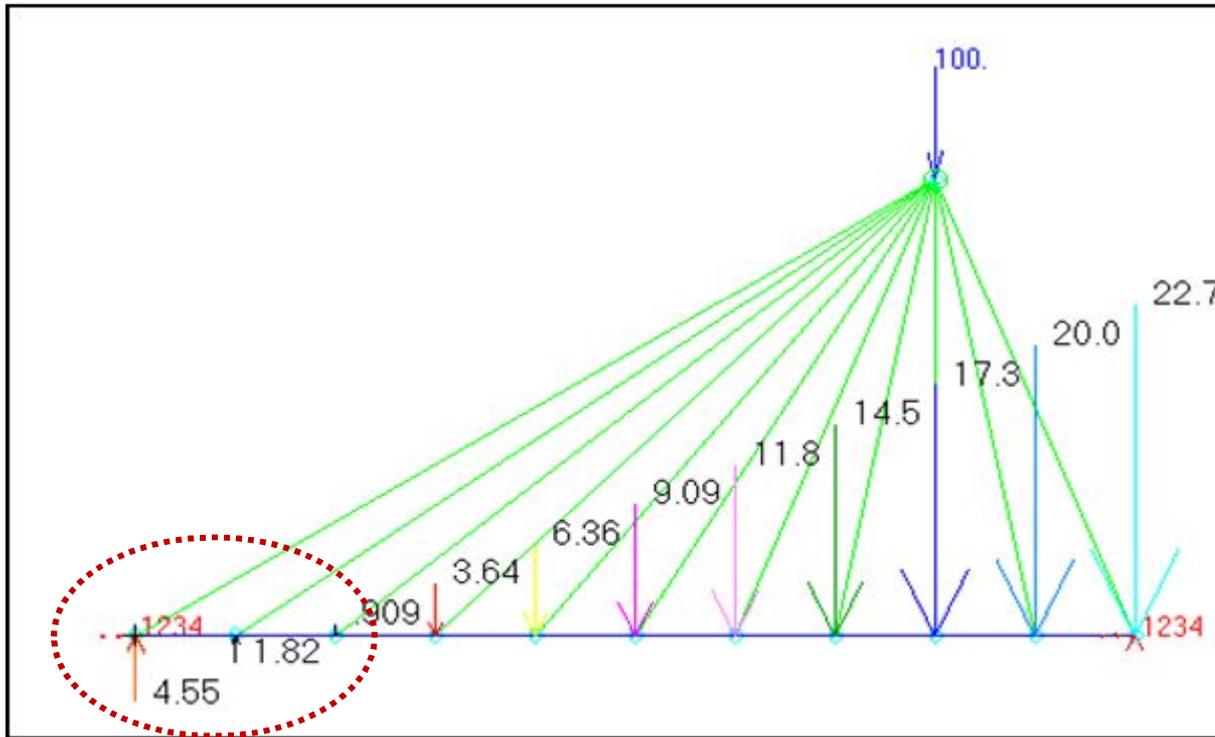
Upward loads on left side of beam result from moment caused by movement of applied load to the CG of master grids.

MSC Software®

EXAMPLE 3: TRANSVERSE LOAD ON BEAM

- **Use of weighting factors to generate realistic load distribution: 100 LB. transverse load on 3D beam.**
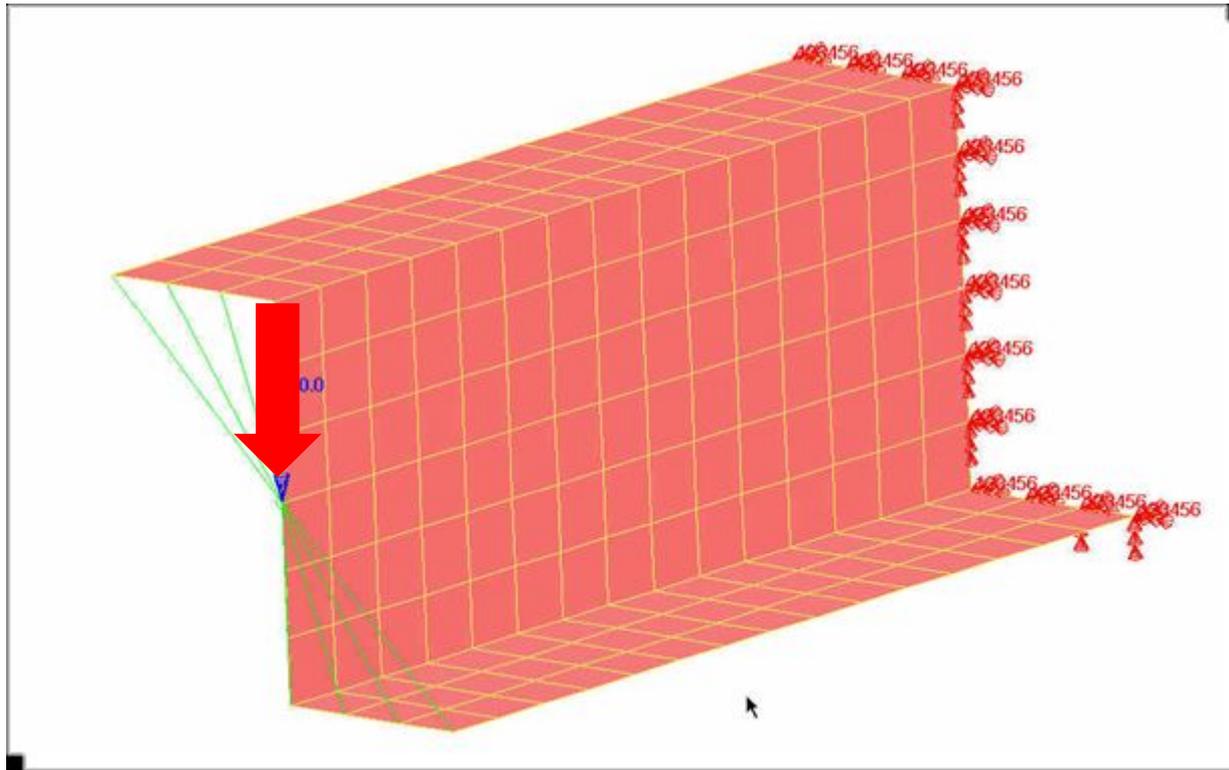
- **If uniform weighting factors are used, the load is equally distributed to all grids.**
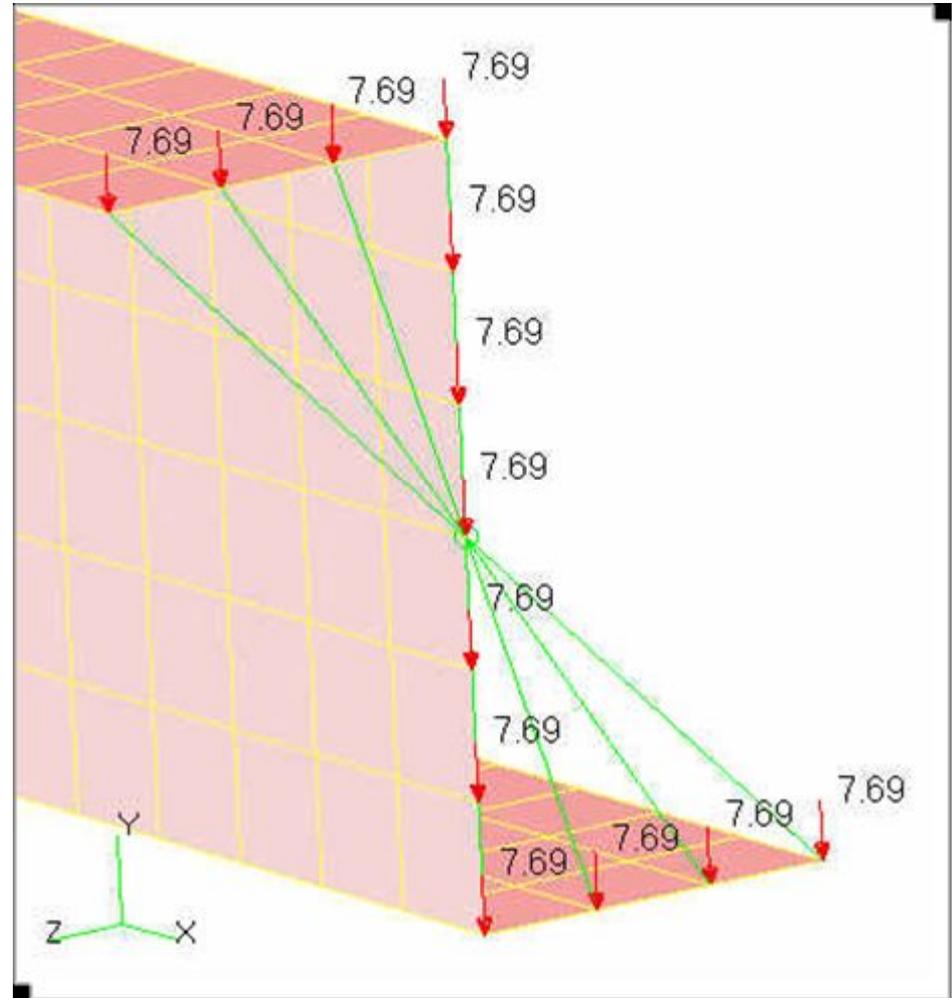
EXAMPLE 3: TRANSVERSE LOAD ON BEAM (Cont.)

- **The uniform load distribution results in too much transverse load in flanges causing them to droop.**

Displacement Contour

MSC Software
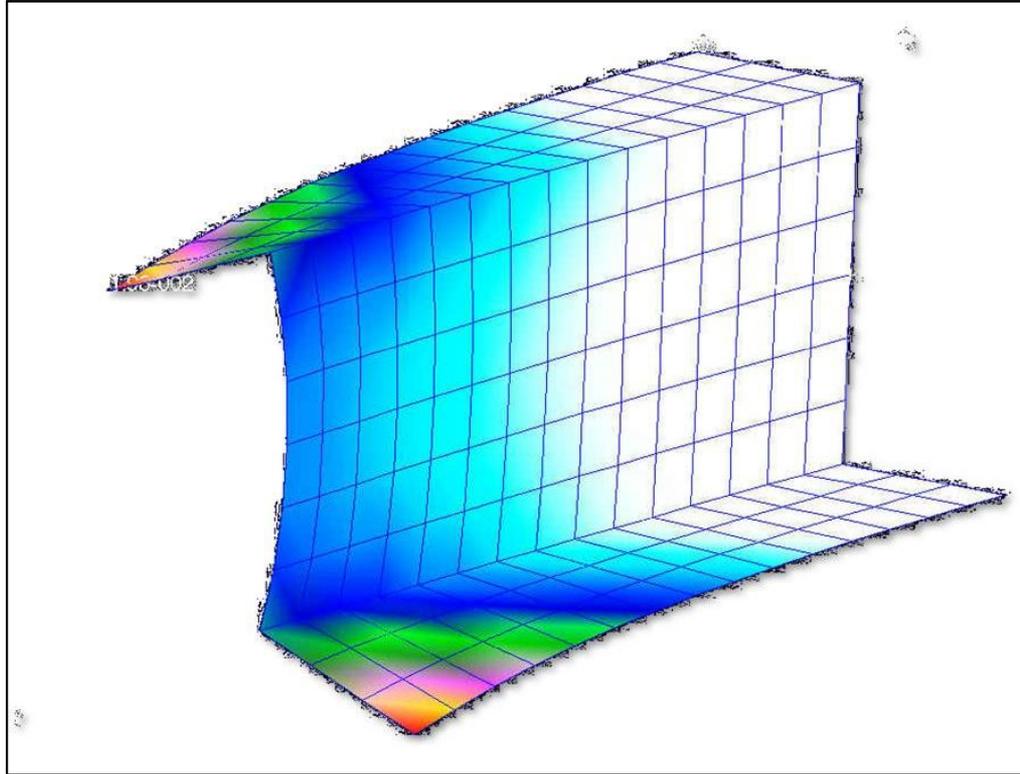
EXAMPLE 3: TRANSVERSE LOAD ON BEAM (Cont.)

- **Assume quadratic distribution of load in web**

- **Assume thin flanges carry zero transverse load**

- **Master DOF 1235. DOF 5 added to make RY rigid body motion determinate**
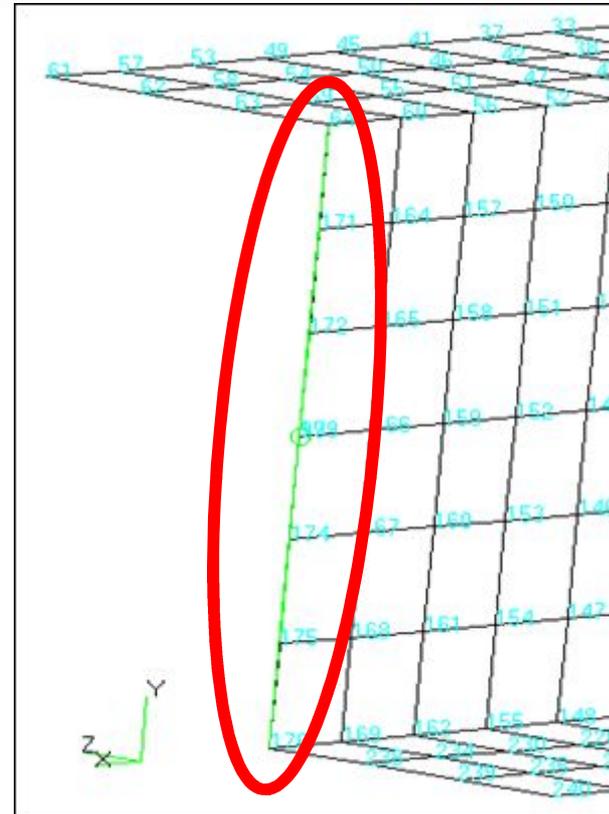
RBE3	181		999	123456	.153	1235	64	.556
	1235	171	.889	1235	172	1.0	1235	173
	.889	1235	174	.556	1235	175	.153	1235
	176							

EXAMPLE 3: TRANSVERSE LOAD ON BEAM (Cont.)

- **Displacements with quadratic weighting factors virtually equivalent to those from RBE2 (Beam Theory), but do not impose "plane sections remain planar" as does RBE2.**

EXAMPLE 3: TRANSVERSE LOAD ON BEAM (Cont.)

- **RBE3 Displacement Contour**
  - Max Y disp=.00685

MSC Software
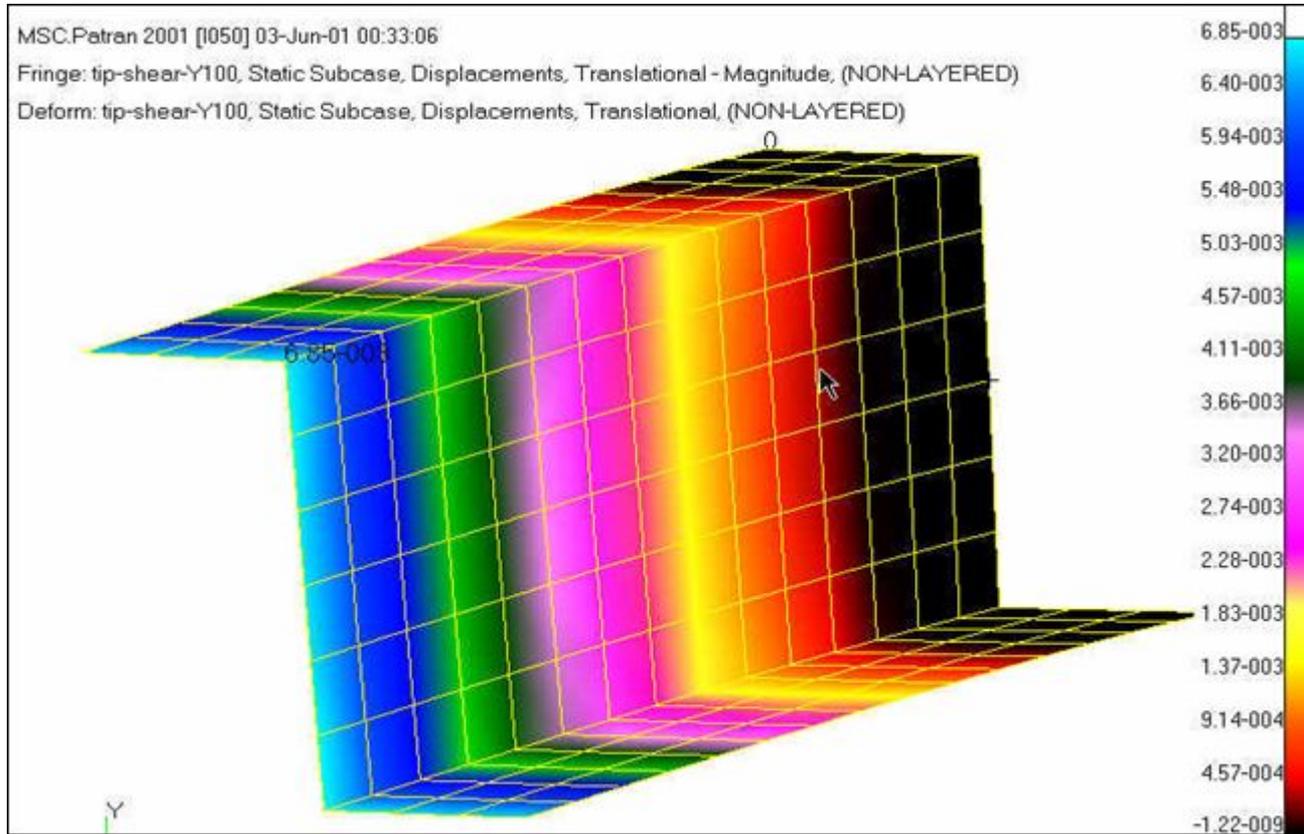
EXAMPLE 3: TRANSVERSE LOAD ON BEAM (Cont.)

- **RBE2 Displacement contour**
  - Max Y disp=.00685

MSC Software
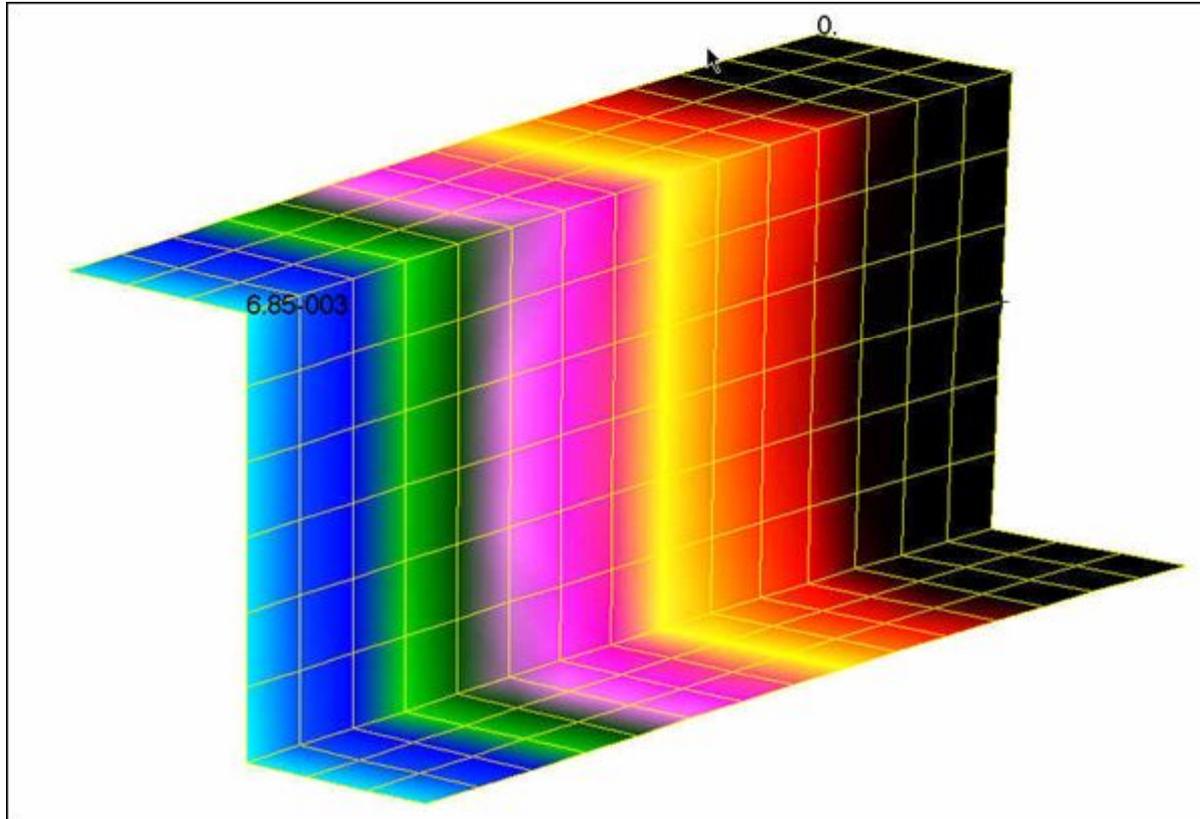
EXAMPLE 4: USE RBE3 FOR UNCONSTRAINED MOTION

- **Use RBE3 to get "unconstrained" motion**

- **Cylinder under pressure**

- **Which Grid(s) do you pick to constrain out Rigid body motion, but still allow for free expansion due to pressure?**

MSC Software
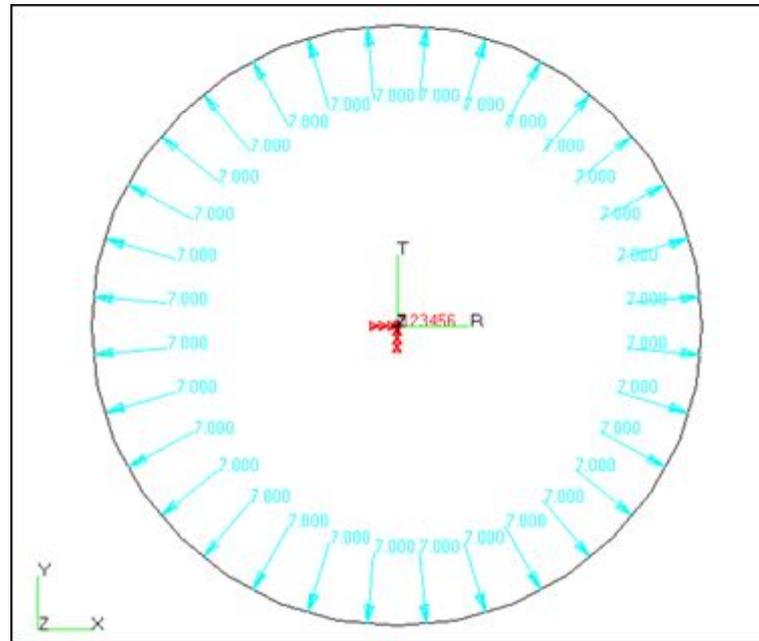
EXAMPLE 4: USE RBE3 FOR UNCONSTRAINED MOTION (Cont.)

- **Solution:**
  - Use RBE3
  - Move dependent DOF from reference grid to selected master grids with UM option on RBE3 (otherwise, reference grid cannot be SPC'd)
  - Apply SPC to reference grid

MSC Software®
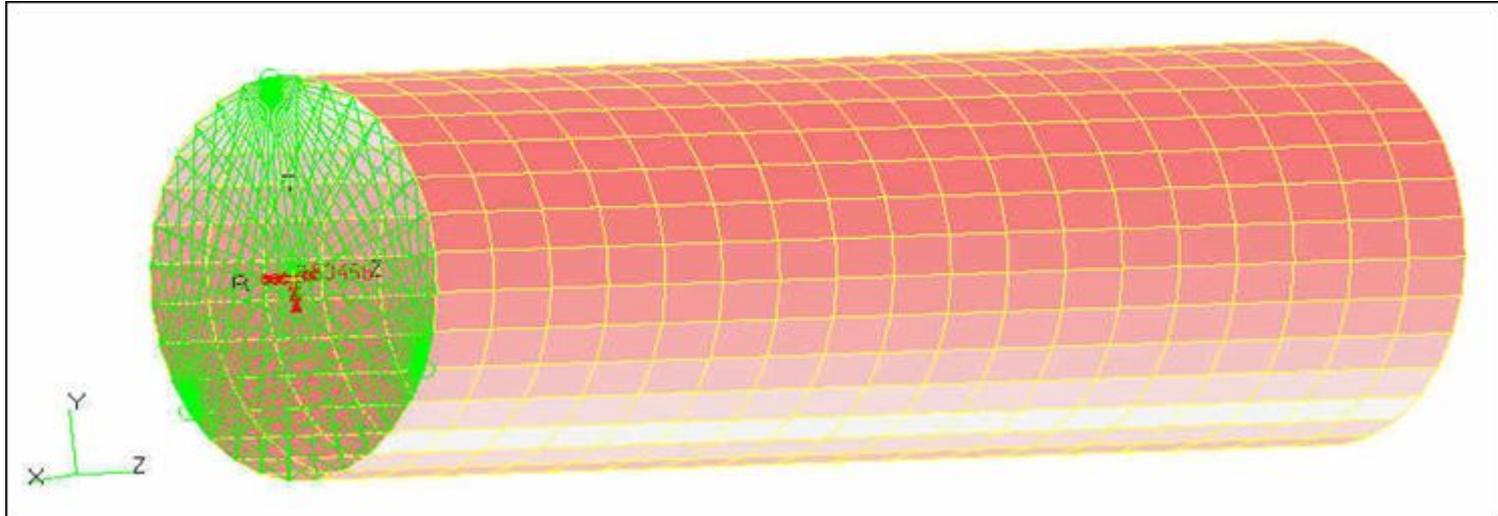
- **Since reference grid has 6 DOF, we must assign 6 "UM" DOF to a set of master grids**
  - Pick 3 points, forming a nice triangle for best numerical conditioning
  - Select a total of 6 DOF over the three UM grids to determine the 6 rigid body motions of the RBE3
  - Note: "M" is the MSC NASTRAN DOF set name for dependent DOF
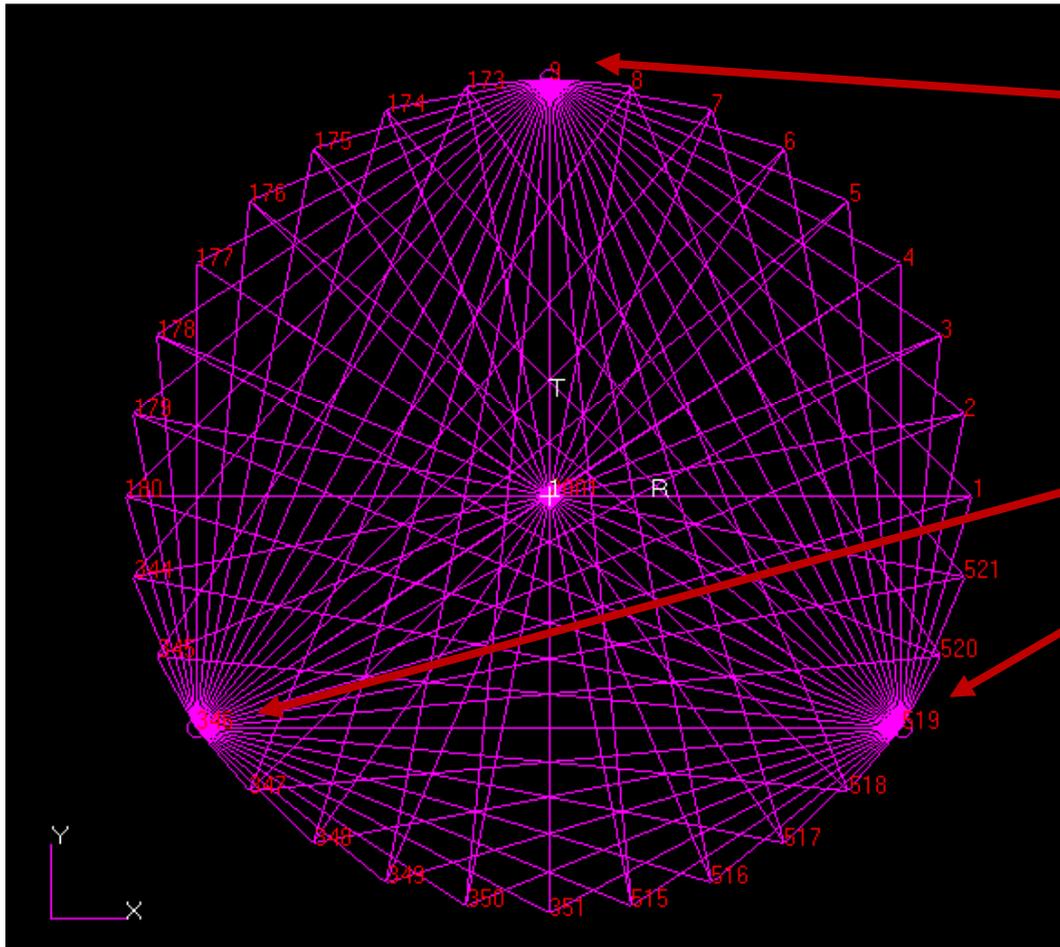
"UM" Grids

- **For circular geometry, it's convenient to use a cylindrical coordinate system for the master grids.**
  - Put THETA and Z DOF in UM set for each of the three UM grids to determine RBE3 rigid body motion

RBE3	1009		2001	123456	1.	123	1	2
	3	4	5	6	7	8	9	173
	174	175	176	177	178	179	180	344
	345	346	347	348	349	350	351	515
	516	517	518	519	520	521		
UM		346	23	519	23	9	23	

- **Result is free expansion due to internal pressure. (note: Poisson effect causes shortening**
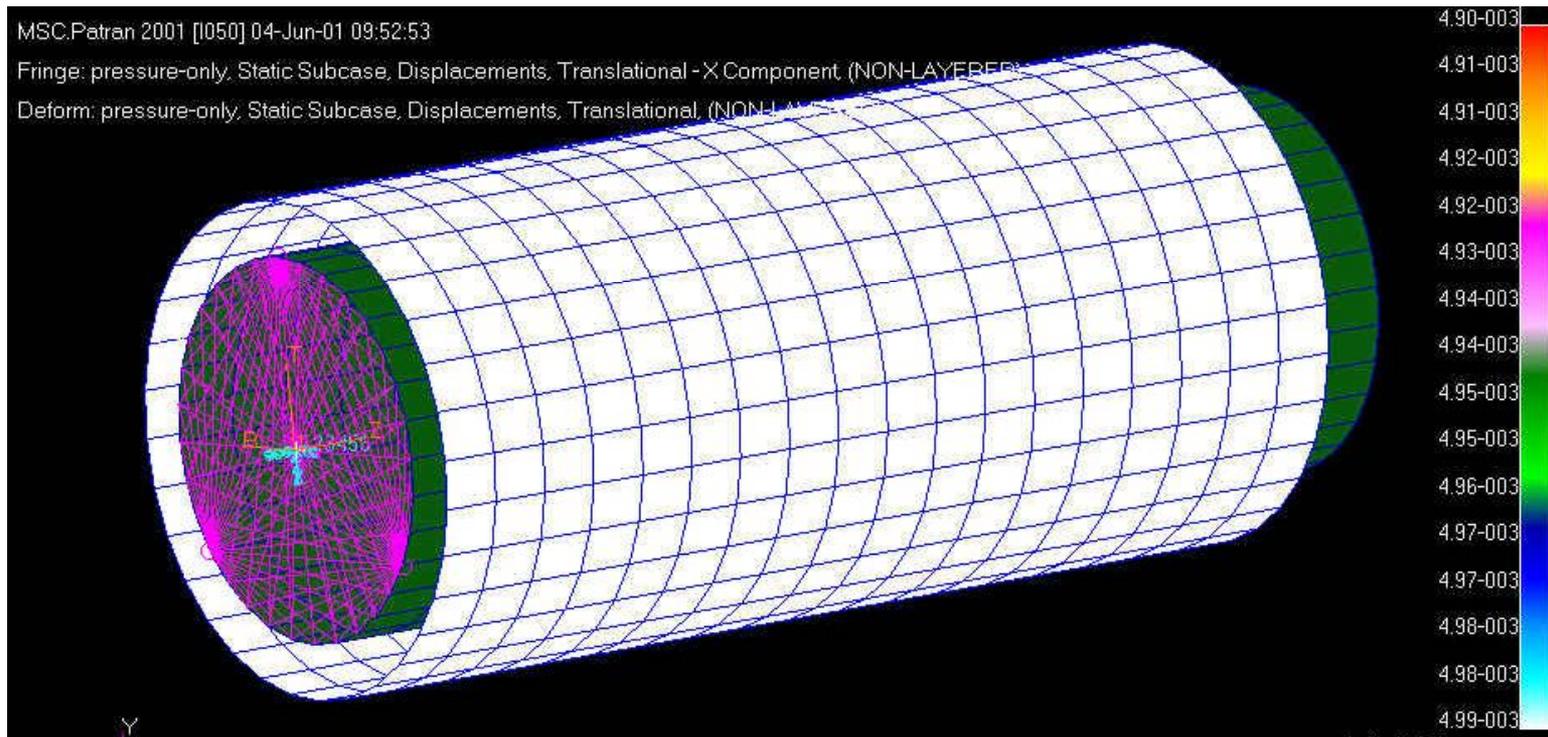
EXAMPLE 4: USE RBE3 FOR UNCONSTRAINED MOTION (Cont.)

- **Resulting MPC Forces are numeric zeroes verifying that no stiffness has been added.**
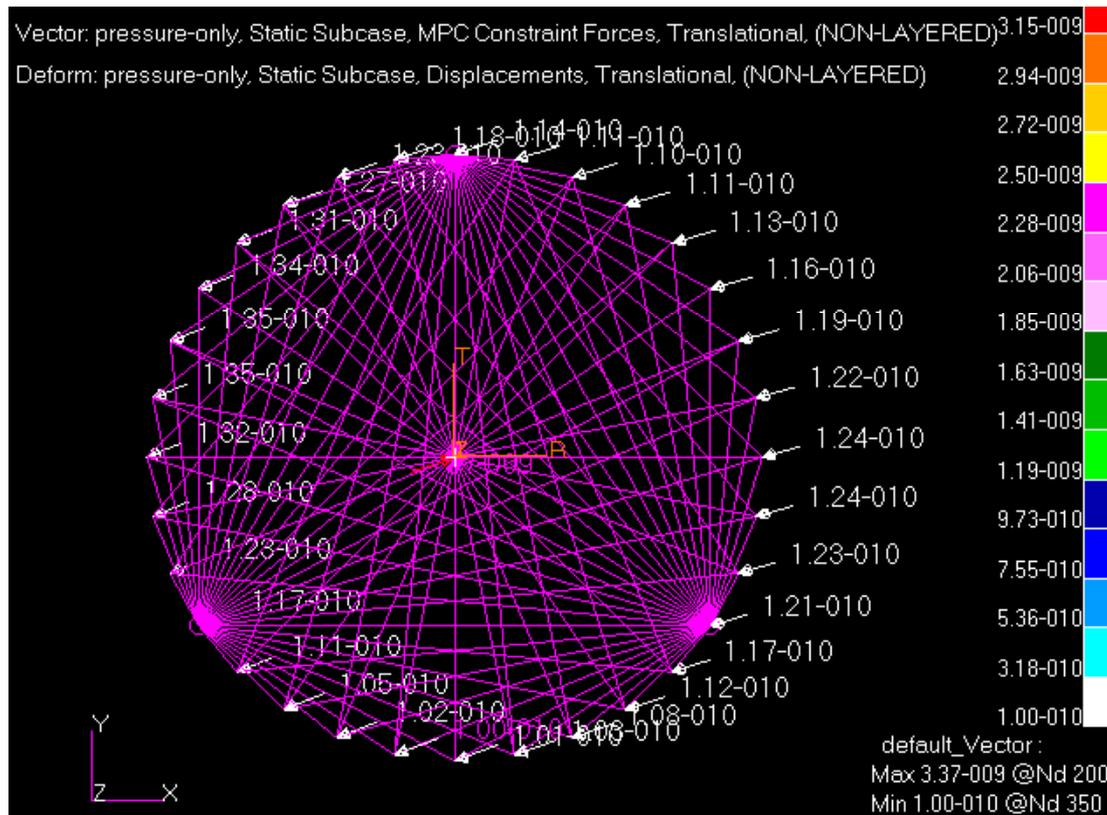
RSPLINE – THE LINEAR SPLINE

- **The RSPLINE is an "interpolation" element using the equations of a beam – Example**

MSC Software
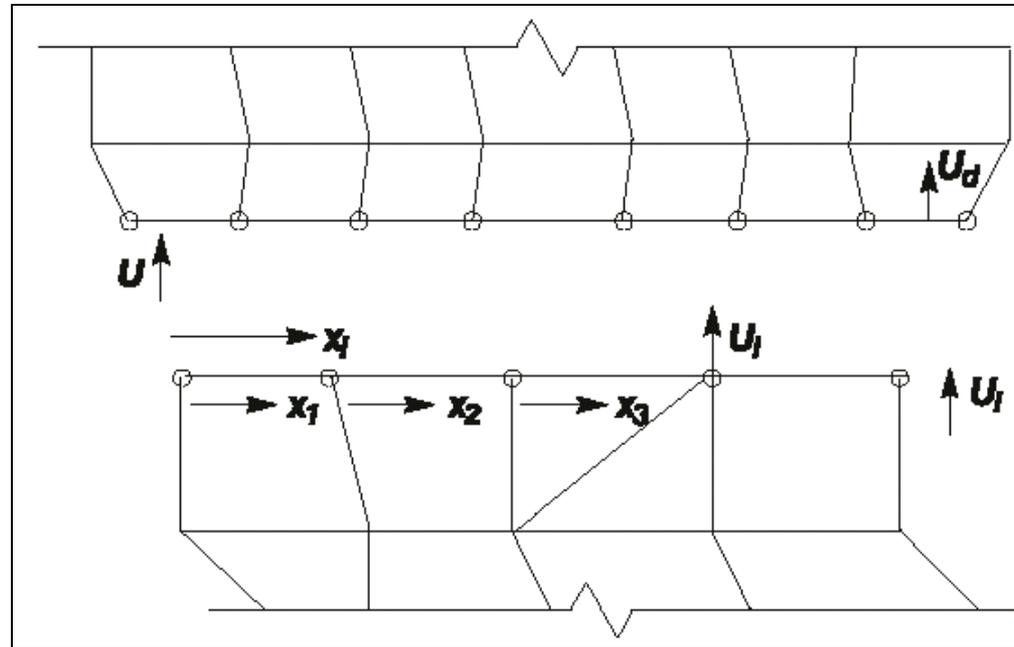
RSPLINE – THE LINEAR SPLINE (Cont.)

- **Basic equations**

$$U^i(x) = C_0^i + C_1^i x_i + C_2^i x_i^2 + C_3^i x_i^3$$

$$U_i, \frac{\partial U_i}{\partial x}, \frac{\partial^2 U_i}{\partial x^2}$$

match at grid points

- **In matrix notation**

$$\{C\} = [H_i]^{-1}\{U_i\}$$

$$\{U_d\} = [H_d]\{C\}$$

MSC Software®

RSPLINE – BULK DATA ENTRY

- **Defines multipoint constraints for the interpolation of displacements at grid points.**

1	2	3	4	5	6	7	8	9	10
RSPLINE	EID	D/L	G1	G2	C2	G3	C3	G4	
RSPLINE	73	.05	27	28	123456	29		30	
	C4	G5	C5	G6	-etc.-				
	123	75	123	71					

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000).
D/L	Ratio of the diameter of the elastic tube to the sum of the lengths of all segments. (Real > 0.0; Default = 0.1).
Gi	Grid point identification number. (Integer >0).
Ci	Components to be constrained. (Blank or any combination of the integers 1 through 6).

MSC Software

RSPLINE – BULK DATA ENTRY (Cont.)

Remarks:

1. Displacements are interpolated from the equations of an elastic beam passing through the grid points. This is a linear method only element, and not controlled with the Case Control command RIGID.

2. A blank field for C_i indicates that all six degrees-of-freedom at G_i are independent. Since G1 must be independent, no field is provided for C1. Since the last grid point must also be independent, the last field must be a G_i, not a C_i. For the example shown G1, G3, and G6 are independent. G2 has six constrained degrees-of-freedom while G4 and G5 each have three.

3. Dependent (i.e., constrained) degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.

4. Degrees-of-freedom declared to be independent by one rigid body element can be made dependent by another rigid body element or by a multipoint constraint.

5. EIDs must be unique.

6. Rigid elements (including RSPLINE), unlike MPCs, are not selected through the Case Control Section.

MSC Software®

RSPLINE – BULK DATA ENTRY (Cont.)

Remarks: (Cont.)

7. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.

8. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry in the Reference Guide.

9. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See Degree-of-Freedom Sets in the Reference Guide.

10. The constraint coefficient matrix is affected by the order of the Gi Ci pairs on the RSPLINE entry. The order of the pairs should be specified in the same order that they appear along the line that joins the two regions. If this order is not followed then the RSPLINE will have folds in it that may yield some unexpected interpolation results

11. The independent degrees-of-freedom that are the rotation components most nearly parallel to the line joining the regions should not normally be constrained.

MSC Software

RSSCON – BULK DATA ENTRY

- **Defines multipoint constraints to model clamped connections of shell-to-solid elements.**

1	2	3	4	5	6	7	8	9	10
RSSCON	RBID	TYPE	ES1	EA1	EB1	ES2	EA2	EB2	
RSSCON	110	GRID	11	12	13	14	15	16	
RSSCON	111	GRID	31	74	75				
RSSCON	115	ELEM	311	741					

Field	Contents
RBID	Elements identification number. (Integer = 0)
TYPE	Type of connectivity TYPE = "ELAM" connection is described with element identification numbers. TYPE = "GRID" connection is described with grid point identification numbers. (Character: "GRID" or "ELAM"; Default = "ELAM")

MSC Software®

RSSCON – BULK DATA ENTRY (Cont.)

Field	Contents
ES1	Shell element identification number if TYPE = "ELAM". Shell grid point identification number if TYPE = "GRID". See Figure 1. (Integer > 0)
EA1	Solid element identification number if TYPE = "ELEM". Solid grid point identification number if TYPE = "GRID". (Integer > 0)
EB1	Solid grid-point identification number for TYPE = "GRID" only. (Integer > 0 or blank)
ES2	Shell grid-point identification number for TYPE = "GRID" only. (Integer > 0 or blank)
EA2	Solid grid-point identification number for TYPE = "GRID" only. (Integer > 0 or blank)
EB2	Solid grid-point identification number for TYPE = "GRID" only. (Integer > 0 or blank)

MSC Software

RSSCON – BULK DATA ENTRY (Cont.)

Remarks:

1. RSSCON generates a multipoint constraint that models a clamped connection between a shell and a solid element. The shell degrees of freedom are put in the dependent set (m-set). The translational degrees of freedom of the shell edge are connected to the translational degrees of freedom of the upper and lower solid edge. The rotational degrees of freedom of the shell are connected to the translational degrees of freedom of the lower and upper edges of the solid element face. Poisson's ratio effect are considered in the translational degrees of freedom.

2. The shell grid point must lie on the line connecting the two solid grid points. It can have an offset from this line, which can be not be more than 5% of the distance between the two solid grid points. The shell grid points that are out of the tolerance will not be constrained, and a fatal message will be issued. This tolerance is adjustable. Please see PARAM, TOLRSC, and PARAM, SEPIXOVR.

3. When using the TYPE = "ELAM" option.

   a. The elements may be p-elements. The solid elements are CHEXA, CPENTA, and CTETRA with and without midside nodes. The shell elements are CQUAD4, CTRIA3, CQUADR, CTRIAR, CQUAD8, or CTRIA6.

   b. In case of p-elements, the p-value of the shell element edge is adjusted to the higher of the p-value of the upper of lower solid p-element edge. If one of the elements is an h-elements, then the p-value of the adjacent edge is lowered to 1.

MSC Software

RSSCON – BULK DATA ENTRY (Cont.)

Remarks: (Cont.)

    c. Both the shell and solid elements have to belong to the same superelement. This restriction can be bypassed using SEELT entry to reassign the downstream boundary element to an upstream superelement.

    d. When a straight shell p-element edge and a solid p-element are connected, the geometry of the shell edge is not changed to fit the solid face. When a curved shell p-element edge and a solid p-element are connected, the two solid edges and solid face are not changed to match the shell edge.

    e. It is not recommended to connect more than one shell element to the same solid using the ELEM option. If attempted, conflicts in the multipoint constraint relations may leads to UFM 6692.

4. When using TYPE_"GRID" option

    a. The GRID option does not verify that the grids used are valid shell and/or solid grids.

    b. The hierarchical degrees of freedom of p-element edges are not constrained. The GRID option is therefore not recommended for p-elements.

    c. The grids in the GRID option can be different superelements. The shell grid must be in the upstream superelement.

5. It is recommended that the height of the solid element's face is approximately equal to the shell element's thickness of the shell. The shell edge should then be placed in the middle of the solid face.

MSC Software

RSSCON – BULK DATA ENTRY (Cont.)

Remarks: (Cont.)

6. The shell edge may coincide with the upper or lower edge of the solid face.

7. The RSSCON entry, unlike MPCs, cannot be selected through the Case Control Section.

8. Forces of multipoint constraints may be recovered with the MPCFORCE Case Control command.

9. The RSSCON is ignored in heat-transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 9 for that entry.

10. The m-set coordinates (shell degrees-of-freedom) may not be specified on other entries that define mutually exclusive sets. See Degree-of-Freedom Sets, in the reference guide for a list of these entries.
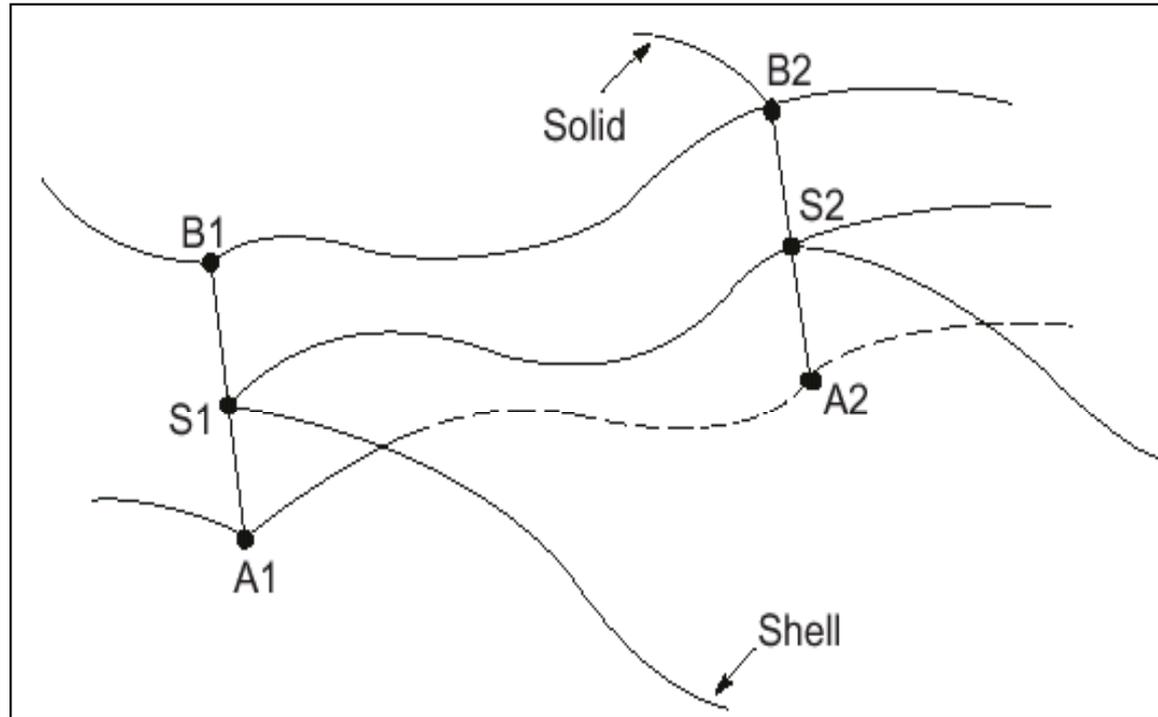
Figure 1.   Shell Elements Connected to the Faces of Solid Elements

- **Perform Workshop 2.**

LINE INTERFACE ELEMENT- CINTC

- **CINTC**
  - Introduced in MSC Nastran 2005R3 (2005.5)
  - Used to connect Dissimilar Meshes
    - 2D Edges
    - 1D Beam
  - Benefits/Uses
    - Connecting independently built meshes
    - Global / Local modeling
    - Design changes to portion of model

Line Interface Element
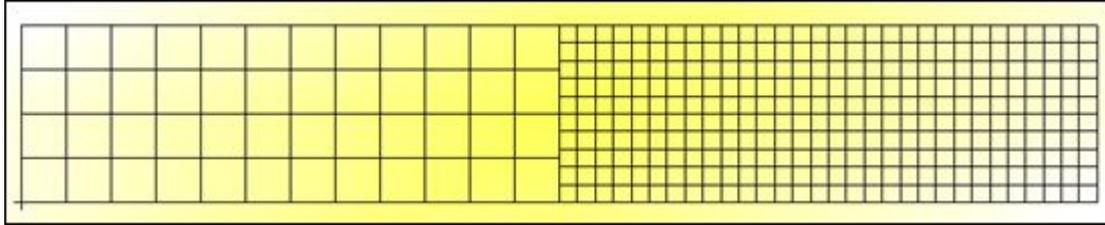
Line interface element is used to connect dissimilar meshes along the edges of finite element mesh subdomains.

A set of MPC (Multipoint Constraint) equations are internally generated with the interface boundary grids.

GRIDI   GRIDF

Open Boundary

GRIDI=GRID

Closed Boundary

MSC Software®
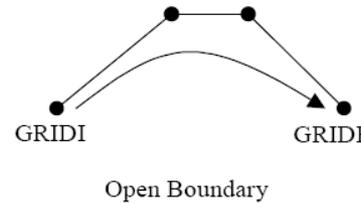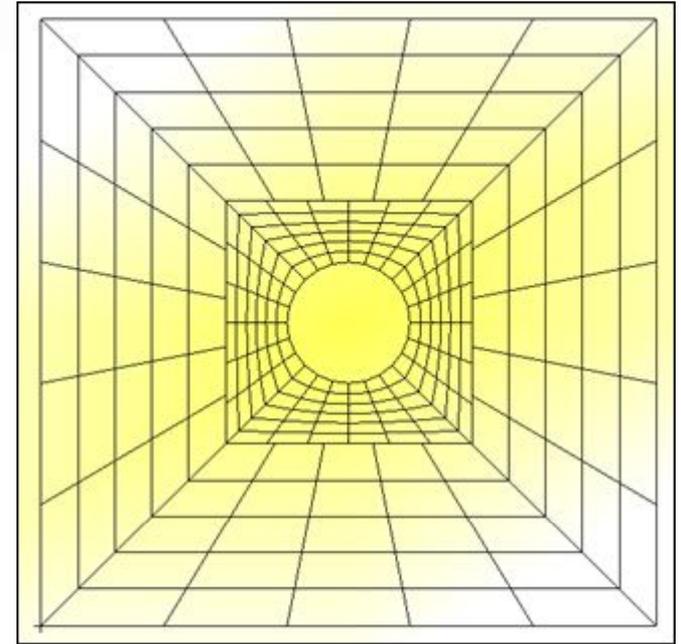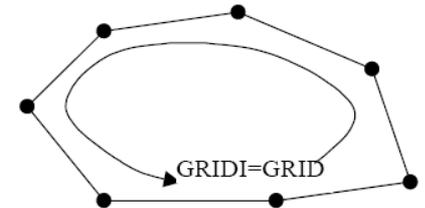
Verification Case - CINTC

- **Open Boundary**
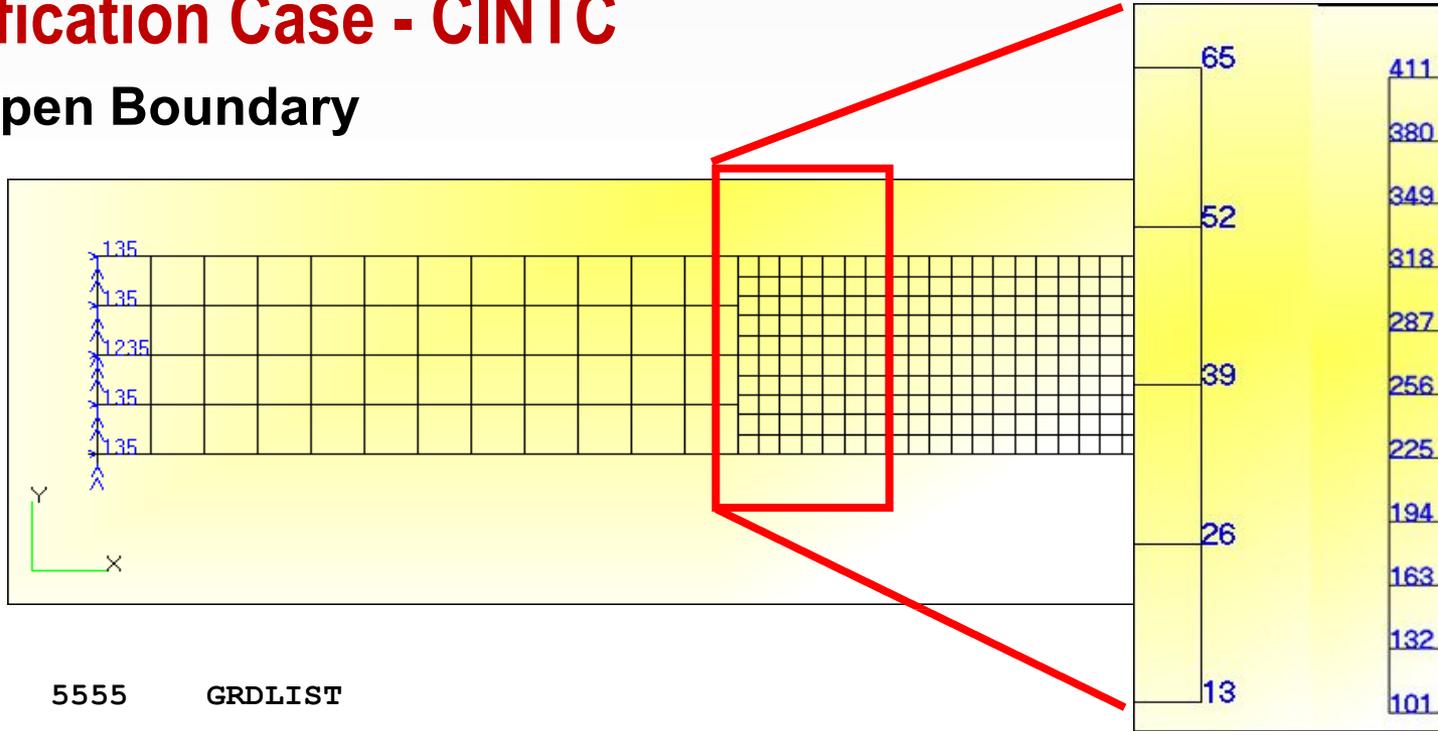

```
CINTC    5555     GRDLIST

         LIST=(9991(L),9992(L))

$GMBDNC BID      GridI     GridF

$       GRID     grid1     grid2     grid3     etc.

GMBNDC  9991     13        65

        GRID     26        39        52

GMBNDC  9992     101       411

        GRID     132       163       194       225       256       287       318

        349      380
```

Coarse Mesh = Independent

(1st BID in LIST)

Verification Case - CINTC

- **Reverse Independent List**
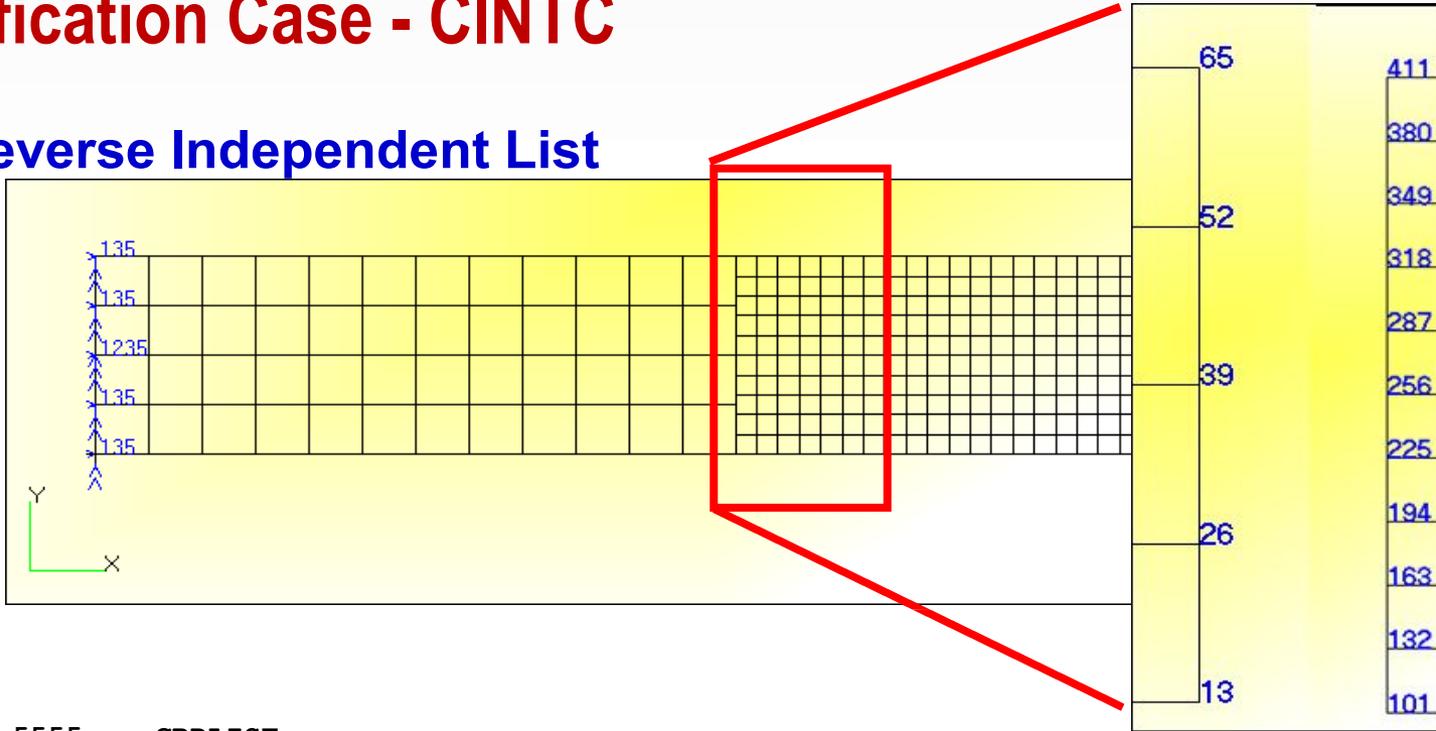

```
CINTC    5555    GRDLIST
         LIST=(9992(L),9991(L))
$GMBDNC BID      GridI    GridF
$        GRID    grid1    grid2    grid3    etc.
GMBNDC  9991    13       65
         GRID    26       39       52
GMBNDC  9992    101      411
         GRID    132      163      194    225    256    287    318
         349     380
```

Fine Mesh = Independent

(1st BID in LIST)

MSC Software®

Independent vs. Dependent

- **Why is having the fine mesh independent better?**
  - Coarse mesh is <u>interpolated</u> if fine mesh is independent
  - Fine mesh is <u>extrapolated</u> if coarse mesh is independent

MSC Software®
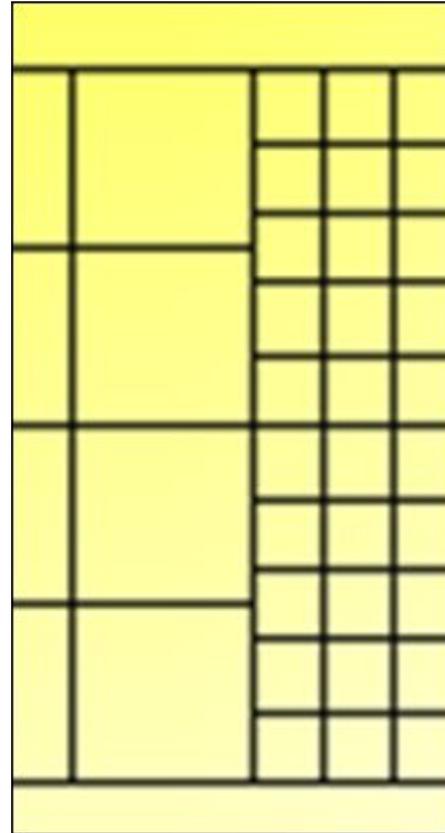
CINTC – BULK DATA ENTRY

Defines a line interface element with specified boundaries.

Format:

	1	2	3	4	5	6	7	8	9	10
	CINTC	EID	TYPE							
	LIST = (BID1(INTP1), BID2(INTP2),...,BIDn(INTPn))									

Example:

	CINTC	1001	GRDLIST							
	LIST=(101,102(Q),-103(Q),104(L))									

- 1st ID in list in Independent by default
- Placing a (-) in front of a listid makes it independent
- (L)=Linear
- (Q)=Quadratic

Field   **Contents**

EID   Element identification number. (0 < Integer < 100,000,000)

TYPE   Connectivity. If TYPE = "GRDLIST" or blank (Default), the user will specify the boundaries via Bulk Data entry, GMBNDC. See Remark 2. (Character; Default = "GRDLIST")

MSC Software

CINTC – BULK DATA ENTRY (Cont.)

Field	Contents Cont..

BIDi   Boundary curve identification number, referenced to Bulk Data entry, GMBNDC. See Remark 2.

INTPi   Interpolation scheme. (Character; Default = "L")

      INTP = "L": Linear interpolation;

      INTP = "Q": Quadratic interpolation.

CINTC – BULK DATA ENTRY (Cont.)

Remarks:

1. Line interface element identification numbers must be unique with respect to all other line interface elements.

2. There must be at least two **BIDi** specified. If all **BIDi** are positive, by default, the degrees of freedom associated with the grids on the boundary represented by the first BID will be taken as the independent (n-set), and the degrees of freedom with the grids on the rest of boundaries are taken as the dependent (m-set). If there is a single negative BID, the degrees of freedom associated with the grids on the boundary represented by this BID will be taken as the independent (n-set), and the rest of the degrees of freedom with other boundaries are used as the dependent (m-set). If there are two or more negative BIDs, the degrees of freedom with the first negative one will be taken as the independent.

3. Forces of multipoint constraints may be recovered with the MPCFORCE Case Control command.

4. The m-set degrees of freedom specified on the boundary grids by this entry may not be specified by other entries that define mutually exclusive sets.

Some Rules…
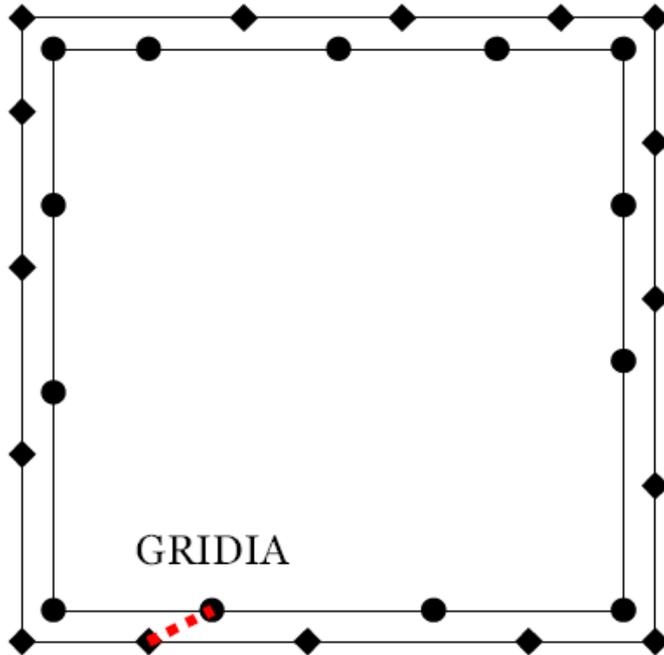
Acceptable Selection of Initial
Grids GRIDIA and GRIDIB
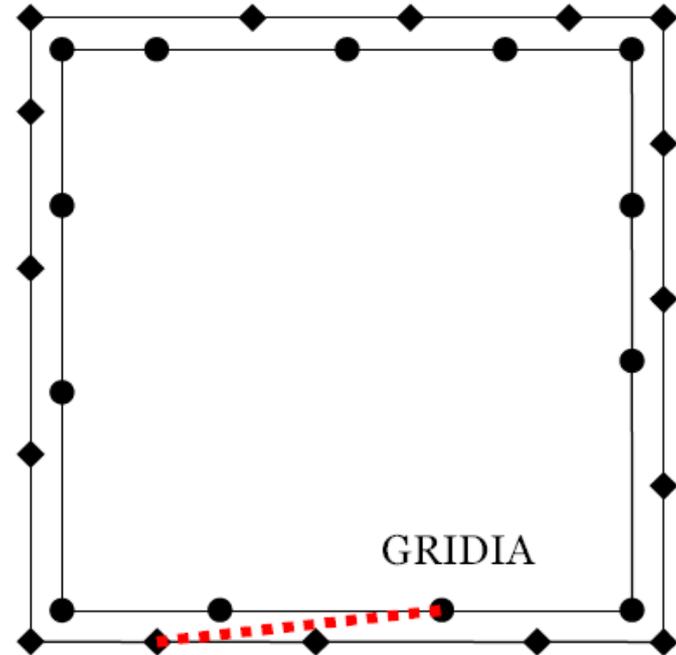
Unsupported Selection of Initial
Grids GRIDIA and GRIDIB

Some More rules…

- **Best to have matching boundaries, but not a <u>MUST</u> requirement!**

Matching Boundaries
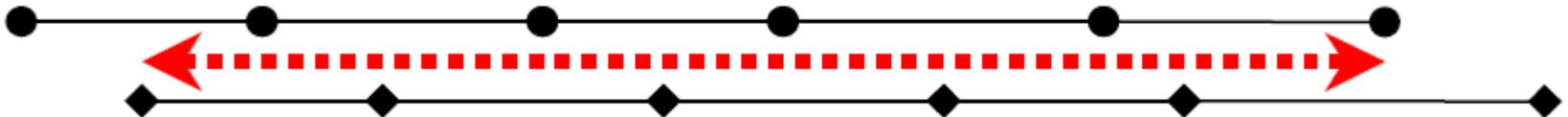
Non-matching Boundaries and Integration Path

Some More rules…

- **Dangling chads?**
  - Detected and excluded from processing

Redundant Dangling Grids
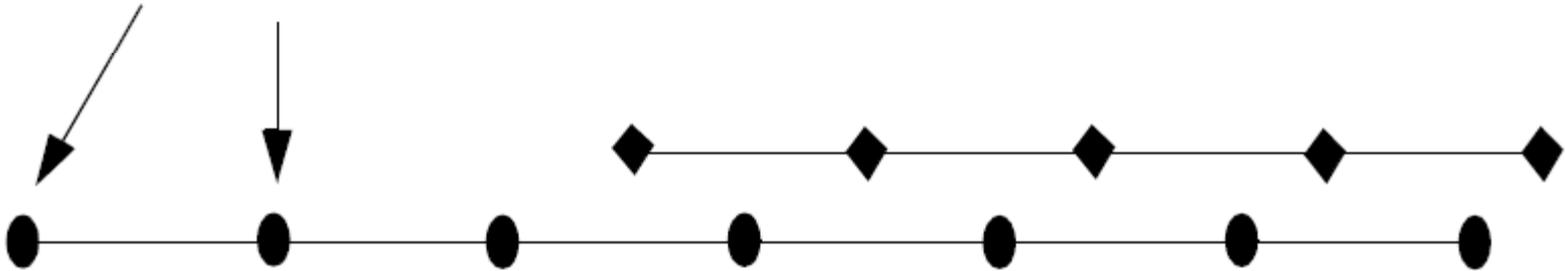
Another Recommendation…

- • • • • • • Actual structure geometry

◆—◆ Finite element model on boundary A

●—● Finite element model on boundary B

- No Checks on Boundary

- User must place grids on actual boundary

- Unexpected results could occur if FEM is not on actual geometry
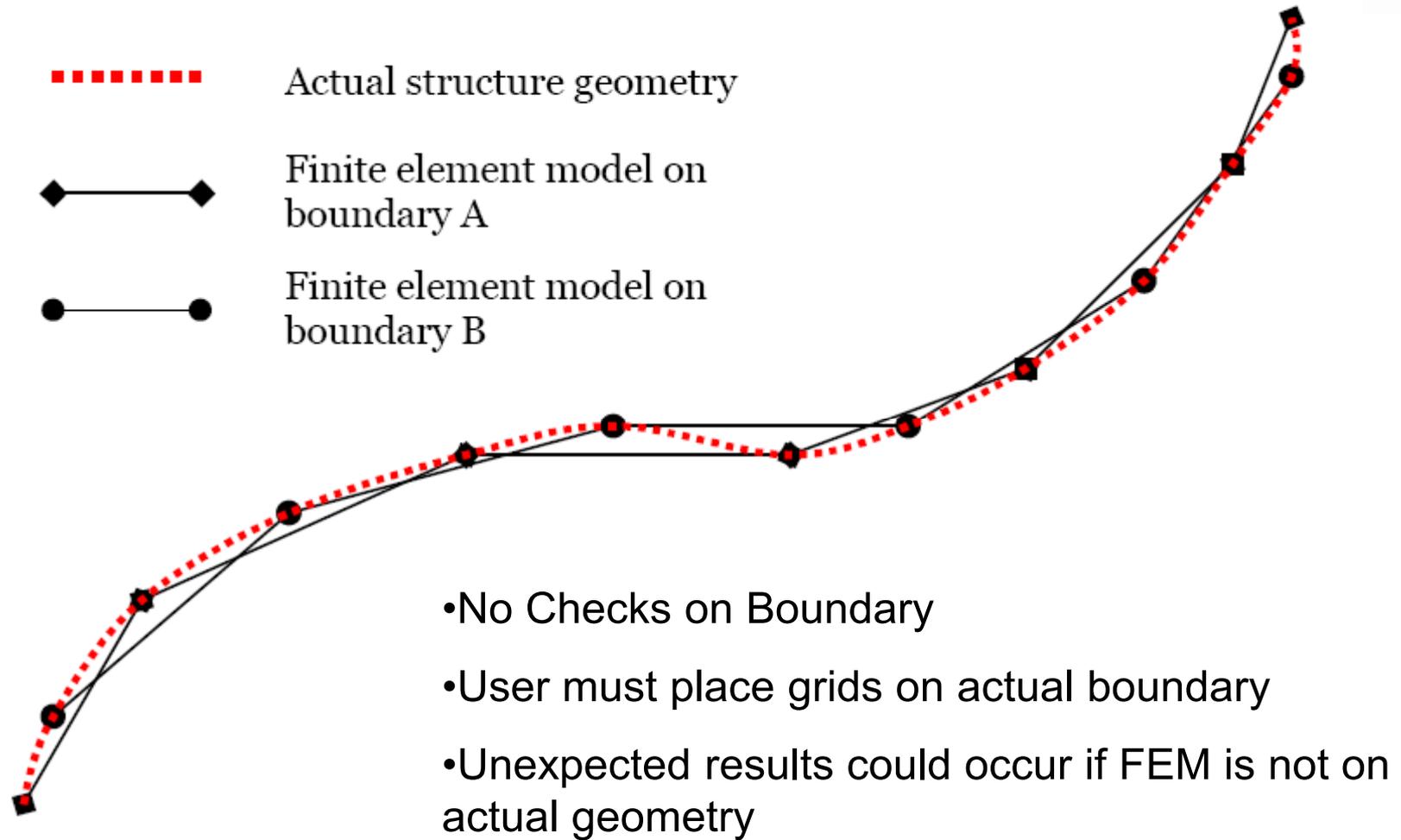
Limitation: GROUNDCHECK ISSUE

- **CINTC passes ground check for ONLY planar problems!**
  - Curved shells do not pass

MSC Software®

CONNECTORS

- **The connector family of elements enable you to provide a <u>flexible</u> connection between two surfaces with <u>differing</u> mesh densities.**

- **There are three types:**
  - **Spot Weld** (CWELD)
  - **Fastener** (CFAST)
  - **Welded Seam** (CWSEAM)

- **This presentation will cover the CWELD and CFAST with an emphasis on the <u>Fastener</u>.**

MSC Software

CONNECTOR PERFORMANCE

- **Good agreement with results from beam and shell theory**

- **6 rigid body modes for all cases: curved geometry, midside nodes, etc.**

- **More accurate than traditional modeling tools like RBAR, RBE2, RBE3 etc.**

- **Constraints are always of proper rank, results are free of spurious modes**

- **CWELD has option without constraints (<u>no</u> m-set DOF)**

- **No K6ROT or SNORM necessary**

- **Minimum number of DOF and constraints**

MSC Software®
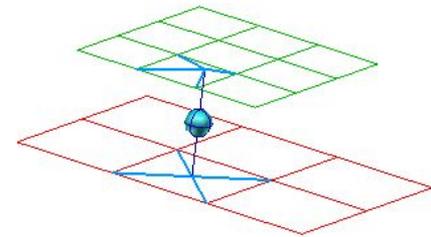
CFAST BENEFITS

- **Existing weld elements focus on providing flexibility and capability at the connection points and the method in which the connector is defined.**

  - These elements enable patch-to-patch or shell-to-shell connections

  - Multi-element connections

  - Alternate methods for projecting the connection onto the two surfaces.

  - Previously the connector itself was considered to be rigid in user defined degrees-of-freedom

- **CFAST element encompasses the above capabilities plus**

  - The ability to define the properties of the connector itself

  - Stiffness, mass, damping

CFAST CONNECTIVITY

- **The CFAST only allows the "Patch-to-Patch" type of connection.**

    – Type:PROP is equivalent to the CWELD PARTPAT

    – Type:ELEM is equivalent to the CWELD ELPAT

patch to patch
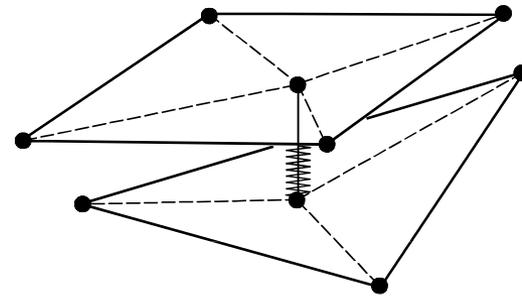
for non congruent meshes area connection

[Type: PROP & ELEM]

MSC Software®

PATCH-TO-PATCH

- **Connection between two shell patches**

- **Pierce points defined by GS or by GA & GB explicitly**

- **Select shell patches with PID or EID**

- **Depending on the location of GA & GB and the Diameter, as many as 9 elements (3x3) on each patch may be used for the connection**

MSC Software®
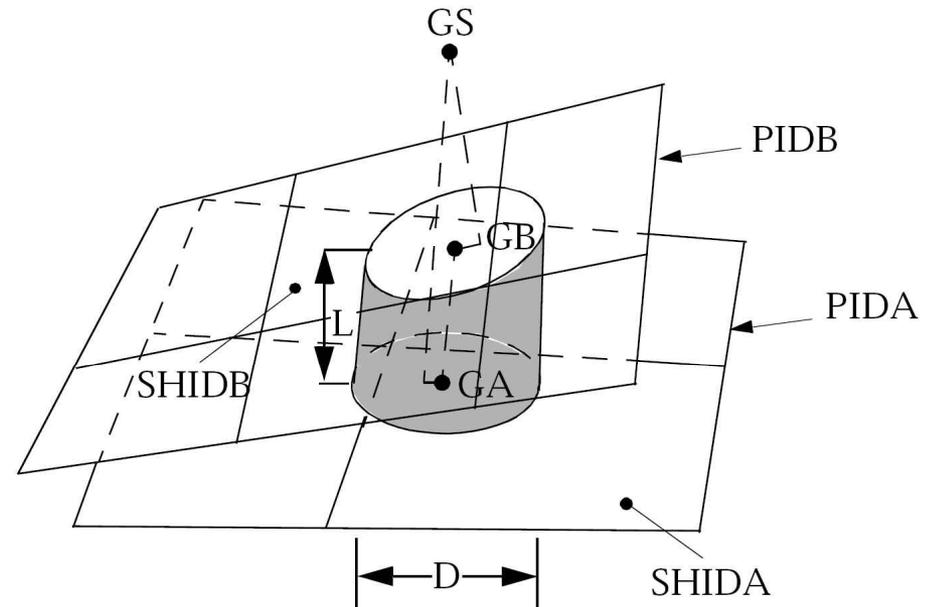
PIERCE POINTS GA & GB

- **Connection points may be calculated automatically.**

  Given grid GS

  1. Normal projection to GA' and GB'
  2. GC' determined between GA' and GB'
  3. GC' projected normal to Shell A and Shell B to locate GA and GB
  4. Connector located at midpoint GC

- **GS may be anywhere… outside of parts, between parts, or on either part.**

MSC Software
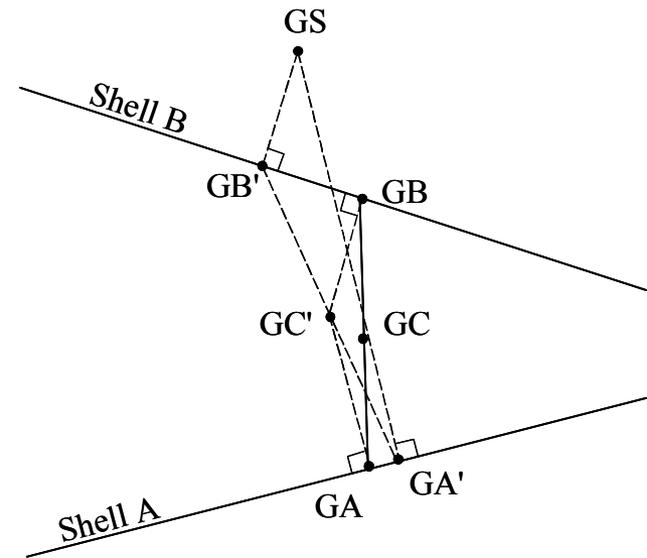
CFAST BULK DATA

Defines a fastener with material orientation connecting two surface patches. Large displacement and large rotational effects are supported in SOL 600 and MSC Nastran SOL 400.

	1	2	3	4	5	6	7	8	9	10
CFAST	EID	PID	TYPE	IDA	IDB	GS	GA	GB	
	XS	YS	ZS						

Field	Contents
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PFAST entry. (Integer > 0; Default =EID)
TYPE	Specifies the surface patch definition: (Character) If TYPE = 'PROP', the surface patch connectivity between patch A and patch B is defined with two PSHELL (or PCOMP) properties with property ids given by IDA and IDB. If TYPE = 'ELEM', the surface patch connectivity between patch A and patch B is defined with two shell element ids given by IDA and IDB. IDA,IDB Property id (for PROP option) or Element id (for ELEM option)defining patches A and B. (Integer > 0)
GS	Grid point defining the location of the fastener. See Remark **2. (Integer >**0 or blank)
GA,GB	Grid ids of piecing points on patches A and B. See Remark **2. (Integer >**0 or blank)
XS,YS,ZS	Location of the fastener in basic. Required if neither GS nor GA is defined.

MSC Software

CFAST BULK DATA (Cont.)

- **Remarks:**

1. The CFAST defines a flexible connection between two surface patches. Depending on the location for the piercing points GA and GB, and the size of the diameter D (see PFAST), the number of unique physical grids per patch ranges from a possibility of 3 to 16 grids. (Currently there is a limitation that there can be only a total of 16 unique grids in the upper patch and only a total of 16 unique grids in the lower patch. Thus, for example, a patch can not hook up to four CQUAD8 elements with mid side nodes and no nodes in common between each CQUAD8 as that would total to 32 unique grids for the patch.)
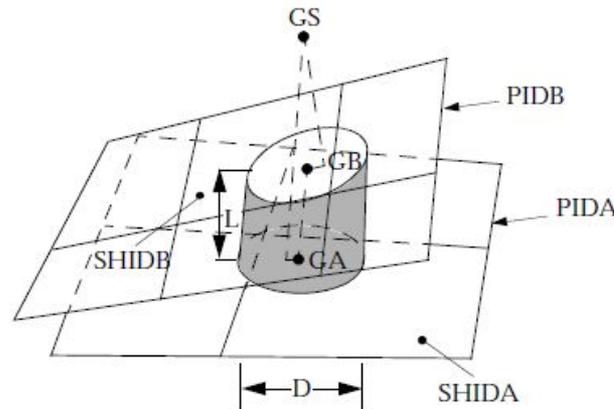
Patches Defined with TYPEj= 'PROP' or TYPE = 'ELEM'

CFAST BULK DATA (Cont.)

2. **GS** defines the approximate location of the fastener in space. GS is projected onto the surface patches A and B. The resulting piercing points GA and GB define the axis of the fastener. GS does not have to lie on the surfaces of the patches. GS must be able to project normals to the two patches. GA can be specified in lieu of GS, in which case GS will be ignored. If neither GS nor GA is specified, then (XS, YS, ZS) in basic must be specified. If both GA and GB are specified, they must lie on or at least have projections onto surface patches A and B respectively. The locations will then be corrected so that they lie on the surface patches A and B within machine precision. The length of the fastener is the final distance between GA and GB. If the length is zero, the normal to patch A is used to define the axis of the fastener.

3. **Only GS or GA & GB or XS,YS,ZS need to be input. If GA is specified, GS is ignored.**

4. **Diagnostic printouts, checkout runs and control of search and projection parameters are requested on the SWDPRM Bulk Data entry.**

5. The use of **param,cfdiagp,yes** and **param,cfrandel,real_fraction_value** allows for the random removal of a percentage of CFAST elements for failure studies.

PFAST BULK DATA

Defines the CFAST fastener property values

1	2	3	4	5	6	7	8	9	10
PFAST	PID	D	MCID	MFLAG	KT1	KT2	KT3	KR1	
	KR2	KR3	MASS	GE					

Field	Contents
PID	Property identification number. (Integer > 0)
D	Diameter of the fastener. See Remark 2. (Real > 0)
MCID	Specifies the element stiffness coordinate system. See Remark 1. (Integer > -1 or blank; Default = -1)
MFLAG	Defines if the coordinate system defined by MCID is absolute or relative. Integer 0 or 1;Default = 0) If MFLAG = 0, MCID defines a relative coordinate system. If MFLAG = 1,MCID defines an absolute coordinate system.
KTi	Stiffness values in directions 1 through 3. (Real)
KRi	Rotational stiffness values in directions 1 through 3. (Real; Default = 0.0)
MASS	Lumped mass of fastener. (Real; Default = 0.0)
GE	Structural damping. (Real; Default = 0.0)

MSC Software

PFAST BULK DATA (Cont.)

Remarks:

1.

   a. If $MCID \geq 0$ and $MFLAG = 0$ (Default), then the KT1 stiffness will be applied along the x_{elem} axis direction of the fastener defined as

$$\vec{e}_1 = \frac{\vec{x}_B - \vec{x}_A}{\left\| \vec{x}_B - \vec{x}_A \right\|}$$

The T2 direction defined by MCID will be used to define the orientation vector $\vec{v}$ of the fastener. Then the element z_{elem} axis will be defined as

$$\vec{e}_3 = \frac{\vec{e}_1 \times \vec{v}}{\left\| \vec{e}_1 \times \vec{v} \right\|}$$

The KT3 stiffness will lie along the z_{elem} axis. The element y_{elem} axis is defined as

$$\hat{e}_2 = \hat{e}_3 \times \hat{e}_1$$

The KT2 stiffness will lie along the y_{elem} axis

This option allows the user to define orthotropic material properties normal to the axis of the fastener that will "slide" with the curve of the patches.

b. If MICD = -1, MFLAG is ignored, and the following element system is defined: the x_{elem} axis direction of the fastener defined as

$$\hat{e}_1 = \frac{\bar{x}_B - \bar{x}_A}{\|\bar{x}_B - \bar{x}_A\|}$$

Relative to the basic system, find the smallest component j of the element x_{elem} axis unit vector. If two such components are equal, take the first one. Form a unit vector in the basic system. For example, assuming the $j = 3$ component of $\hat{e}_1$ was the smallest.

$$b_j = b_3 = \begin{Bmatrix} 0 \\ 0 \\ 1 \end{Bmatrix}$$

Form the following orthogonal vector:

$$\hat{e}_2 = \bar{b}_j - \frac{\hat{e}_1 \cdot \bar{b}_j}{\hat{e}_1 \cdot \hat{e}_1} \hat{e}_1$$

$$\hat{e}_2 = \frac{\bar{e}_2}{\|\bar{e}_2\|}$$

Form $\hat{e}_3$ as

$$\hat{e}_3 = \hat{e}_1 \times \hat{e}_2$$

c. If MCID $\geq$ 0 and MFLAG = 1, then the MCID will be used to compute stiffness. KT1 will be applied along the MCID T1 axis, KT2 along the MCID T2 axis, and KT3 along the MCID T3 axis. The element forces will be computed in the coordinate system defined in Remark 1b.

d. If the length of GA - GB is zero, then the element x_{elem} axis is defined to lie along the projected normal to patch A.

PFAST BULK DATA (Cont.)

2. The diameter D is used along with the piercing points of GA and GB to determine the location of fictitious grid points to form a fictitious hexa volume that determines the elements and physical grids used for the fastener element. Four points are positioned at $\pm a$ positions parallel to the element axis where $a = f(D)$. The stiffness contribution of the fastener depends on both the stiffness values specified and the diameter D. It is a function of D, because the $\pm a$ positions are used along with the surface shape functions of the fictitious hexa to weight the contribution of the physical grids used to the grids GA and GB of the fastener element.

3. The CFAST element (see Figure 8-170), for stiffness and structural damping calculations, is designed to satisfy rigid body equilibrium requirements. When $\dot{x}_B - \dot{x}_A$ has finite length, internal rigid links connect grids GA and GB. This may result in coupling between translational and rotational degrees-of freedom even when no rotational stiffness (KR1-KR3) are specified.

For mass calculations, half the specified mass value is placed directly onto the projected grid A and grid B translational degrees-of-freedom.
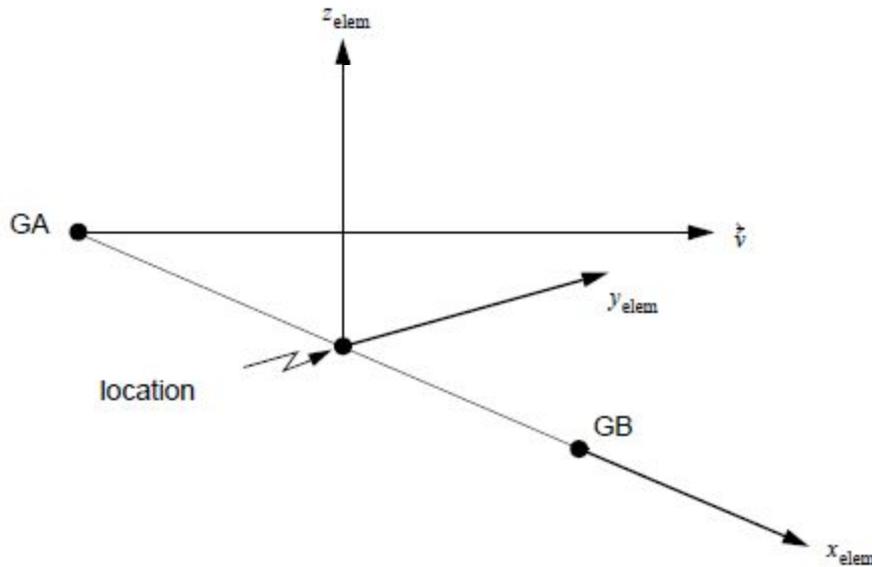
Figure 8-170   CFAST Element

PFAST BULK DATA (Cont.)

4. The CFAST Element lies midway between GA and GB.

5. Values for KTi and KRi are specified at the user's discretion. Assuming a short stubby beam where shear is dominate, possible values might be:

$$KT1 = \frac{EA}{L}$$

$$KT2 = \frac{G_2 A_s}{L}$$

$$KT3 = \frac{G_3 A_s}{L}$$

$$KR1 = \frac{GJ}{L}$$

$$KR2 = \frac{EI}{L} + \frac{G_2 A_s L}{3}$$

$$KR3 = \frac{EI}{L} + \frac{G_3 A_s L}{3}$$

where

$$A = \pi D^2 / 4$$

$$I = \tau D^4 / 64$$

$$J = \pi D^4 / 32$$

$$L = \left| \vec{x_B} - \vec{x_A} \right|$$

$$A_s = A_s = A / \alpha_s$$

$$\alpha_s = 4 / 3$$

E, G_2, G_3, and G are the material properties of the fastener.

PFAST BULK DATA (Cont.)

The fastener stiffness is not, however, independent of the surrounding structure. The values of stiffness specified should not overwhelm the stiffness of the local structure or max ratio's will occur. One possible way to estimate the local stiffness S is by the relationship.

$$S = \frac{t_p E_p E}{E_p + E}$$

where t_p is a shell thickness and E_p is the modulus of the shell.

6. The element force and strain are computed as follows:

$\{f_e\} = [K_e]\{u_e\}$ for statics

$\{f_e\} = ([K_e] + i(g + g_e)[K_e])(\{u_e\}_{real} + i\{u_e\}_{imag})$ for frequency

$\{f_e\} = [K_e]\{u_e\} + \left(\frac{g}{w3} + \frac{g_e}{w4}\right)[K_e]\{v_e\}$ for transient

where $[K_e]$ is the 6 x 6 element stiffness matrix, $\{u_e\} = \{u_b\} - \{u_a\}$ relative displacement in the element coordinate system, and $\{v_e\} = \{v_b\} - \{v_a\}$ relative velocity in the element coordinate system. The subscripts a and b stand for end A and end B of the fastener. g is defined by param,g; $w3$ is defined by param,w3, $w4$ is defined by param,w4; and g_e is the GE entry of the PFAST. $\{u_e\}$ is the strain output. Stress output is the same as force output.

PARAMETERS FOR CFAST

- **These two parameters enable you to randomly remove a percentage of CFAST elements for failure studies.**

CFDIAGP   **Default = NO**, If YES, randomly deleted CFAST elements will be printed. (See CFRANDEL**)**

CFRANDEL   **Default = 0**, Represents a percent, expressed as a decimal fraction, of the number of CFAST elements to be randomly deleted

Example CFAST

```
sol 101
cend
load = 10
set 7 = 777
force=7
stress=all
disp=all
begin bulk
param,post,0
$
$    BOTTOM GRIDS
$
$    TOP GRIDS
$
```

```
$      BOTTOM ELEMENTS
cquad4      101         2         201       202       206       205
cquad4      102         2         202       203       207       206
.  .  .  .  .  .
.  .  .  .  .  .
cquad4      108         2         210       211       215       214
cquad4      109         2         211       212       216       215
pshell      2           100       .1        100
$    TOP ELEMENTS:
cquad4      1           2         101       102       106       105
cquad4      2           2         102       103       107       106
.  .  .  .  .  .
.  .  .  .  .  .
cquad4      8           2         110       111       115       114
cquad4      9           2         111       112       116       115
pshell      20          100       .1        100
$
mat1        100         1.+6                .3
$
force       10          104                 1000.     1.
force       10          116                 1000.     1.
```

MSC Software®

Example CFAST (Cont.)

```
$PRTSW 1=print diagnostic output in exponential format to f06 file
swldprm   prtsw      1
$1.....><2.....><3.....><4.....><5.....><6.....><7.....><8.....><9.....>
$CFAST   EID      PID      TYPE     IDA      IDB      GS       GA       GB
CFAST   777      1000     ELEM     105      5        999
$
$1.....><2.....><3.....><4.....><5.....><6.....><7.....><8.....><9.....>
$PFAST   PID      D        MCID     MFLAG    KT1      KT2      KT3      KR1
$        KR2      KR3      MASS     GE
PFAST   1000     30.                        1.18+8   4.53+8   4.53+8   5.09+9
        6.62+9   6.62+9
$
grid     999                       15.      15.      .005                123456
enddata
```

MSC Software®

CFAST RESULT OUTPUT

- **SWLDPRM,PRTSW,1   Output**

```
CFAST EID=        777 WITH ELEM OR PROP
        AUXILIARY POINTS= ( 1.7066E+00, 1.7066E+00, 0.0000E+00)  ( 2.8293E+01, 1.7066E+00, 0.0000E+00)
                         ( 2.8293E+01, 2.8293E+01, 0.0000E+00)  ( 1.7066E+00, 2.8293E+01, 0.0000E+00)
                         ( 1.7066E+00, 1.7066E+00, 1.0000E-02)  ( 2.8293E+01, 1.7066E+00, 1.0000E-02)
                         ( 2.8293E+01, 2.8293E+01, 1.0000E-02)  ( 1.7066E+00, 2.8293E+01, 1.0000E-02)
        NUMBER OF TIMES GS MOVES=    0
        NUMBER OF TIMES DA IS REDUCED=    0
        ANGLE BETWEEN TWO SHELL NORMALS=    0.00
        GS=( 1.500E+01, 1.500E+01, 5.000E-03) GA=( 1.500E+01, 1.500E+01, 0.000E+00) GB=( 1.500E+01, 1.500E+01, 1.000E-02)
        T_BE MATRIX:    0.0000E+00    1.0000E+00    0.0000E+00
                        0.0000E+00    0.0000E+00    1.0000E+00
                        1.0000E+00    0.0000E+00    0.0000E+00
        GA ID    = 101000001    GB ID    = 101000002
        PATCH A:  EID=     101    GIDS=       201     202     206     205     0     0     0     0
                  EID=     103    GIDS=       203     204     208     207     0     0     0     0
                  EID=     109    GIDS=       211     212     216     215     0     0     0     0
                  EID=     107    GIDS=       209     210     214     213     0     0     0     0
        PATCH B:  EID=       1    GIDS=       101     102     106     105     0     0     0     0
                  EID=       3    GIDS=       103     104     108     107     0     0     0     0
                  EID=       9    GIDS=       111     112     116     115     0     0     0     0
                  EID=       7    GIDS=       109     110     114     113     0     0     0     0


                F O R C E S    I N    F A S T E N E R    E L E M E N T S    ( C F A S T )

ELEMENT_ID         FORCE-X        FORCE-Y        FORCE-Z        MOMENT-X       MOMENT-Y       MOMENT-Z
      777          0.0       2.000000E+03   -5.170960E-10   6.467096E-10   1.722580E-07   1.000000E+01


                S T R E S S    I N    F A S T E N E R    E L E M E N T S    ( C F A S T )

ELEMENT_ID         FORCE-X        FORCE-Y        FORCE-Z        MOMENT-X       MOMENT-Y       MOMENT-Z
      777          0.0       2.000000E+03   -5.170960E-10   6.467096E-10   1.722580E-07   1.000000E+01
```

MSC Software

CWELD BULK DATA

Defines a weld or fastener connecting two surface patches or points. Large displacement and large rotational effects are supported when using SOL 600 and MSC Nastran SOL 400 only.

Format PARTPAT:

1	2	3	4	5	6	7	8	9	10
CWELD	EWID	PWID	GS	"PARTPAT"	GA	GB		MCID	
	PIDA	PIDB							
	XS	YS	ZS						

Format ELPAT:

CWELD	EWID	PWID	GS	"ELPAT"	GA	GB		MCID	
	SHIDA	SHIDB							
	XS	YS	ZS						

Format ELEMID:

CWELD	EWID	PWID	GS	"ELEMID"	GA	GB		MCID	
	SHIDA	SHIDB							

Format GRIDID:

CWELD	EWID	PWID	GS	"GRIDID"	GA	GB	SPTYP	MCID	
	GA1	GA2	GA3	GA4	GA5	GA6	GA7	GA8	
	GB1	GB2	GB3	GB4	GB5	GB6	GB7	GB8	

Format ALIGN:

CWELD	EWID	PWID		"ALIGN"	GA	GB		MCID	

CWELD BULK DATA (Cont.)

Field	Contents
EWID	CWELD element identification number. $0 <$ Integer $< 100,000,000$ Default=Required
PWID	Property identification number of a PWELD entry. Integer > 0 Default=Required
GS	Identification number of a grid point which defines the location of the connector. Integer > 0 or blank
"PARTPAT"	Character string indicating the type of connection. The format of the subsequent entries depends on the type. "PARTPAT", for example, indicates that the connectivity of surface patch A to surface patch B is defined with two property identification numbers of PSHELL entries, PIDA and PIDB respectively. The "PARTPAT" format connects up to 3x3 elements per patch. Type=Character Default=Required
GA, GB	Grid point identification numbers of piercing points on surface A and surface B, respectively. Integer > 0 or blank Default=Blank
MCID	Specifies the element stiffness coordinate system Integer > -1 or blank Default = -1
PIDA, PIDB	Property identification numbers of PSHELL entries defining surface A and B respectively. Integer > 0 Required for "PARTPAT"
XS, YS, ZS	Coordinates of spot weld location in basic. Real Required if GS and GA are not defined.

CWELD BULK DATA (Cont.)

Field   **Contents**

"ELPAT"   Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA and SHIDB, respectively. The "ELPAT" format connects up to 3x3 elements per patch.
Type=Character   Default=Required

SHIDA,SHIDB   Shell element identification numbers of elements on patch A and B, respectively. Integer > 0 Required for "ELPAT"

"ELEMID" Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA and SHIDB, respectively. The "ELEMID" format connects one shell element per patch.
Type=Character   Default=Required

SHIDA,SHIDB   Shell element identification numbers of elements on patch A and B, respectively. Integer > 0 Required for "ELEMID"

"GRIDID"   Character string indicating that the connectivity of surface patch A to surface patch B is defined with two sequences of grid point identification numbers, GAi and GBi, respectively. The "GRIDID" format connects the surface of any element.
Type=Character   Default=Required

MSC Software®

CWELD BULK DATA (Cont.)

Field	Contents
SPTYP	Character string indicating types of surface patches A and B. SPTYP = "QQ", "TT", "QT", "TQ", "Q" or "T". Type=Character Required for "GRIDID"
GAi	Grid identification numbers of surface patch A. GA1 to GA3 are required. Integer > 0 Required for "GRIDID"
GBi	Grid identification numbers of surface patch B. Integer > 0
"ALIGN"	Character string indicating that the connectivity of surface A to surface B is defined with two shell vertex grid points GA and GB, respectively. Type=Character Default=Required
GA, GB	Vertex grid identification number of shell A and B, respectively. Integer > 0 Required for "ALIGN"
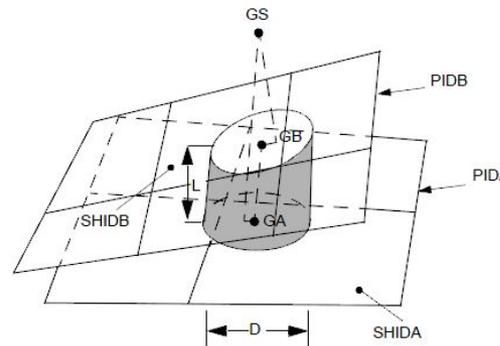
Patch to Patch Connection Defined with the Formats PARTPAT or ELPAT

MSC Software®

CWELD CONNECTIVITY
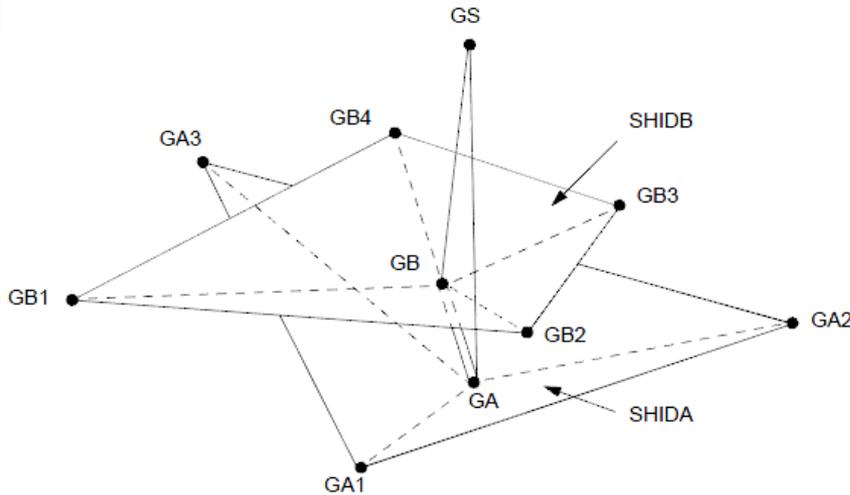
Patch to Patch Connection Defined with Format ELEMID or GRIDID

for non congruent meshes area connection (recommended method)
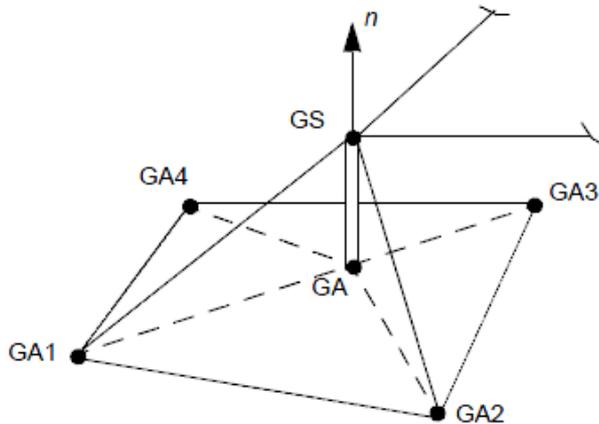
Point to Patch Connection Defined with Format ELEMID or GRID

for non congruent meshes point to area connection (not recommended for spot weld modeling)

MSC Software

CWELD CONNECTIVITY
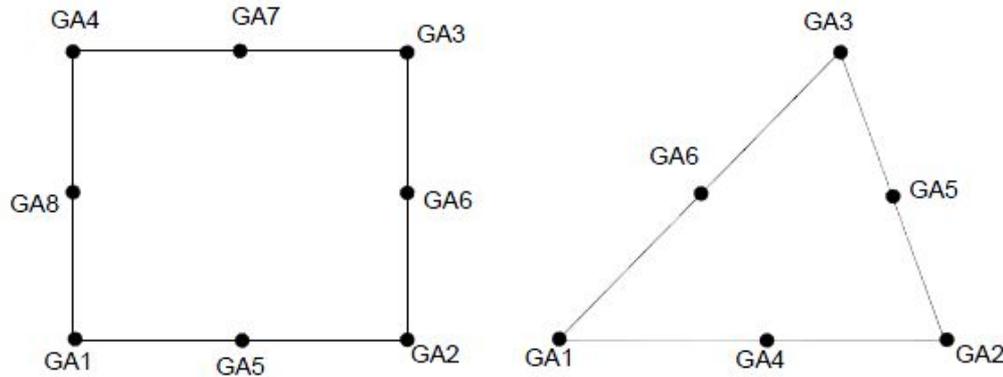
Quadrilateral and Triangular Surface Patches defined with Format GRIDID
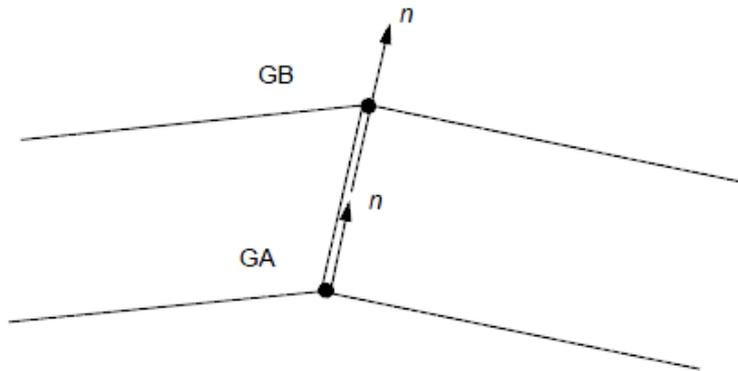
Point to Point Connection Defined with Format ALIGN for nearly congruent meshes point wise connection

CWELD POINT-TO-PATCH

- **Although not recommended for spot welds, this method can be helpful in connecting rigid elements, concentrated masses, etc. to structural parts independent of the mesh.**
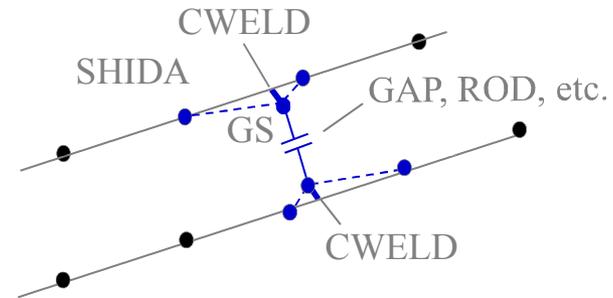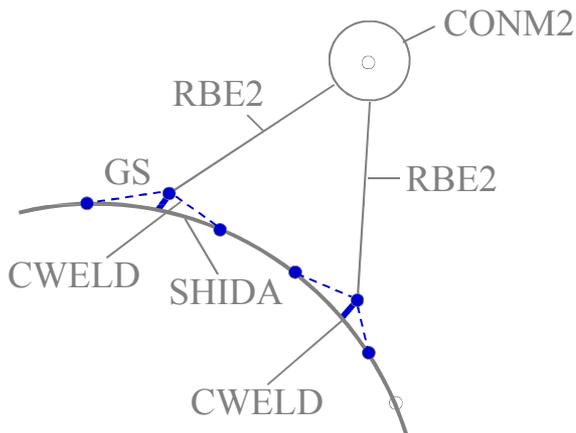
PWELD BULK DATA

Defines the property of connector (CWELD) elements

	1	2	3	4	5	6	7	8	9	10
PWELD	PID	MID	D			MSET		TYPE	
	LDMIN	LDMAX							

Field   **Contents**

PID   Property identification number. Integer > 0 Default=Required

MID   Material identification number. See Remark 1. Integer > 0 Default=Required

D   Diameter of the connector. See Remark 1. Real > 0 Default=Required

MSET   Flag to eliminate m-set degrees-of-freedom (DOFs). The MSET parameter has no effect in a nonlinear MSC Nastran SOL 400 analysis.

          =OFF m-set DOFs are eliminated, constraints are incorporated in the stiffness

          =ON m-set DOFs are not eliminated, constraints are generated. Type=Character Default=OFF

TYPE   Character string indicating the type of connection

          =blank general connector

          = "SPOT" spot weld connector Character Blank

LDMIN   Smallest ratio of length to diameter for stiffness calculation, Real or blank 0.2

LDMAX   Largest ratio of length to diameter for stiffness calculation. Real or blank 5.0

MSC Software

Example CWELD

```
SOL 101

CEND

TITLE= Two elements, identical location for GA, GB, GS

OLOAD= ALL

FORCE= ALL

stress=all

SUBCASE 1

    SUBTITLE= shear the weld

    SPC= 1

    LOAD= 1

    DISP= ALL

SUBCASE 2

    SUBTITLE= in plane twist

    set 21 = 1002,1003,2011,thru,2014

    spc= 1

    LOAD= 2

    DISP(CONN=ALL)=21

SUBCASE 3

    SUBTITLE= upper bending

    set 32 = 4

    set 33 = 1012,1013,2001,thru,2004

    spc= 1

    LOAD= 3

    DISP(CONN=32)=33
```

```
BEGIN BULK

$

$ Grids of lower shell

grid, 1001, , 0., 0., 0.

grid, 1002, , 20., 0., 0.

. . . . .

grid, 1013, , 20., 10., 5.

grid, 1014, , 0., 10., 5.

$ Grids of upper shell

grid, 2001, , 0., 0., 0.0

grid, 2002, , 20., 0., 0.0

. . . . . .

grid, 2013, , 20., 10., 6.0

grid, 2014, , 0., 10., 6.0

$ spot weld grid

grid, 3001, , 10.0, 5.0, 0.0

grid, 3011, , 10.0, 5.0, 10.0

$ quad4s

cquad4, 4001, 10, 1001, 1002, 1003, 1004

cquad4, 5001, 10, 2001, 2002, 2003, 2004

cquad4, 4011, 10, 1011, 1012, 1013, 1014

cquad4, 5011, 10, 2011, 2012, 2013, 2014
```

Example CWELD (Cont.)

```
$ boundary conditions and loads                 $ spot welds
$                                               $
spc1, 1, 123456, 1001, 1004, 1011, 1014         $ diameter 5.0
spc1, 3, 123456, 1001, 1004, 2001, 2004         $1.....><2.....><3.....><4.....><5.....><6.....><7.....><8.....><9.....>
spc1, 3, 123456, 1011, 1014, 2011, 2014         $CWELD   EWID    PWID    GS      PARTPAT GA      GB              MCID
$                                               $       PIDA    PIDB
load, 1, 1., 1.e3, 301                          $       XS      YS      ZS
force, 301, 2002, 0, 0.5, 1., 0., 0.            CWELD   4       4       3001    ELEMID
. . . . . . .                                           4001    5001
$                                               $1.....><2.....><3.....><4.....><5.....><6.....><7.....><8.....><9.....>
load, 3, 1., 100., 303                          $CWELD   EWID    PWID    GS      PARTPAT GA      GB              MCID
force, 303, 2002, 0, 0.25, 0., 0., -1.          $       PIDA    PIDB
. . . . . . .                                   $       XS      YS      ZS
force, 303, 1013, 0, 0.25, 0., 0., -1.          CWELD   5       4       3011    ELEMID
$                                                       4011    5011
load, 2, 1., 1.e3, 302                          $1.....><2.....><3.....><4.....><5.....><6.....><7.....><8.....><9.....>
force, 302, 2002, 0, 0.5, 1., 0., 0.            $PWELD   PID     MID     D                       MSET            TYPE
. . . . . .                                     $       LDMIN   LDMAX
force, 302, 2013, 0, 0.5, -1., 0., 0.           PWELD   4       10      5.0     FF      CLAMP   ON
$                                               $
$ property and material                         enddata
pshell, 10, 10, 1.0, 10
mat1, 10, 2.e+5 , , 0.3, 0.785e-8
```

MSC Software

CWELD RESULTS OUTPUT

								SUBCASE 1

F O R C E S   I N   W E L D   E L E M E N T S   (C W E L D)

	ELEMENT		BEND-MOMENT END-A		BEND-MOMENT END-B		- SHEAR -		AXIAL
ID	PLANE 1 (MZ)	PLANE 2 (MY)	PLANE 1 (MZ)	PLANE 2 (MY)	PLANE 1 (FY)	PLANE 2 (FZ)	FORCE FX	TORQUE MX
4	0.0	0.0	0.0	0.0	1.000000E+03	-4.093896E-14	0.0	6.396713E-13
5	1.000000E+03	3.420015E-10	-1.455192E-09	3.428817E-10	1.000000E+03	-8.801877E-13	-2.328306E-10	1.599178E-11
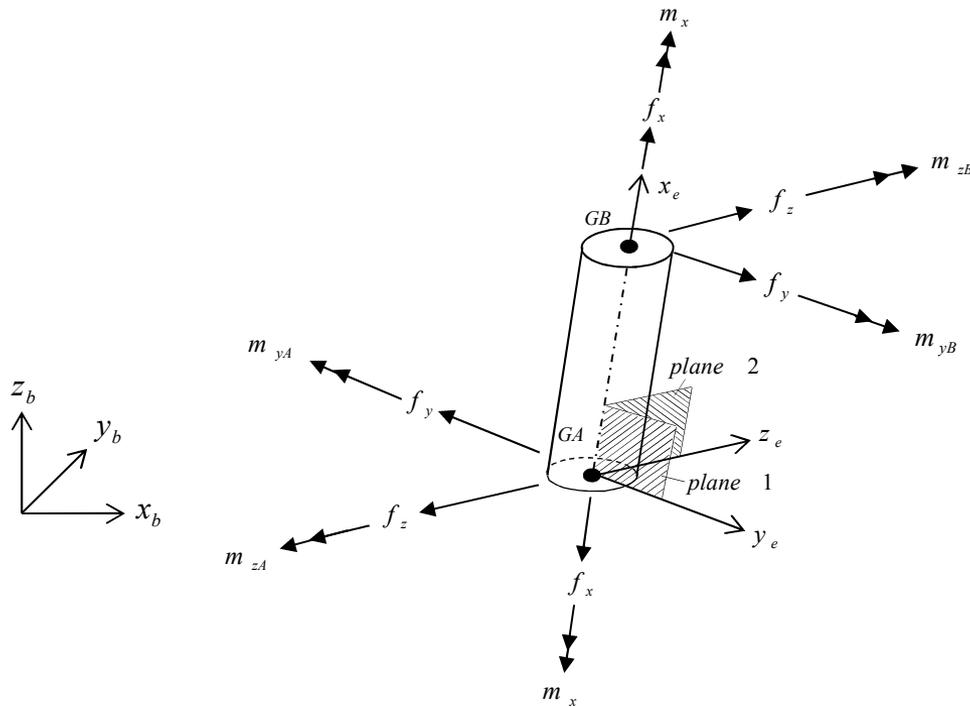
f_x axial force

f_y shear force plane 1

f_z shear force plane 2

m_x torque

m_{yA} bending moment end A, plane 2

m_{yB} bending moment end B, plane 2

m_{zA} bending moment end A, plane 1

m_{zB} bending moment end B, plane 1

CWELD RESULT OUTPUT

CWELD vs. CFAST

- **Internal formulation is identical**

  – CWELD has more connectivity options, but simple properties.

  – CFAST has fewer connectivity options, but more complex property definition.

MSC Software
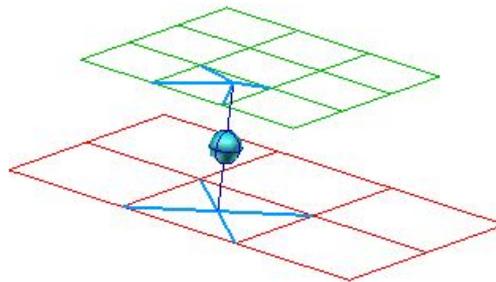
CSEAM ELEMENT

- **CSEAM is a connector elements which defines a seam line to connect two surface patches.**

- **To define the seam weld, user must select two surface patches by their property IDs and specify the width and the thickness of the seam. Also have to specify start point "GS" and end point "GE" of the seam segment.**

- **The CSEAM element provides the following key features:**

    - A seam line is considered continuous between two CSEAM elements that have a common face based on either common GS/GE points or XYZ coordinates. Note that the SMLN label on a CSEAM element does not determine the definition of a seam line. It is only intended for ease of seam line visualization.

    - This element can connect up to 64 shell grids, which allows the connection of higher order shell elements.

    - Besides selecting the connected surface patches by property IDs, the user may define the connection by specifying shell element IDs directly.

    - This element type supports the MAT9 anisotropic material properties.

MSC Software

CSEAM BULK DATA

- **Defines a SEAM connecting two surface patches**

1	2	3	4	5	6	7	8	9	10
CSEAM	EID	PID	SMLN	CTYPE	IDAS	IDBS	IDAE	IDBE	
	GS	GE							

Alternate Format:

1	2	3	4	5	6	7	8	9	10
CSEAM	EID	PID		CTYPE	IDAS	IDBS	IDAE	IDBE	
	XS	YS	ZS	XE	YE	ZE			

Field	Contents
EID	Element identification number. ($0 <$ Integer $< 100,000,000$)
PID	Property identification number of a PSEAM entry. (Integer > 0)
SMLN	SEAM line identification. (CHAR or blank)
CTYPE	Connectivity search type. (Character)
	If CTYPE = "PSHELL", IDAS and IDBS are property identification numbers of PSHELL's. (Default)
	If CTYPE = "ELEM", IDAS and IDBS are element identification numbers.

MSC Software

CSEAM BULK DATA (Cont.)

Field   **Contents**

IDAS,IDBS Used to define patch A and B or the start of patch A or B for a tailored blank.(Integer > 0)

     If CTYPE = "PSHELL", required property id defining patches A and B.

     If CTYPE = "PSHELL" and IDAS = IDBS or IDBS = blank the patch will be considered as two-sided and the property identification numbers of PSHELL's will be the same for both the top and bottom.

     If CTYPE = "ELEM", required element id defining patches A and B. IDAS ≠ IDBS.

IDAE,IDBE Used to define the end of patch A and the end of patch B for a tailored blank. (Integer > 0 or blank)

     If CTYPE = "PSHELL", property id defining patches A and B.

     If CTYPE = 'PSHELL' and IDAE = IDBS or IDBS=blank the patch will be considered as two-sided and the property identification numbers of PSHELL's will be the same for both the top and bottom.

     If CTYPE = "ELEM", element id defining patches A and B. IDAE≠ IDBE.

GS, GE   Grid ids of piercing points on patches A and B of the Start and End of the SEAM. (Integer > 0)

XS,YS,ZS Location of the SEAM Start. (Real or blank)

XE,YE,ZE Location of the SEAM End. (Real or blank)

MSC Software

PSEAM BULK DATA

Defines the PSEAM property values.

	1	2	3	4	5	6	7	8	9	10
	PSEAM	PID	MID	TYPE	W	T				

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
TYPE	"KEYWORD" type of Seam Weld generated. (Character; Default =LINE)
W	Width of the SEAM. (Real > 0.)
T	Thickness of the SEAM. (Real > 0. or blank)
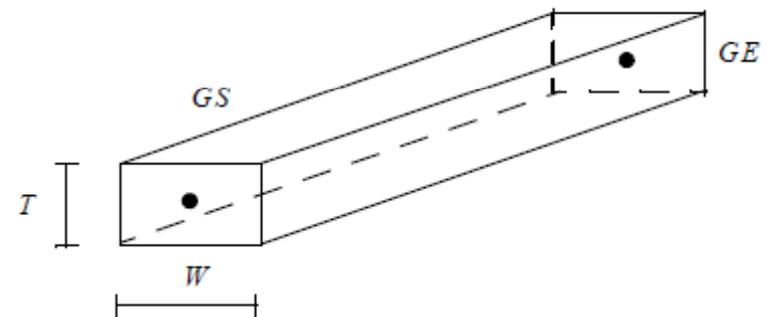
Dimensions of a CSEAM Element

MSC Software

LIMITATIONS

- **Each CSEAM element can connect up to three shell elements on patch A and three shell elements on patch B.**

- **Only Line type of seam is supported.**

- **Super-element modeling is not supported.**

- **FORCE, STRESS and STRAIN output requests are not supported.**

Example CSEAM

```
SOL 101
CEND
TITLE= seam line pshell option
echo=none
SPC= 600
DISPL= ALL
gpforce=all
ese=all
SUBCASE 1
    SUBTITLE= shear the weld
    LOAD= 1
    DISPL= ALL
BEGIN BULK
$
$ Grids of lower shell
grid, 101, , 0., 0., 0.
.  .  .  .  .  .  .  .  .  .
grid, 1022, , 2., .6, 0.
$ Grids of upper shell
grid, 111, , 0., 0., 0.1
.  .  .  .  .  .  .  .  .  .
grid, 1122, , 2., .6, 0.1
$ spot weld grid
grid, 74, , .25, .25, 0.5
grid, 71, , .75, .25, 0.5
grid, 73, ,1.25, .25, 0.5
grid, 72, ,1.75, .25, 0.5
grid, 82, , .25, 0.5, 0.5
grid, 84, , .75, 0.5, 0.5
grid, 83, ,1.25, 0.5, 0.5
grid, 81, ,1.75, 0.5, 0.5
grid, 91, , .25, .75, 0.5
grid, 93, , .75, .75, 0.5
grid, 94, ,1.25, .75, 0.5
grid, 92, ,1.75, .75, 0.5
```

```
$ quad4s
cquad4,  2, 10,  101, 1011, 1016, 1015
cquad4,  3, 10, 1011, 1012, 1017, 1016
cquad4,  4, 10, 1012,  102, 1018, 1017
cquad4,  5, 10, 1015, 1016, 1020, 1019
cquad4,  6, 10, 1017, 1018, 1022, 1021
cquad4,  7, 10, 1019, 1020, 1014,  104
cquad4,  8, 10, 1020, 1021, 1013, 1014
cquad4,  9, 10, 1021, 1022,  103, 1013
cquad4, 10, 10, 1016, 1017, 1021, 1020
cquad4, 11, 20, 1116, 1117, 1121, 1120
cquad4, 12, 20,  111, 1111, 1116, 1115
cquad4, 13, 20, 1111, 1112, 1117, 1116
cquad4, 14, 20, 1112,  112, 1118, 1117
cquad4, 15, 20, 1115, 1116, 1120, 1119
cquad4, 16, 20, 1117, 1118, 1122, 1121
cquad4, 17, 20, 1119, 1120, 1114,  114
cquad4, 18, 20, 1120, 1121, 1113, 1114
cquad4, 19, 20, 1121, 1122,  113, 1113
$
$ boundary conditions and loads
$
spc1, 600, 123456, 101, 104, 1015, 1019
load, 1, 1., 100., 301
force, 301, 112, 0, 0.5, 1., 0., 0.
force, 301, 113, 0, 0.5, 1., 0., 0.
force, 301, 1118, 0, 0.5, 1., 0., 0.
force, 301, 1122, 0, 0.5, 1., 0., 0.
$
$ property and material
pshell, 10, 10, 0.101, 10
pshell, 20, 10, 0.103, 10
mat1, 10, 1.+7, , 0.3, 1.e-4
$
```

Example CSEAM (Cont.)

```
$ seam weld
$1.....><2.....><3.....><4.....><5.....><6.....><7.....><8.....><9.....>
$PSEAM   PID    MID    TYPE    W       T
PSEAM    30     10     LINE    0.1     0.12
$1.....><2.....><3.....><4.....><5.....><6.....><7.....><8.....><9.....>
$CSEAM   EID    PID    SMLN    CTYPE   IDAS    IDBS    IDAE    IDBE
$        GS     GE
CSEAM    77     30     smln_1  pshell  20      10
         74     71
CSEAM    78     30     smln_1  pshell  20      10
         71     73
CSEAM    79     30     smln_1  pshell  20      10
         73     72
CSEAM    87     30             pshell  20      10
         82     84
CSEAM    88     30             pshell  20      10
         84     83
CSEAM    89     30             pshell  20      10
         83     81
$1.....><2.....><3.....><4.....><5.....><6.....><7.....><8.....><9.....>
$CSEAM   EID    PID    SMLN    CTYPE   IDAS    IDBS    IDAE    IDBE
CSEAM    97     30     seamln_2 pshell 20      10
         91     93
CSEAM    98     30     seamln_2 pshell 20      10
         93     94
CSEAM    99     30     seamln_2 pshell 20      10
         94     92
$
swldprm,prtsw,1
$
enddata
```

MSC Software

CSEAM RESULTS OUTPUT

MSC Software®
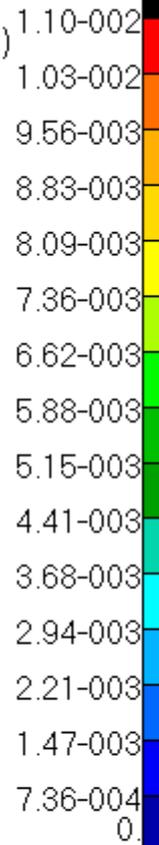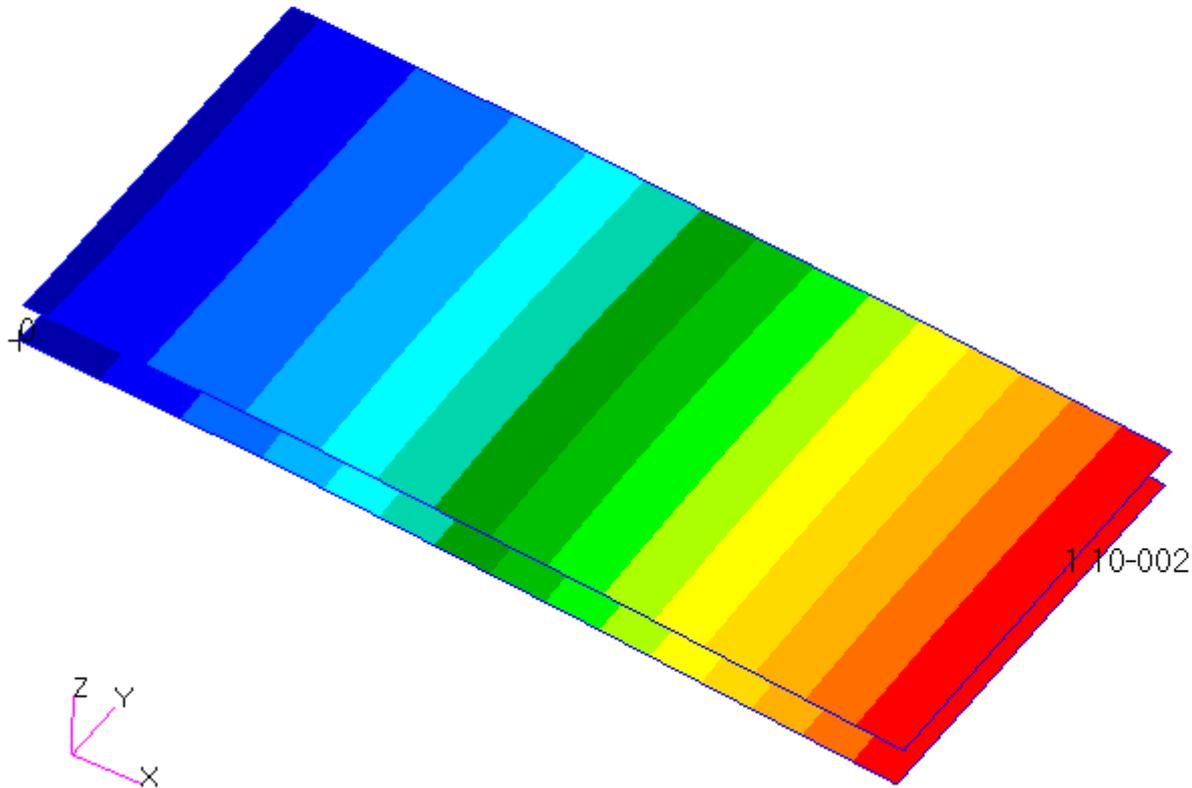
CSEAM RESULTS OUTPUT (Cont.)

SWLDPRM,PRTSW,1   Output

```
CSEAM EID=      77
        AUXILIARY POINTS=  ( 2.5025E-01, 2.9995E-01, 1.1000E-01)   ( 2.5000E-01, 2.0000E-01, 1.1000E-01)
                           ( 2.5000E-01, 2.0000E-01,-1.0000E-02)   ( 2.5025E-01, 2.9995E-01,-1.0000E-02)
                           ( 7.4975E-01, 2.9995E-01, 1.1000E-01)   ( 7.5000E-01, 2.0000E-01, 1.1000E-01)
                           ( 7.5000E-01, 2.0000E-01,-1.0000E-02)   ( 7.4975E-01, 2.9995E-01,-1.0000E-02)
        GS=( 2.501E-01, 2.500E-01, 5.000E-02) GE=( 7.499E-01, 2.500E-01, 5.000E-02)
        T1_S VECTOR:      0.0000E+00  -1.0000E+00   0.0000E+00
        T1_E VECTOR:      2.7756E-17  -1.0000E+00   0.0000E+00
        T2_S VECTOR:      0.0000E+00   0.0000E+00  -1.0000E+00
        T2_E VECTOR:      0.0000E+00   0.0000E+00  -1.0000E+00
        AX GSA1 ID = 101000001   GSA2 ID = 101000002   GSB2 ID = 101000003   GSB1 ID = 101000004
        AX GEA1 ID = 101000005   GEA2 ID = 101000006   GEB2 ID = 101000007   GEB1 ID = 101000008
        STA PATCH A:  EID=      12   GIDS=       111     1111     1116     1115        0        0        0        0
                      EID=      12   GIDS=       111     1111     1116     1115        0        0        0        0
            PATCH B:  EID=       2   GIDS=       101     1011     1016     1015        0        0        0        0
                      EID=       2   GIDS=       101     1011     1016     1015        0        0        0        0
        END PATCH A:  EID=      13   GIDS=      1111     1112     1117     1116        0        0        0        0
                      EID=      13   GIDS=      1111     1112     1117     1116        0        0        0        0
            PATCH B:  EID=       3   GIDS=      1011     1012     1017     1016        0        0        0        0
                      EID=       3   GIDS=      1011     1012     1017     1016        0        0        0        0
```

SOLUTION TYPES

- **Can be used with all Solutions except for SOL600 and SOL700.**

- **The CFAST element, for stiffness and structural damping calculations, is designed to satisfy rigid body equilibrium requirements.**

- **For mass calculations, half the specified mass value is placed directly onto GA and GB translational DOFs.**

MSC Software®

SECTION 4

COMPOSITE ANALYSIS

CONTENTS

CONTENTS

MSC Software

OVERVIEW

- **Classical lamination theory is used.**

- **Family of plate elements, QUAD4, QUAD8, TRIA3, TRIA6, QUADR, and TRIAR available for modeling composites.**

- **User input is simple.**

- **Stress output for user-requested plies is available.**

- **Can be used in optimization (SOL 200)**

- **Failure indices and strength ratio for elements can be requested.**

CLASSICAL LAMINATION THEORY (CLT)

- **By this theory, equations for laminate are derived from those of laminas.**

- **Each individual lamina is in plane stress.**

- **The laminate is presumed to consist of perfectly bonded lamina. allowing no relative slip between layers.**

- **A distinct feature of MSC NASTRAN plate elements is the provision for including transverse shear stiffness:**

$$\left\{ \begin{matrix} V_x \\ V_y \end{matrix} \right\} = [\mathbf{G_3}] \left\{ \begin{matrix} q_{xz} \\ q_{yz} \end{matrix} \right\}$$

- **The effective transverse shear stiffness matrix (G3) for composite plate elements is evaluated on the assumption of the applicability of elementary beam theory type equations for plates. This introduces an approximation that the effects of twisting moments are negligible. In the vast majority of cases such an approximation is satisfactory.**

MSC Software

COMPOSITE MATERIALS

- Composite material is defined as one where two or more materials are combined on a macroscopic scale.

- This is done to obtain the best qualities of the constituent materials (and in some cases, additional qualities that the constituents do not have).

- The following properties are improved and are of major interest:

  Strength

  Stiffness

  Lower weight

  Tailored properties

MSC Software®

CLASSIFICATION OF COMPOSITE MATERIALS

- **Lamina is a group of unidirectional fibers (or sometimes woven fibers) arranged to form a flat or curved load resisting member by the use of a matrix.**

- **The principal material axes are parallel and perpendicular to the fiber directions.**

- **The principal directions are also referred to as:**
  - fiber direction, longitudinal direction or 1-direction
  - matrix direction, transverse direction or 2-direction

UNIDIRECTIONAL FILAMENTARY LAMINA
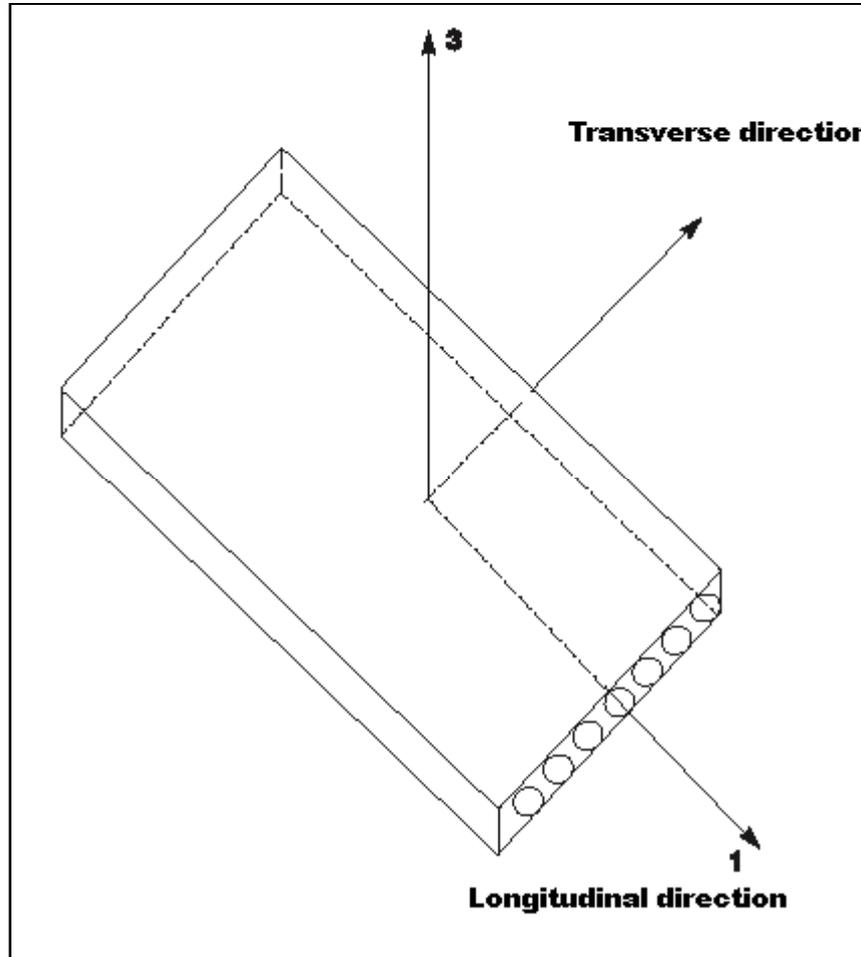
LAMINATE CONSTRUCTION

- A laminate is a stack of lamina arranged with the principal directions of each lamina at different orientations so as to obtain the desired strength and stiffness properties.

- The various layers of a laminate are bonded together by the same matrix material that is used in the lamina.

- Curing bonds the lamina together, usually in the presence of heat and pressure.

LAMINA ARRANGEMENT IN A 0/90/0 LAMINATE
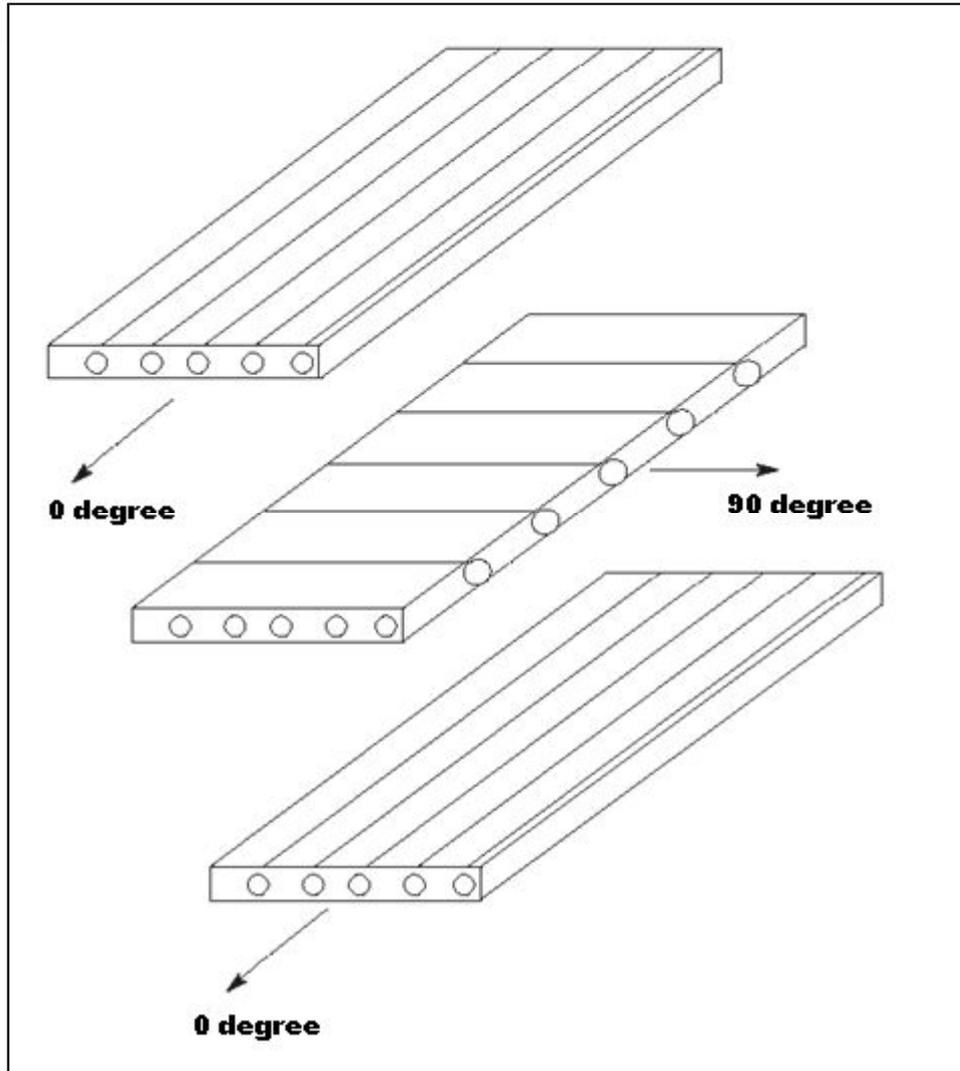
COMPOSITES

- **Composites generally specify lamina properties as a 2-D orthotropic material**

- **The stress-strain relations in principal lamina material directions are**

$$
\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} =
\begin{bmatrix}
\dfrac{E_1}{1 - v_{12}v_{21}} & \dfrac{E_1 v_{21}}{1 - v_{12}v_{21}} & 0 \\[2ex]
\dfrac{E_2 v_{12}}{1 - v_{12}v_{21}} & \dfrac{E_2}{1 - v_{12}v_{21}} & 0 \\[2ex]
0 & 0 & G_{12}
\end{bmatrix}
\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix}
$$

- **There are four independent constants in the relationship,**

  E_1, E_2, v_{12} (or v_{21}), G_{12}

- **In many references, this is also written as**

$$
\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_6 \end{Bmatrix} =
\begin{bmatrix}
Q_{11} & Q_{12} & 0 \\
Q_{12} & Q_{22} & 0 \\
0 & 0 & Q_{66}
\end{bmatrix}
\begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_6 \end{Bmatrix}
\qquad i.e., \{\sigma\} = [Q]\{\varepsilon\}
$$

MSC Software®

ROTATION TO MATERIAL COORDINATE SYSTEM

- **To form laminate properties, first the lamina properties are rotated to the element material coordinate system [x_m Y_m]**
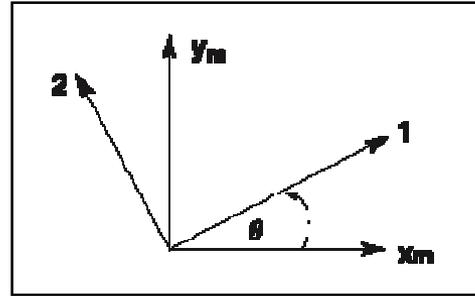
Using the equation:

$$[\overline{Q}] = [U]^T[Q][U]$$

where $\overline{Q}$ **= lamina properties rotated to material coordinate system.**

[u] = the stress-transformation matrix for transforming stresses from the 1-2 system to the x-y system that is given by

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

CALCULATION OF COMPOSITE ELEMENT PROPERTIES

- **The $\overline{Q}$ for the stacked lamina are then integrated through the thickness, to relate the curvatures and mid-surface strains with Forces and Moments:**

$$\left\{\frac{F}{M}\right\} = \begin{bmatrix} A & B \\ B & D \end{bmatrix} \left\{\begin{array}{c} \varepsilon^0 \\ \chi \end{array}\right\}$$

where

$$[A] = \begin{bmatrix} A_{11} & A_{12} & A_{16} \\ A_{12} & A_{22} & A_{26} \\ A_{16} & A_{26} & A_{66} \end{bmatrix}$$

$$[B] = \begin{bmatrix} B_{11} & B_{12} & B_{16} \\ B_{12} & B_{22} & B_{26} \\ B_{16} & B_{26} & B_{66} \end{bmatrix}$$

$$[D] = \begin{bmatrix} D_{11} & D_{12} & D_{16} \\ D_{12} & D_{22} & D_{26} \\ D_{16} & D_{26} & D_{66} \end{bmatrix}$$

CALCULATION OF COMPOSITE ELEMENT PROPERTIES (Cont.)

and where

- A_{ij} – Extensional Stiffness

$$A_{ij} = \sum_{1}^{N} (\overline{Q}_{ij})_k (z_k - z_{k-1})$$

$$A_{ij} = \sum_{1}^{N} (\overline{Q}_{ij})_k t_k$$

- B_{ij} – Coupling Stiffness

$$B_{ij} = \frac{1}{2} \sum_{1}^{N} (\overline{Q}_{ij})_k (z_k^2 - z_{k-1}^2)$$

$$B_{ij} = \sum_{1}^{N} (\overline{Q}_{ij})_k t_k \overline{z}_k$$

- D_{ij} – Bending Stiffness

$$D_{ij} = \frac{1}{3} \sum_{1}^{N} (\overline{Q}_{ij})_k (z_k^3 - z_{k-1}^3)$$

$$D_{ij} = \sum_{1}^{N} (\overline{Q}_{ij})_k (t_k \overline{z}_k^2 + \frac{t_k^3}{12})$$
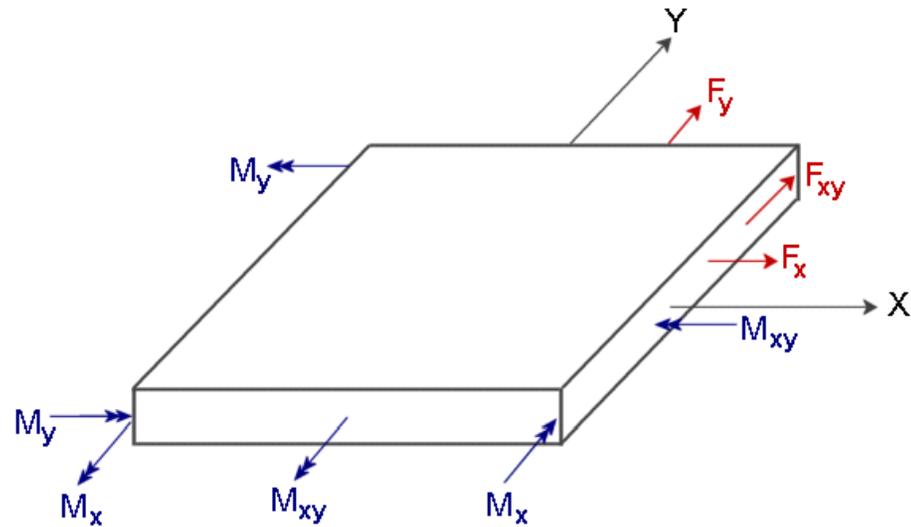
where:

n = number of plies

k = ply number

t_k = ply k thickness

$\overline{z}_k$ = distance from composite center to ply k center

MSC Software®

CALCULATION OF COMPOSITE ELEMENT PROPERTIES (Cont.)

- **MSC NASTRAN uses the G1, G2, G3, and G4 matrices to define element properties.**
- **The relation between forces and strains used for MSC NASTRAN plate elements is**

$$\left\{ \begin{array}{c} F \\ M \\ Q \end{array} \right\} = \left[\begin{array}{ccc} TG_1 & T^2G_4 & 0 \\ T^2G_4 & IG_2 & 0 \\ 0 & 0 & T_sG_3 \end{array} \right] \left\{ \begin{array}{c} \varepsilon_M \\ \chi \\ \gamma \end{array} \right\}$$

where $\{F\} = \left\{ \begin{array}{c} F_x \\ F_y \\ F_{xy} \end{array} \right\},$ **membrane forces per unit length**

$\{M\} = \left\{ \begin{array}{c} M_x \\ M_y \\ M_{xy} \end{array} \right\},$ **bending moments per unit length** $\{\chi\} = \left\{ \begin{array}{c} \chi_x \\ \chi_y \\ \chi_{xy} \end{array} \right\},$ **curvatures;**

strains $\{Q\} = \left\{ \begin{array}{c} Q_x \\ Q_y \end{array} \right\},$ **transverse shear force per unit length** $\{\gamma\} = \left\{ \begin{array}{c} \gamma_x \\ \gamma_y \end{array} \right\},$ **transverse shear**

$\{\varepsilon_M\} = \left\{ \begin{array}{c} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_{xy} \end{array} \right\},$ **membrane strains in mean plane**

MSC Software

CALCULATION OF COMPOSITE ELEMENT PROPERTIES (Cont.)

- **Note that**

$$TG1_{ij} = A_{ij}$$

$$G1_{ij} = \frac{A_{ij}}{T}$$

$$T^2 G4_{ij}$$

$$G4_{ij} = \frac{B_{ij}}{T^2}$$

$$\frac{T^3}{12} G2_{ij} = -D_{ij}$$

$$G2_{ij} = -\frac{D_{ij}}{\left(\frac{T^3}{12}\right)}$$

where $\boldsymbol{T}$ = **Laminate thickness**

$\dfrac{\boldsymbol{T}^3}{12}$ = **I = Bending inertia**

CALCULATION OF COMPOSITE ELEMENT PROPERTIES(Cont.)

- **Note that MSC NASTRAN'S M_x, M_y, and M_{xy} terms are reversed from classical lamination theory.**
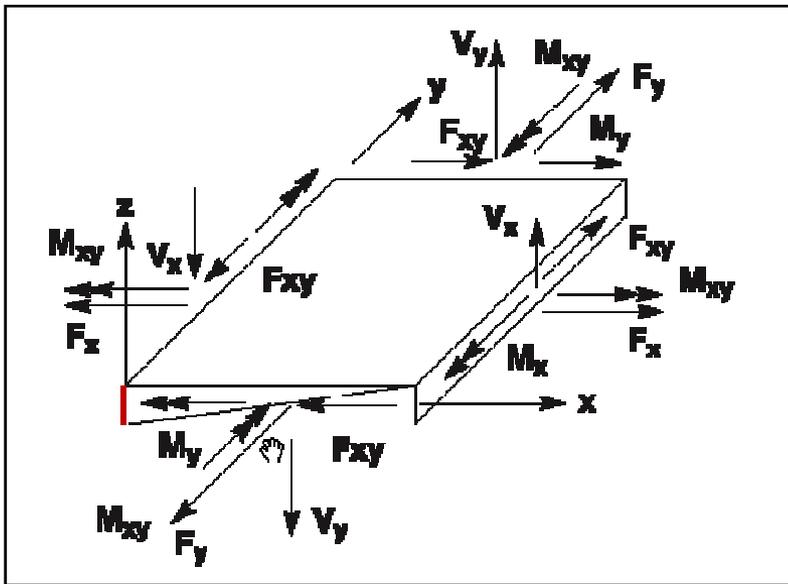
MSC NASTRAN Forces In Plate Elements
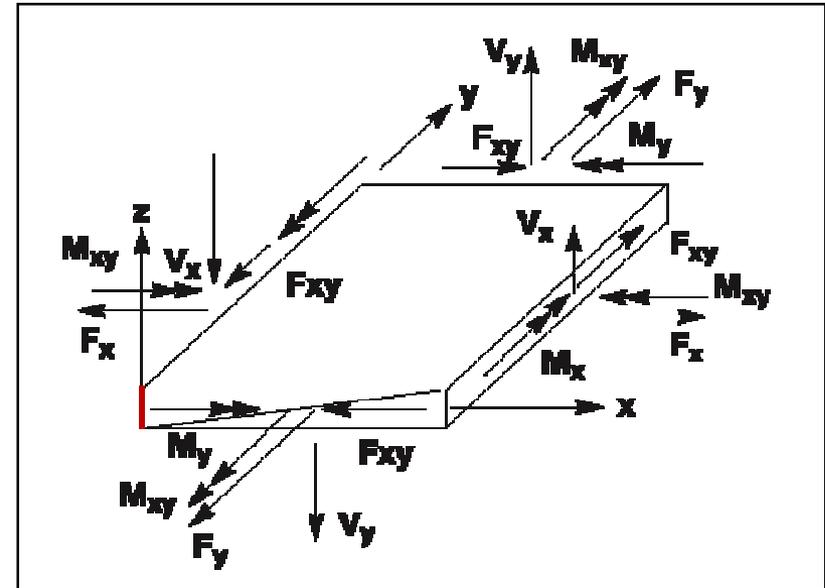
Classical Lamination Theory Forces In Plate Elements

MSC Software

SYMMETRY IN COMPOSITES

- **[B] is zero for symmetric laminator symmetry that occurs if for each lamina above the midplane, there is an identical ply (in properties and orientation) located at the same distance below the midplane.**

- **In MSC NASTRAN, $[G_4]$ is similar to [B].**

- **Examples of symmetric laminates**
  - +45/0/ +45
  - +45/ +45/ -45/ -45/ -45/ -45/ +45/ +45/
  - +45/ -45/ +45/ -45/ -45/ +45/ -45/ +45/

MSC Software

FINITE ELEMENT ANALYSIS OF COMPOSITE MATERIAL STRUCTURES

- **2-D analysis using lamination theory is found to give good results where the laminate is thin relative to its length.**

- **Otherwise, a full 3-D anisotropic material analysis is desirable.**

- **3-D analysis is also needed near free edges.**

- **3-D analysis uses HEXA elements to represent either single lamina or sets of lamina**

SAMPLE LAMINATE DEFINITION

```
CQUAD4,  101, 1,  A, B, C, D,  5
PCOMP, 1, , ,5000., STRN, , , ,+
+, 2,.003, 0.,YES, 1,.005, 45.,YES,+
+, 1,.005, 0.,YES, 1,.005,-45.,YES,+
+, 3,.007,90.,YES
MAT8, 1, 1.+7, 1.+7, .05, 1.+6, 1.+6, 1.+6, ,+
+, , , , .007, .006, .007, .006, .001
MAT8, 2, 2.+7, 2.+6, .35, 1.+6, 1.+6, 1.+6, ,+
+, , , , .007, .006, .007, .006, .001
MAT8, 3, 8.+6, 8.+6, .05, 7.+5, 7.+5, 1.+6, ,+
+, , , , .006, .005, .006, .005, .001
CORD2R, 5, ,0., 0., 0.,   0., -1., 0.,+CORD
+CORD,  0., 0., 1.
```


ply material
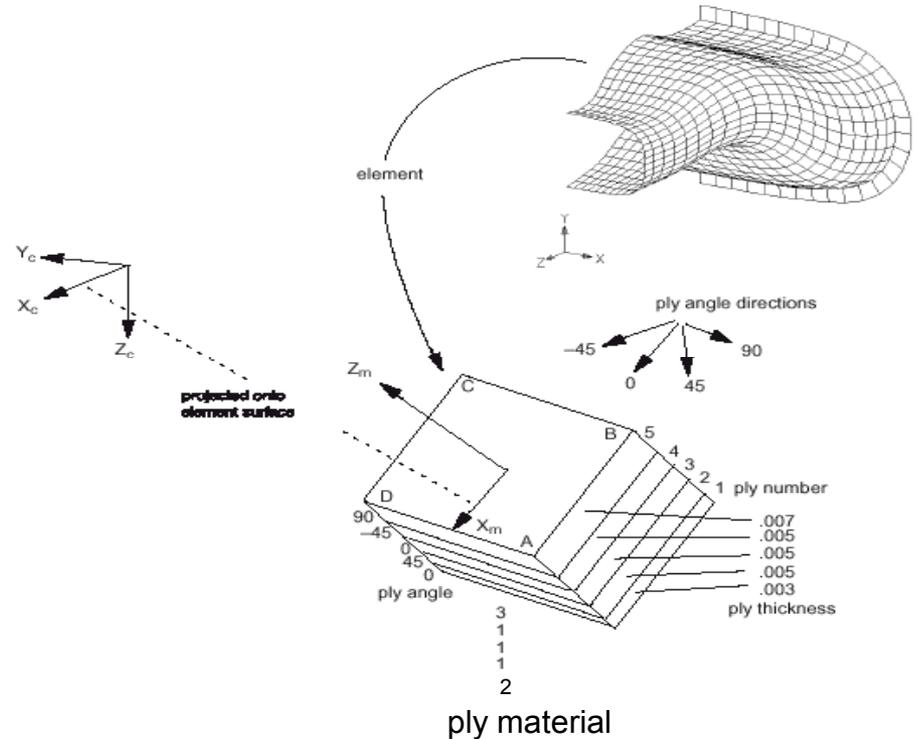
SPECIFICATION OF REFERENCE DIRECTION

- **Specify angle theta in element connection entries**

- **Provision to specify coordinate system ID in theta field**
  - The X axis is projected onto the element to define the direction of the X axis of the element material coordinate system.
  - The Z axis of the material coordinate system is defined by the element coordinate system Z axis (in other words, by the grid order in the element).

ELEMENT INPUT – CQUAD4

- **Defines a quadrilateral plate element (QUAD4) of the structural model.**
- **This is an isoparametric membrane-bending element.**
- **Format and Example:**

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	q	ZOFFS	
CQUAD4	111	203	31	74	75	32	2.6	0.3	ABC
		TFLAG	T1	T2	T3	T4			
+BC			1.77	2.04	2.09	1.80			
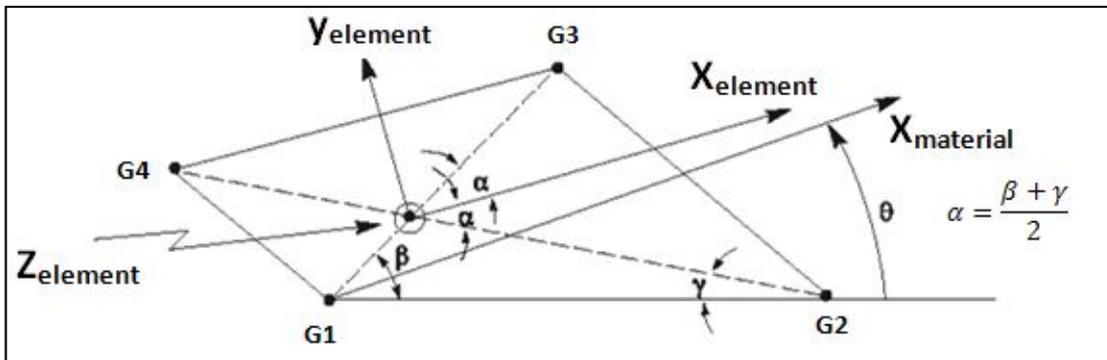
.bdf file extract

```
CQUAD4, 111,203,31,74,75,32,2.6,0.3,ABC
+BC,,,1.77,2.04,2.09,1.80,,,,
```

$$\alpha = \frac{\beta + \gamma}{2}$$

PCOMP BULK DATA ENTRY

- **Defines the composite layout**

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
PCOMP	1			5000.0	HILL	0.0			
	MID1	T1	THETA1	SOUT1	MID2	T2	THETA2	SOUT2	
	1	0.0054	0.0	YES	1	0.0054	45.0	YES	
	MID3	T3	THETA3	SOUT3	etc.				
	1	0.0054	90.0						

- Z0 is composite offset
    - Use default = -(composite thickness)/2
- NSM is nonstructural mass
- SB is allowable interlaminar shear stress
    - Put as Bonding Shear Stress in Patran 2D Orthotropic Material (page 2-6)
    - Required for failure indices
- FT is the ply failure theorem
    - Required for failure indices
- TREF is reference temperature
    - Overrides TREFs on ply MAT8s

- GE is element damping
    - Overrides GE on ply MAT8s
- LAM is layup options
- MIDi is ply material ID
    - MAT8 ID
- Ti is ply thickness
- THETAi is ply angle
- SOUTi is data recovery option

MSC Software®

PCOMP BULK DATA ENTRY (cont.)

- **The example composite below is an 8 ply layup, symmetric about it's centerline, with an equal number of plies in each of the 0, +45, 90 degree directions.**

.bdf file extract

```
PCOMP, 1,,, 5000., HILL
, 1, .0054,   0., YES
, 1, .0054,  45., YES
, 1, .0054, -45., YES
, 1, .0054,  90., YES
, 1, .0054,  90., YES
, 1, .0054, -45., YES
, 1, .0054,  45., YES
, 1, .0054,   0., YES
```

MSC Software

MATERIAL INPUT – MAT8

- **Defines the ply orthotropic properties**
  - Elastic properties are E1, E2, NU12, G12, G1Z, G2Z.
  - Allowables are Xt, Xc, Yt, Yc, S.
  - Use STRN=1.0 if allowables are in units of strain.
  - F12 is for the Tsai-Wu failure theorem.
  - Thermal coefficients of expansion are A1 and A2.
  - The MAT8 TREF reference temperature is not used since it is overridden by the PCOMP TREF.
  - Density is RHO.
  - The MAT8 GE structural damping is not used since it is overridden by the PCOMP GE.
- **The example below is typical for a graphite/epoxy tape:**

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	
MAT8	1	20.+6	2.+6	0.35	1.6+6	1.6+6	1.6+6	1.4-4	
	A1	A2	TREF	Xt	Xc	Yt	Yc	S	
	-2.4-7	4.5-6		1.3+5	1.2+5	1.1 +4	1.2+4	1.25+4	
	GE	F12	STRN						

.bdf file extract

```
mat8, 1, 20.+6, 2.+6, 0.35, 1.0+6, 1.0+6, 1.0+6, 1.4-4,+
+, -2.4-7, 4.5-6,, 1.3+5, 1.2+5, 1.1+4, 1.2+4, 1.25+4
```

MSC Software

MATERIAL INPUT – MAT2

- **Defines the material property for a 2-D orthotropic material**

- **The MAT2 (along with equivalent PSHELL) represents the equivalent homogeneous properties of the composite based on all the individual layers.**

- **Format and Example:**

1	2	3	4	5	6	7	8	9	10
MAT2	MID	G11	G12	G13	G22	G23	G33	RHO	
MAT2	1001	9.7318E+06	2.1133E+06	6.4575E-11	9.7318E+06	1.6562E-09	2.4046E+06	0.0000E+00	
	A1	A2	A3	TREF	GE	ST	SC	SS	
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	
	MCSID								
	0								

.bdf file extract

```
mat2, 1001, 9.7318E+06, 2.1133E+06, 6.4575E-11, 9.7318E+06, 1.6562E-09, 2.4046E+06, 0.0,+
+,0.0,0.0,0.0,0.0, 0.0, 0.0, 0.0, 0.0, 0.0
+,0
```

MSC Software

PROPERTY INPUT - PSHELL

- **Defines the membrane, bending, transverse shear, and coupling properties of thin shell elements**

- **Equivalent PSHELL have different MID for membrane, bending, transverse shear et al.**

- **Format and Example:**

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	12/T**3	MID3	TS/T	NSM	
PSHELL	1	1001	3.2400E-02	2001	1.0000E+00	3001	1.0000E+00	0.0000E+00	
	Z1	Z2	MID4						
	0.0000E+00	0.0000E+00	4001						

.bdf file extract

```
PSHELL,1, 1001, 3.2400E-02, 2001, 1.0000E+00, 3001, 1.0000E+00, 0.0000E+00,+
+, 0.0000E+00, 0.0000E+00, 0.0000E+00,,,,,,
```

MSC Software®

MSC NASTRAN COMPOSITE THEORY

- **PCOMP is a composite preprocessor**
  - PCOMP uses an equivalent shell
  - Writes equivalent PSHELL/MAT2 that are used internally

- **PSHELL/MAT2 are viewed with:**
  - To f06:
    - MSC NASTRAN SYSTEM (361)=1 or
    - MSC NASTRAN PRTPCOMP=1
  - ECHO=PUNCH writes to the .pch file

MSC Software
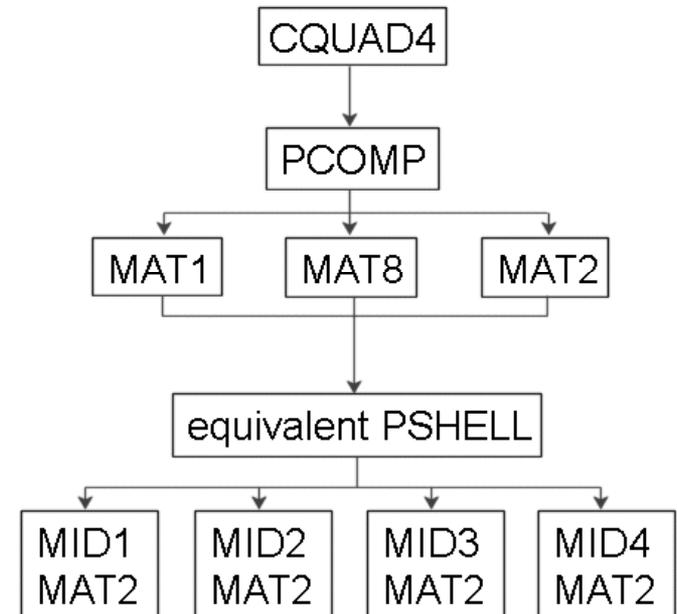
MSC NASTRAN COMPOSITE THEORY (Cont.)

```
*** USER INFORMATION MESSAGE 4379 (IFP6CD)
    THE USER SUPPLIED PCOMP BULK DATA CARDS ARE REPLACED BY THE FOLLOWING PSHELL AND MAT2 CARDS.
    WARNING, MAT2 RECORDS WITH MID GREATER THAN 400000000 USE A SPECIAL FORMAT FOR PCOMPS.
    REFER TO REMARK 13 OF THE MAT2 DESCRIPTION IN THE MSC.NASTRAN QUICK REFERENCE GUIDE.
 PSHELL             1   100000001  3.2400E-02   200000001  1.0000E+00    300000001  1.0000E+00  0.0000E+00
          -1.6200E-02  1.6200E-02   400000001
 MAT2      100000001  9.7318E+06  2.1133E+06  6.4575E-11  9.7318E+06  1.6562E-09  2.4046E+06  0.0000E+00
           0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
                   0
 MAT2      200000001  1.5574E+07  1.3330E+06  5.0620E+05  5.4501E+06  5.0620E+05  1.6243E+06  0.0000E+00
           0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
                   0
 MAT2      300000001  8.8986E+05  0.0000E+00  0.0000E+00  6.7546E+05  0.0000E+00  0.0000E+00  0.0000E+00
           0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
                   0
 MAT2      400000001  2.7210E+05  2.3410E+05  1.2655E+05 -7.4030E+05  1.2655E+05  2.3410E+05  0.0000E+00
           0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
                   0
```

.f06 file extract

OUTPUT

- **Stresses in individual lamina including approximate interlaminar shear stresses**

- **Element strains**

- **Element forces**

- **Failure Index**

PLY STRESS AND STRAIN OUTPUT

- **To obtain ply stresses and strains, use the following Case Control Commands respectively:**
  - STRESS = ALL
  - STRAIN = ALL

- **To obtain Failure Index Table, allowables (X_t, X_c, Y_t, Y_c, S) must be supplied in MAT8 Bulk Data, and S_b can be supplied in PCOMP Bulk Data entry.**

- **Interlaminar shear stresses are output between the lamina.**

- **Individual lamina stresses can be sorted (use NUMOUT1 and BIGER1 parameters).**

- **Failure Index table can be sorted (use NUMOUT2 and BIGER2 parameters).**

- **Available in SOL101 and 103 without alters.**

MSC Software

REQUESTING PLY STRESSES AND STRAINS IN MSC NASTRAN

- **For ply stresses, use STRESS case control command.**

- **For ply strains, use STRAIN case control command.**

.bdf file extract

```
SOL 101
CEND
TITLE = Composite Workshop Chapter 2 - Sample
Composite Input
    SPC = 1
    LOAD = 1
    DISP = ALL
    STRESS =ALL
$
BEGIN BULK
 .
```

REQUESTING PLY STRESSES AND STRAINS IN MSC NASTRAN (Cont.)

- **The given case control produces the following ply stresses:**

.f06 file extract

```
   S T R E S S E S   I N   L A Y E R E D   C O M P O S I T E   E L E M E N T S   ( Q U A D 4 )
    ELEMENT   PLY   STRESSES IN FIBRE AND MATRIX DIRECTIONS      INTER-LAMINAR  STRESSES   PRINCIPAL STRESSES (ZERO SHEAR)      MAX
      ID      ID      NORMAL-1      NORMAL-2      SHEAR-12     SHEAR XZ-MAT   SHEAR YZ-MAT   ANGLE     MAJOR        MINOR        SHEAR
  0       1    1   -2.32938E+04   2.66174E+04   1.78380E+04      0.0            0.0         72.22   3.23372E+04  -2.90136E+04  3.06754E+04
  0       1    2    3.00912E+05   4.77975E+03   1.53469E+04      0.0            0.0          2.96   3.01705E+05   3.98654E+03  1.48859E+05
  0       1    3   -4.76318E+04   2.82568E+04  -1.53469E+04      0.0            0.0        -78.99   3.12429E+04  -5.06179E+04  4.09304E+04
  0       1    4    2.76574E+05   6.41909E+03  -1.78380E+04      0.0            0.0         -3.76   2.77747E+05   5.24636E+03  1.36250E+05
```

MSC Software

ELEMENT FORCE AND STRAIN OUTPUT

- **Element force output and strain output available.**

- **When PARAM,NOCOMPS,-1 bulk data parameter is used along a STRAIN=ALL case control command, the following output is produced:**

.f06 file extract

```
       S T R A I N 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      STRAIN                STRAINS IN ELEMENT COORD SYSTEM              PRINCIPAL  STRAINS (ZERO SHEAR)
      ID.       CURVATURE         NORMAL-X        NORMAL-Y        SHEAR-XY      ANGLE        MAJOR           MINOR         VON MISES
0      1    0.0               1.229821E-02    9.603290E-03    2.730186E-02    42.1813    2.466802E-02    -2.766525E-03    1.744082E-02
           -1.000000E+00      0.0             0.0             0.0              0.0       0.0              0.0             0.0
0      2    0.0               1.129405E-02   -1.183270E-02    9.950880E-03    11.6405    1.231902E-02    -1.285768E-02    1.453689E-02
           -1.000000E+00      0.0             0.0             0.0              0.0       0.0              0.0             0.0
0      3    0.0              -3.134008E-03    6.225147E-03    8.147041E-03    69.4804    7.749762E-03    -4.658625E-03    7.237704E-03
           -1.000000E+00      0.0             0.0             0.0              0.0       0.0              0.0             0.0
0      4    0.0               4.397135E-03   -1.491786E-03    1.420404E-02    33.7407    9.140880E-03    -6.235531E-03    8.930243E-03
           -1.000000E+00      0.0             0.0             0.0              0.0       0.0              0.0             0.0
```

COMPOSITE FAILURE INDICES

- Composite failures are checked at the lamina level.

- Failure index of a lamina checks whether the state of stress can cause a failure.

- If the failure index of the element is less than or equal to 1.0, stresses in all laminas are within or on the respective failure envelopes.

- If the failure index is greater than 1.0 in at least one lamina, then the element is assumed to fail.

ALLOWABLE STRESSES

X_t =   Allowable stress or strain in tension in longitudinal direction (or 1-direction or fiber direction)

X_c =   Allowable stress or strain in compression in longitudinal direction (positive sign will be used for X_c)

Y_t =   Allowable stress or strain in tension in transverse direction (or 2-direction or matrix direction)

Y_c =   Allowable stress or strain in compression in transverse direction (positive sign will be used for Y_c)

S =   Allowable stress in shear (positive or negative shear has the same allowable)

S_b =   Allowable shear stress of bonding material (allowable inter-laminar shear stress)

X_t , X_c , Y_t , Y_c , and S are supplied in MAT8 Bulk Data entry.

S_b can be supplied in PCOMP Bulk Data entry.

PLY FAILURE THEOREMS

- **MSC Nastran failure theorems are calculated at the ply level.**

- **If composite failure is assumed with any ply failure**
  - "first ply failure" technique is being used.
  - This is a conservative approach suitable for limit loads.

- **For residual strength or ultimate load analysis**
  - If the ply failed in the fiber direction, the failed ply may be removed and the analysis run again.
  - If failure is in the matrix direction, higher ultimate load allowables should be used in that direction.

- **Failure indices (FI) greater than 1.0 indicate a failed ply.**
  - However, if the failure index is not a linear function (Hill, Hoffman, Tsai-Wu), then its value cannot be used as a linear measure of residual strength.

- **Strength ratios (SR) less than 1.0 indicate a failed ply.**
  - SR can be used as a linear indication of the amount of residual strength.

- **Ply failure analysis is useful**
  - to easily calculate failures for any layup.
  - To decrease composite failure testing if a different composite material system is used.

LAMINATE FAILURE THEOREMS

- **Laminate failure analysis uses the composite element output instead of ply output.**

- **Element load failure theorems**
  - Set PARAM,NOCOMPS,-1 which will disable ply stress or strain output
    - Note: **In this case the homogeneous stresses are WRONG** (see remark in QRG). One should just look for the STRAIN values, as they are correct.
  - Either set PARAM,OMID,YES which prints or punches element loads, "equivalent stresses", and strains in the material coordinate system (not supported by .op2 or .xdb)
  - Or use equivalent PSHELL/MAT2 combinations from ECHO=PUNCH case control command and
    - MCSID coordinate system on the MAT2s
    - PARAM,CURV,1 postprocessing

- **Laminate failure theory is more accurate since each actual layup is tested.**
  - However, each layup used must be tested.
  - Changing material systems requires retesting.

- **Can be used in all solution sequences.**

REQUESTING PLY FAILURE INDICES IN MSC NASTRAN

- **Allowables must be entered on the MAT8 bulk data entry**
  - Xt, Xc, Yt, Yc, S, F12

- **Sb and FT must be entered on the PCOMP bulk data entry**
  - Sb is interlaminar stress allowable
  - FT is the failure theorem to be used

- **For stress based failure theorems, the STRESS case control must be used.**

- **For strain based failure theorems, both, STRESS and STRAIN case control must be used.**

STRENGTH RATIO OUTPUT FOR LAMINATED COMPOSITES

- **For laminated composites, Strength Ratio (SR) is a direct failure indicator compared to Failure Index (FI) which indicates only if failure occurs. Generally, Strength Ratio is defined as:**

  Strength Ratio (SR) = Allowable Stress / Calculated Stress

  For example a SR = 1.2 indicates that the applied loads can be increased by 20% before failure occurs. A FI = 0.8 indicates that failure has not occurred and does not indicate 20% safety margin. Therefore the SR is a much more practical design indicator for analysis and strength-criteria based design.

- **A parameter (PARAM,SRCOMPS,YES/NO) requests Strength Ratio (SR) output. SR Output requires specification of a failure theory and allowable stress/strain values. Note that SRs will be computed for plies with an output request (PCOMP-SOUTi field).**

MSC Software

FAILURE THEORIES FOR COMPOSITE MATERIALS

- **HILL'S THEORY**

- **HOFFMAN'S THEORY**

- **TSAI-WU THEORY**

- **INTERLAMINAR SHEAR**

MSC Software®

HILL'S THEORY

$$\frac{\sigma_1^{\,2}}{X^{\,2}} + \frac{\sigma_2^{\,2}}{Y^{\,2}} - \frac{\sigma_1 \sigma_2}{X^{\,2}} + \frac{\sigma_{12}^{\,2}}{S^{\,2}} = 1$$

where　　　　$X = X_t$ **if** σ_1 **is tensile**

$X = X_c$ **if** σ_1 **is compressive**

$Y = Y_t$ **if** σ_2 **is tensile**

$Y = Y_c$ **if** σ_2 **is compressive**

- **For the product term, $X = X_t$ if σ_1 and σ_2 are of the same sign;**

　　　　　　　　$X = X_c$ otherwise

- **Basically, the equation represents a failure envelope in the stress space.**

- **If the state of stress in the orthotropic lamina (σ_1 , σ_2 , σ_{12}) is such that the stress point is within or on the envelope, the lamina is said to be "safe." If the point is outside, the lamina is said to have "failed."**

HOFFMAN'S THEORY

- **Hill's theory does not take into account the differing tensile and compressive strengths in the fiber and matrix directions.**

$$\left(\frac{1}{X_t} - \frac{1}{X_c}\right)\sigma_1 + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)\sigma_2 + \frac{\sigma_1^2}{X_t X_c} + \frac{\sigma_2^2}{Y_t Y_c} - \frac{\sigma_1 \sigma_2}{X_t X_c} + \frac{\sigma_{12}^2}{S^2} = 1$$

- **This equation can be thought of as having been derived from Hill's theory by adding linear terms to account for differing strengths in tension and compression.**

MSC Software

HOFFMAN'S THEORY (Cont.)

- **Is an ellipsoid in $\sigma_1, \sigma_2, \tau_{12}$ space:**
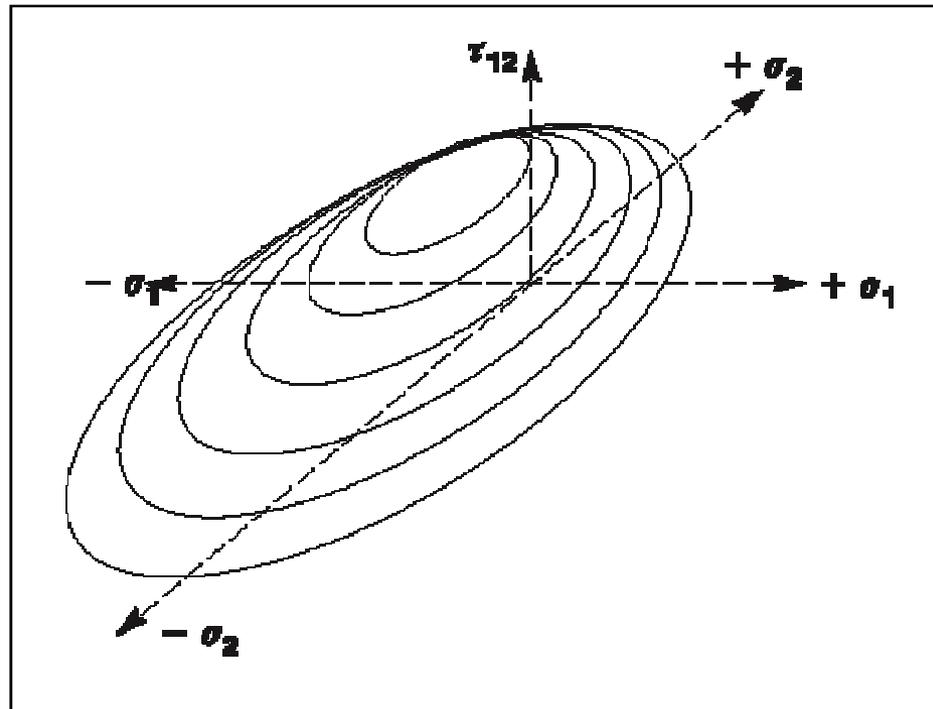
TENSOR POLYNOMIAL THEORY (TSAI-WU THEORY)

- **The theory of strength for anisotropic materials, proposed by Tsai and Wu, specialized to the case of an orthotropic lamina in a general state of plane stress is**

$$F_1\sigma_1 + F_2\sigma_2 + F_{11}\sigma_1^2 + 2F_{12}\sigma_1\sigma_2 + F_{22}\sigma_2^2 + F_{66}\sigma_6^2 = 1$$

where

$$F_1 = \frac{1}{X_t} - \frac{1}{X_c}$$

$$F_2 = \frac{1}{Y_t} - \frac{1}{Y_c}$$

$$F_{11} = \frac{1}{X_t X_c} \qquad F_{22} = \frac{1}{Y_t Y_c} \qquad F_{66} = \frac{1}{S^2}$$

and F_{12} is to be determined experimentally.

MSC Software

TSAI-WU THEORY (Cont.)

- **The magnitude of F_{12} is constrained by the following inequality that is called the "stability criterion" associated with the theory**

$$F_{11}F_{22} - F_{12}^2 > 0$$

- **The need to satisfy the stability criterion together with the requirement that F_{12} be determined experimentally from a combined stress-state poses difficulties.**

- **In the absence of experimental value, Tsai recommends using:**

$$F_{12} = -\frac{1}{2}\sqrt{F_{11}F_{22}}$$

- **Geometrically, this condition ensures that the strength envelope is closed. That is, the shape of the envelope must be ellipsoidal rather than parabolic or hyperbolic ensuring that the material has finite strength in all directions.**

MSC Software

INTERLAMINAR SHEAR FAILURE INDEX

$$\frac{\tau_{1zi}}{S_b} \quad , \quad \frac{\tau_{2zi}}{S_b}$$

where

τ_{1zi} = **Shear stress between the i lamina and the i+1 lamina in the X direction of the element material coordinate system.**

τ_{2zi} = **Shear stress between the i lamina and the i+1 lamina in the Y direction of the element material and coordinate system.**

S_b = **Allowable interlaminar shear stress that is input on the PCOMP entry.**

CONCLUSION

- **MSC NASTRAN layered composite analysis capability**
  - Is user friendly
  - Is easy to use
  - Has simple input
  - Allows stresses in individual plies to be sorted and output
  - Provides failure index for individual plies
  - Supports solid composite plus additional advanced failure theories.

- **3D Composite is also available in MSC Nastran (see NAS113)**

- **Ply stress recovery is now (Nastran 2012+) available in transient and frequency response analysis!!!**

MSC Software®

REFERENCES

- *MSC.NASTRAN Reference Manual*, Sections 13.2.

- "Mechanics of Composite Materials," R.M. Jones; Scripta Book Co., Washington D.C., 1975.

- "Mechanics of Composite Materials," R.M. Christensen; John Wiley & Sons, New York, 1979.

- "Primer on Composite Materials Analysis," J.E. Ashton, J.C. Halph, P.H. Petit; Technomic Publishing Co., Inc., Stamford, Connecticut, 1969.

MSC Software®

WORKSHOP

- **Perform Workshop 3 for composite**

SECTION 5

LINEAR CONTACT AND GLUED CONTACT

MSC Software

CONTENTS

INTRODUCTION TO CONTACT ANALYSIS

MSC Software

WHAT IS CONTACT ANALYSIS?

- **Contact analysis is the analysis of contact bodies (deformable or rigid) interacting with each other**

- **Contact analysis types**
  - Touching Contact
  - Glued Contact
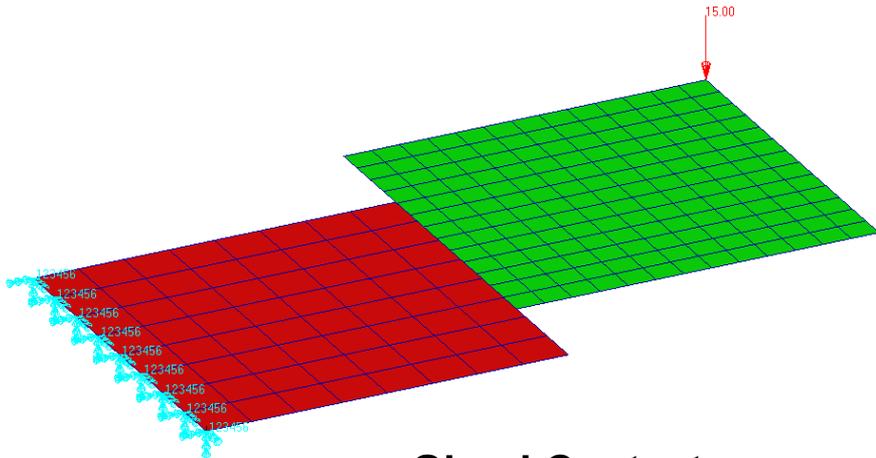
Glued Contact

Touching Contact

MSC Software
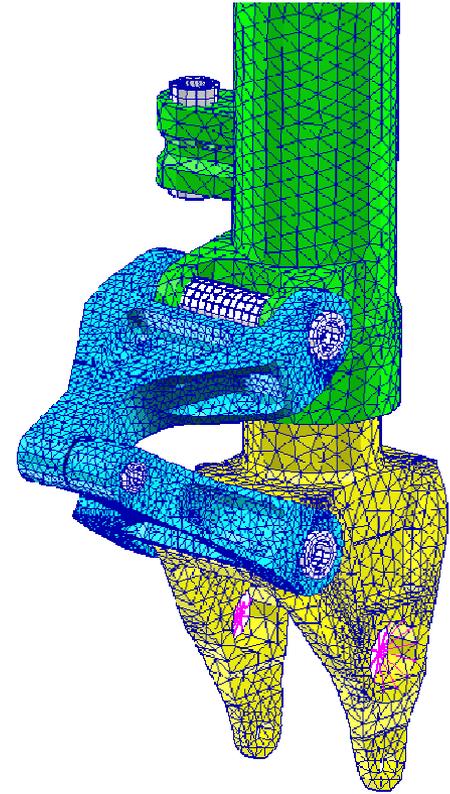
CONTACT ANALYSIS EXAMPLES

- **Solid-to-solid contact examples**
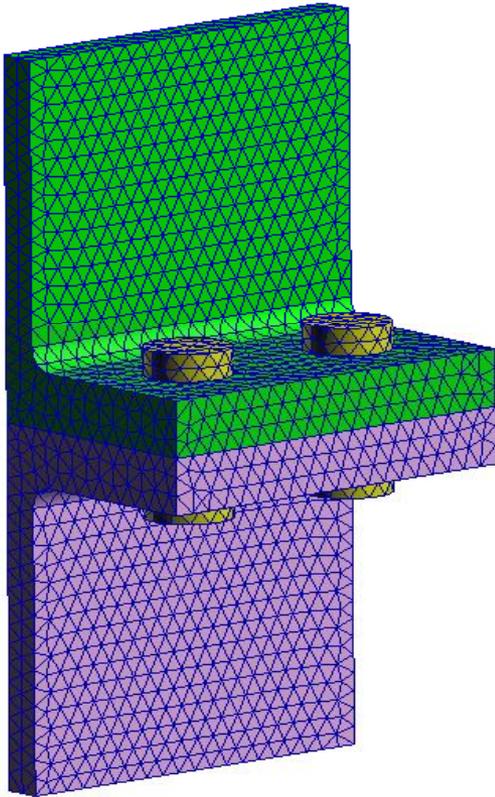
Preloaded Bolted Joint
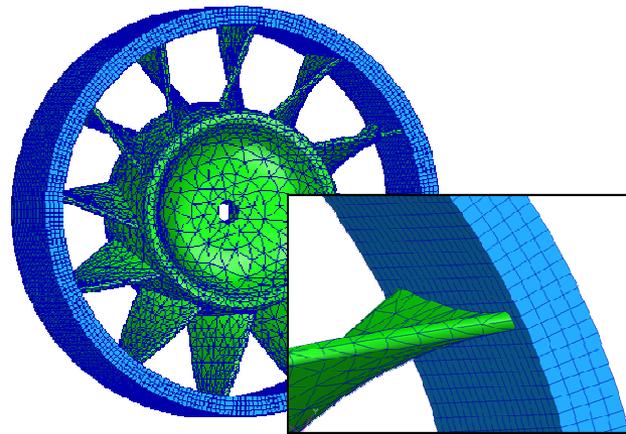
Glued Assembly

Lug-Clevis-Pin

MSC Software
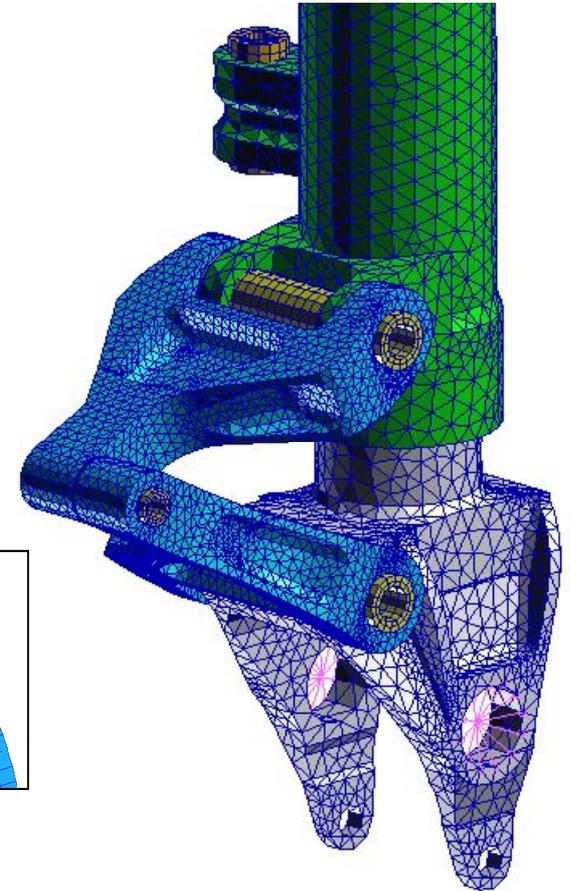
CONTACT ANALYSIS EXAMPLES (Cont.)

- **Shell-to-solid contact examples**

- Shell-to-shell contact examples

Face-to-face contact

Face to face contact

Edge-to-edge glued contact

MSC Software
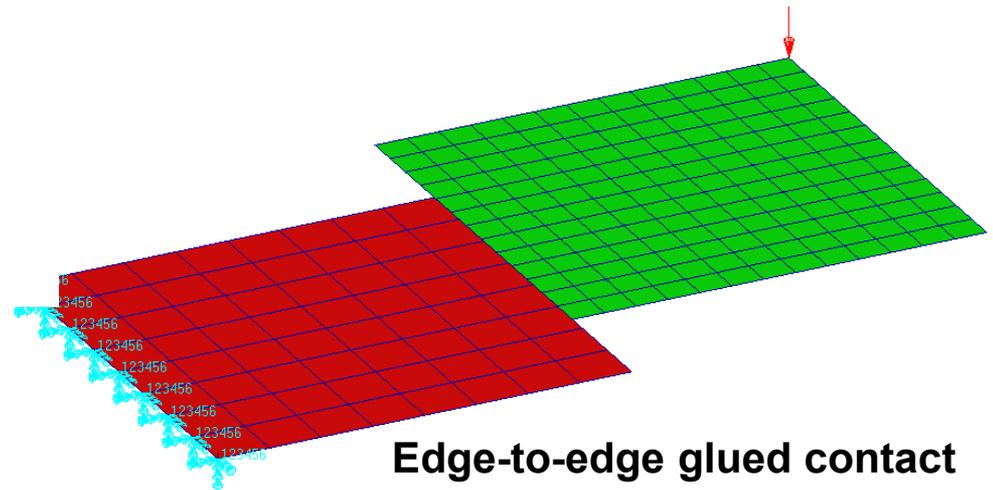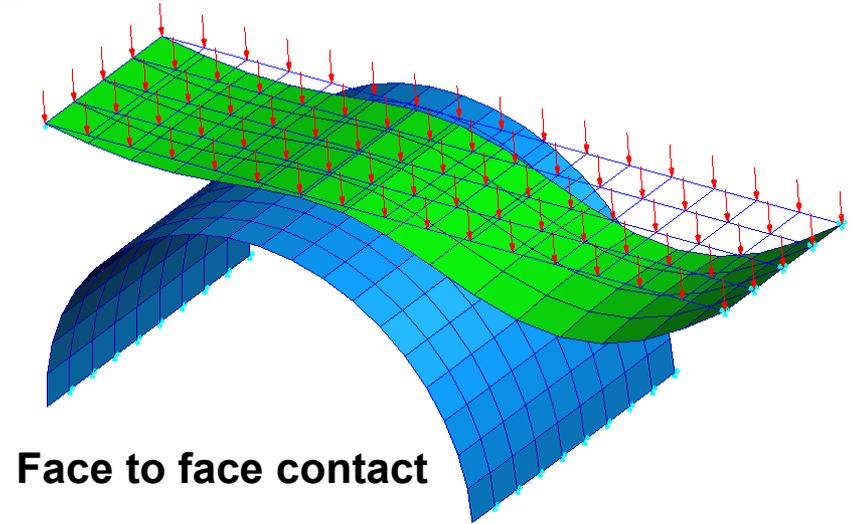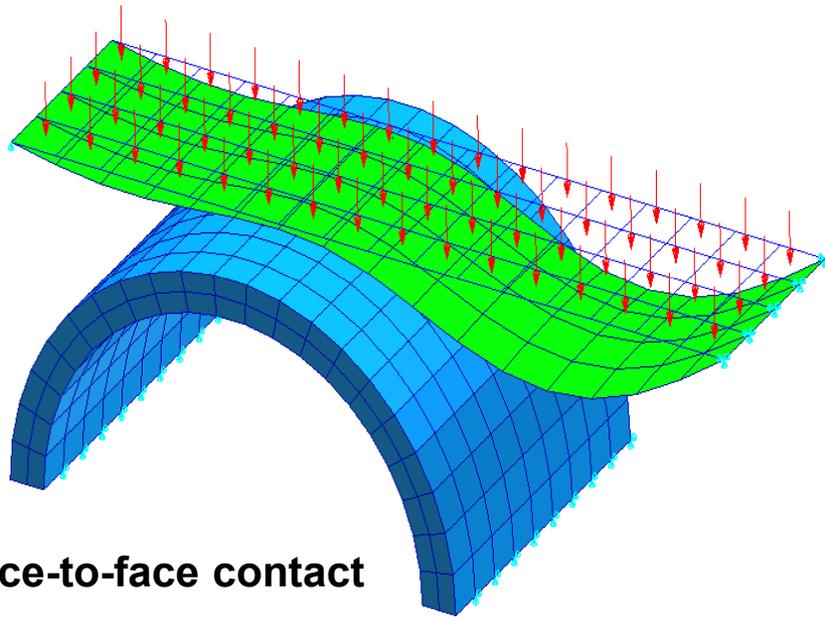
CASE STUDY 1

CASE STUDY 1

- **Two parts with different mesh densities**
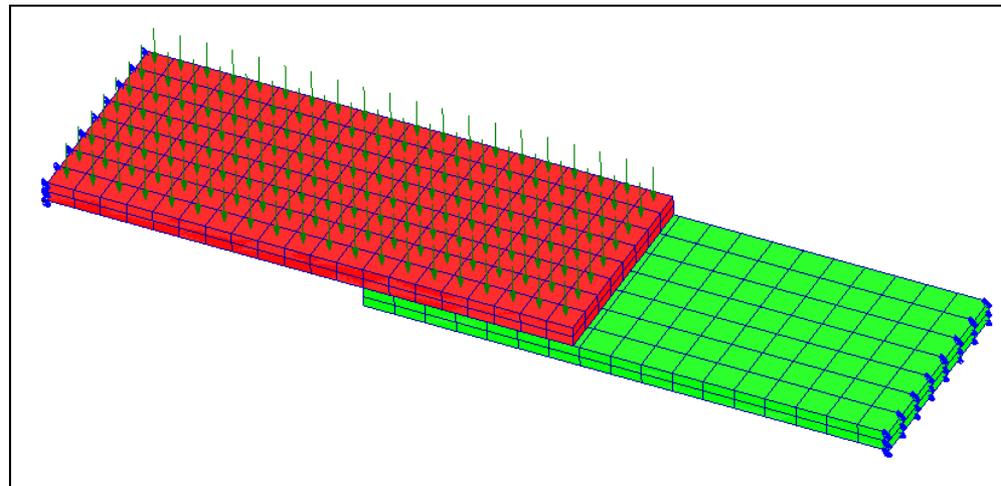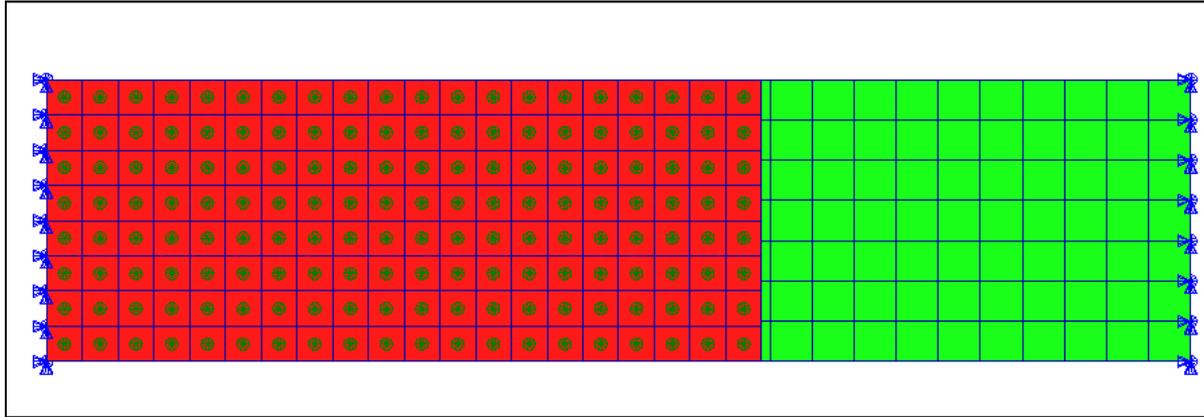
CASE STUDY 1 (Cont.)

- **Initial Pass: No Contact definition**

  – Parts don't see each other

  – Parts pass through each other

No Contact

- **Second Pass: Add Contact body definition**

  – Parts now contact each other

Contact added

MSC Software
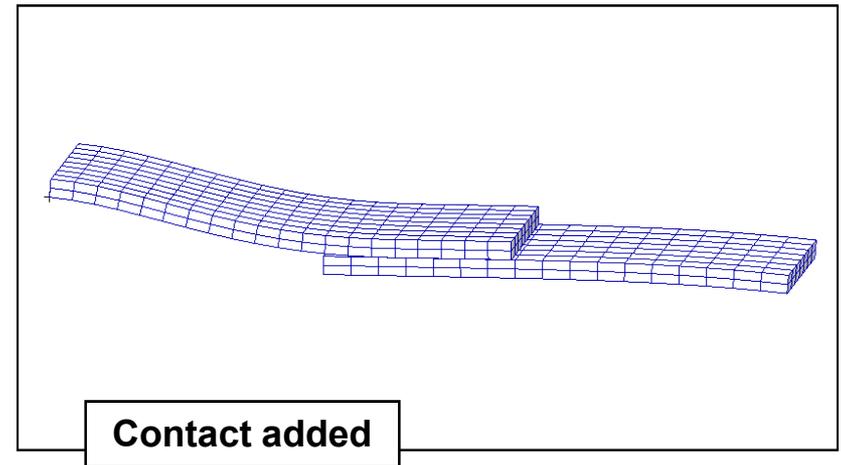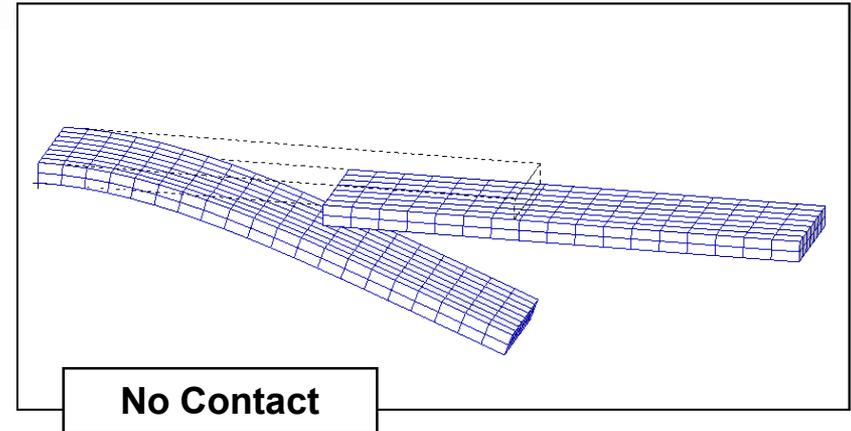
CONTACT BODY DEFINITION AND CONTACT BODY TYPES

- **Contact bodies can be deformable or rigid.**

- **Deformable bodies will be covered in this class.**

- **Rigid bodies will be covered in the Advanced Contact Analysis class.**

DEFINITION OF DEFORMABLE BODIES

- **A deformable body is a collection of finite elements**

- **A deformable body must contain elements of the same class:**

    - Linear shell   CTRIA3/CQUAD4

    - Quadratic shell   CTRIA6/CQUAD8

    - Linear solid   CHEXA/CPENTA/CTETRA

    - Quadratic solid   CHEXA/CPENTA/CTETRA

    - Beams   CBAR/CBEAM/CROD

DEFINITION OF DEFORMABLE BODIES (Cont.)

- **Each deformable body consists of one or more finite elements**

- **Nodes or elements must belong to no more than one deformable body**

- **A deformable body does not need to completely correspond with a physical body**

- **On the other hand be careful with a subset of elements**

Limited number of elements defined in contact body
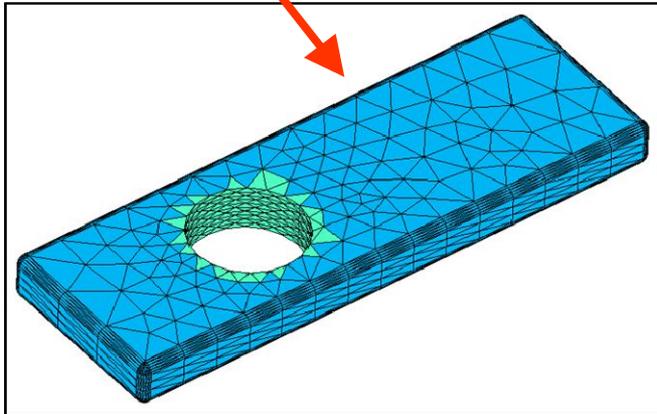
Local normal vector to the outer boundary may be completely wrong

MSC Software
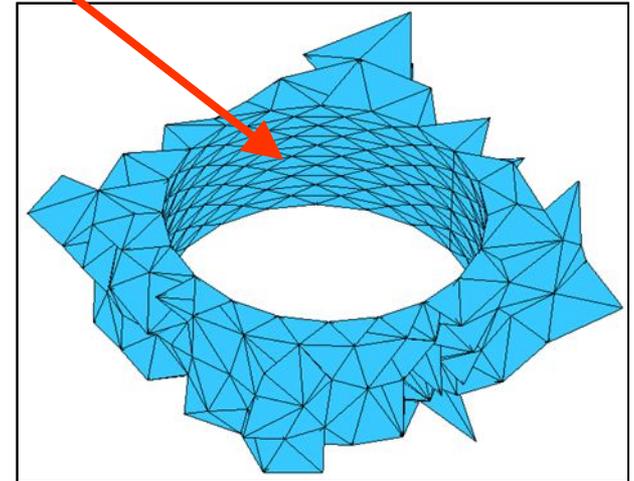
DEFINITION OF DEFORMABLE BODIES (Cont.)

- **All deformable bodies can come in contact with each other, including self-contact**

- **MSC Nastran automatically figures out the free faces as potential contact surfaces**

- **MSC Nastran also automatically accounts for shell thicknesses**

CREATING A DEFORMABLE BODY

- **From the Patran Loads/BCs Menu:**

1. Create / Contact / Element Uniform
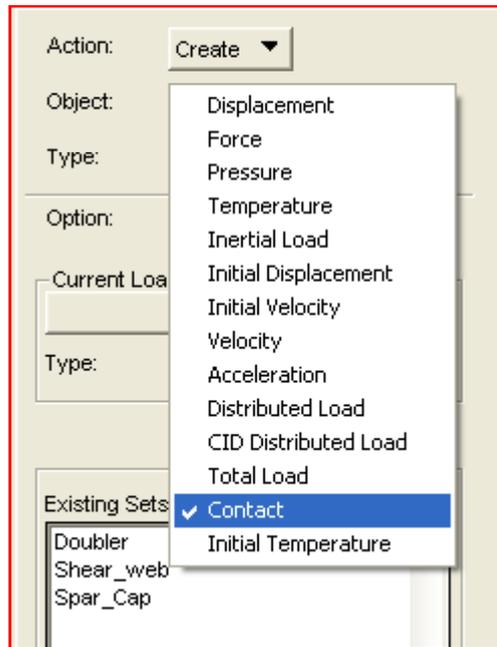
2. Select "Deformable Body"

MSC Software
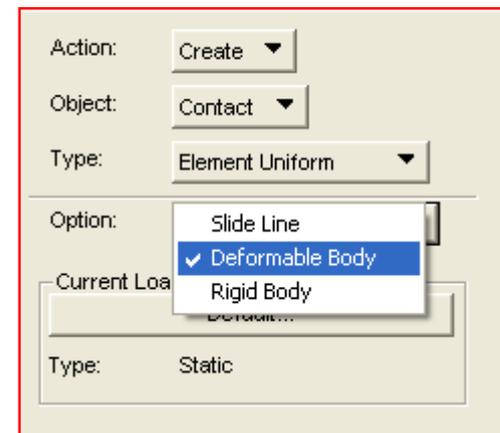
CREATING A DEFORMABLE BODY (Cont.)

- **On "Select Application Region" form:**
  3. **Select elements based on geometry, elements, or property** to create a deformable body.

(or)

(or)
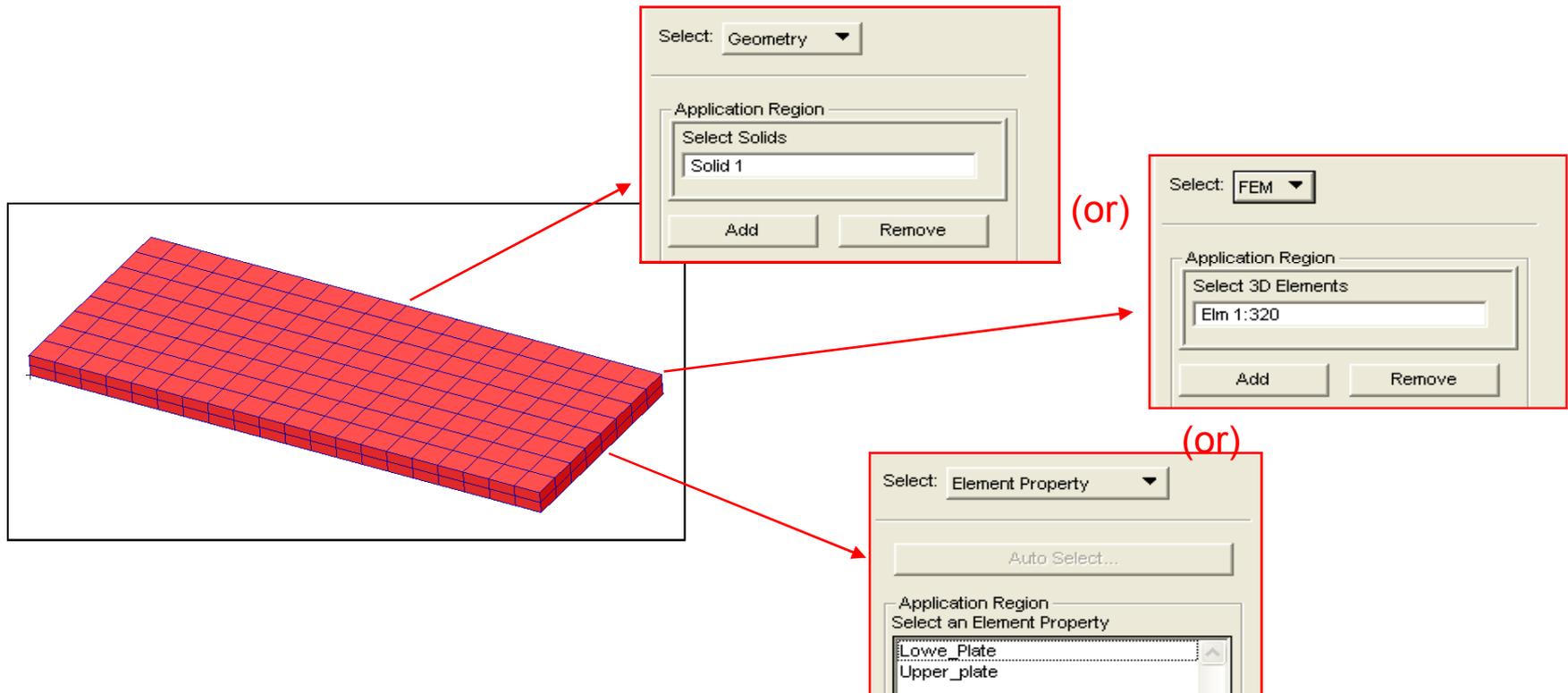
CREATING A DEFORMABLE BODY (Cont.)

- **Contact Body MSC Nastran entries:**
- **BCBODY – Flexible or Rigid Contact Body**

BCBODY	BID	DIM	BEHAV	BSID	ISTYP	FRIC	IDSPL	CONTROL	
	NLOAD	ANGVEL	DCOS1	DCOS2	DCOS3	VELRB1	VELRB2	VELRB3	
	"ADVANCE"	SANGLE	COPTB						
	"RIGID"	CGID	NENT	--- Rigid Body Name ---					
	"GROW"	GF1	GF2	GF3	TAB-GF1	TAB-GF2	TAB-GF3		

- **BSURF – Defines a contact body by Element IDs**

    (Referenced by BSID in BCBODY)

BSURF	ID	ELID1	ELID2	ELID3	ELID4	ELID5	ELID6	ELID7	

CREATING A DEFORMABLE BODY (Cont.)

- **Sample MSC Nastran input file:**

CREATING A DEFORMABLE BODY (Cont.)

- **Property-based MSC Nastran entry:**

- **BCPROP – Defines a contact body by Element Property**

  (Referenced by **BSID** in **BCBODY**)

1	2	3	4	5	6	7	8	9	10
BCPROP	ID	IP1	IP2	IP3	IP4	IP5	IP6	IP7	
	IP8	IP9	etc.						

Example:

BCPROP	1	101	201	301					

Other Deformable Body Methods for BSID :

BCBOX and BCMATL entry if BEHAV=DEFORM. (Integer > 0)

MSC Software

CREATING A DEFORMABLE BODY (Cont.)

- **Sample MSC Nastran input file:**


```
$
$
$ Deform Body Contact LBC set: left
BCBODY      1          3D          DEFORM      6          0
BCPROP      6          1
$
$
$ Deform Body Contact LBC set: right
BCBODY      2          3D          DEFORM      7          0
BCPROP      7          2
$
$
```

BCPROP
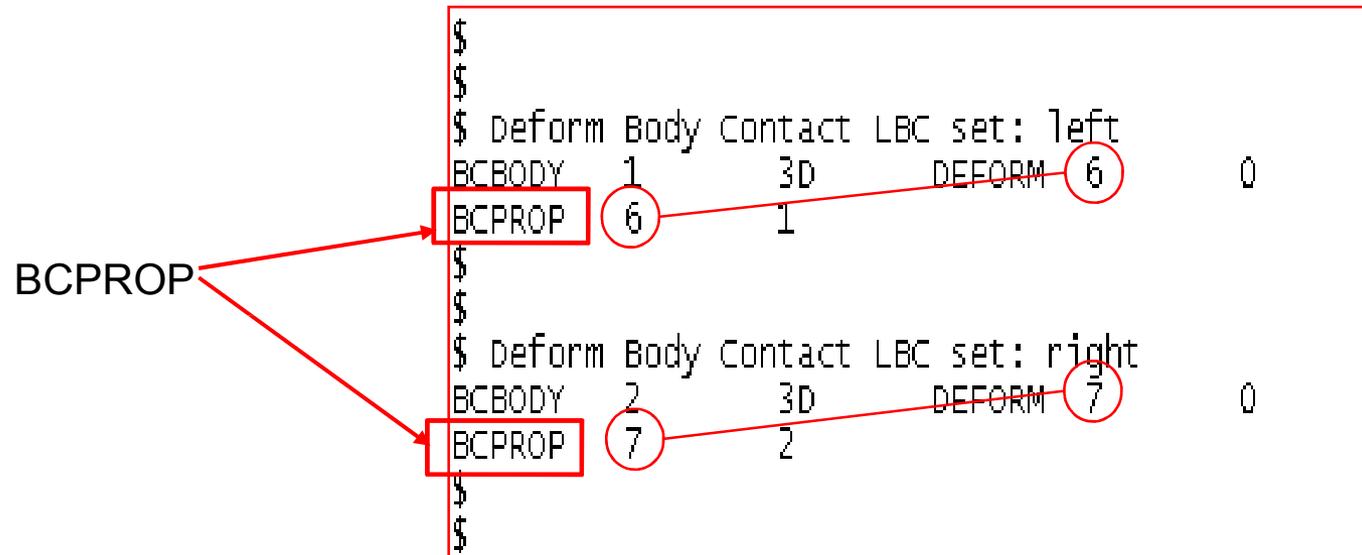
CHOOSING A SOLUTION SEQUENCE

- **Contact analysis is supported in both SOL 400 and 101**
  - Use SOL 400 for general nonlinear contact analysis
  - Use SOL 101 for "linear" contact analysis where
    - Contact is the only nonlinearity present
    - No material nonlinearity is present
    - No geometric nonlinearity is present
  - SOL 101 is a good starting point
  - But if the computed displacements are large, or material plasticity needs to be modeled, then SOL 400 should be used instead of SOL 101

SELECTING RESULTS OUTPUT FORMAT

- **The traditional MSC Nastran results are available in both XDB and MASTER/DBALL**

- **Contact analysis results are only available in MASTER/DBALL**

- **Do the following when setting up a contact analysis job**

  - Set SCR=NO and specify MSC NASTRAN SYSTEM(316)=19 to generate MASTER and DBALL files suitable for contact results post processing

  - Alternatively set SCR=POST

  - If restart is required, specify MSC NASTRAN SYSTEM(316)=7

REQUESTING FOR OUTPUT

- **Traditional MSC Nastran results such as displacement and stress are requested as usual in the Case Control Section**

- **To obtain contact results such as contact normal force and contact normal stress, specify the following in the Case Control Section**

    – BOUTPUT=ALL

MSC Software®

SOLID TO SHELL CONTACT: WORKSHOP 4

Performing Workshop 4
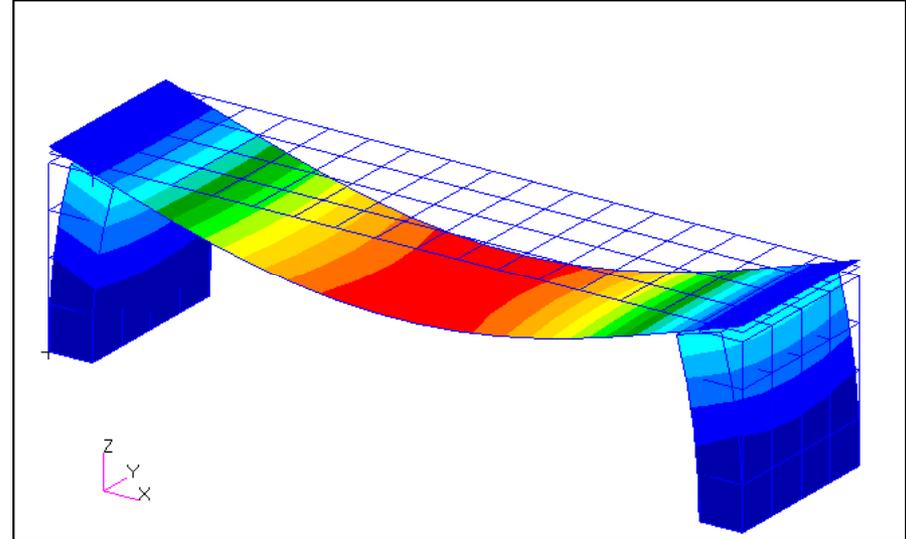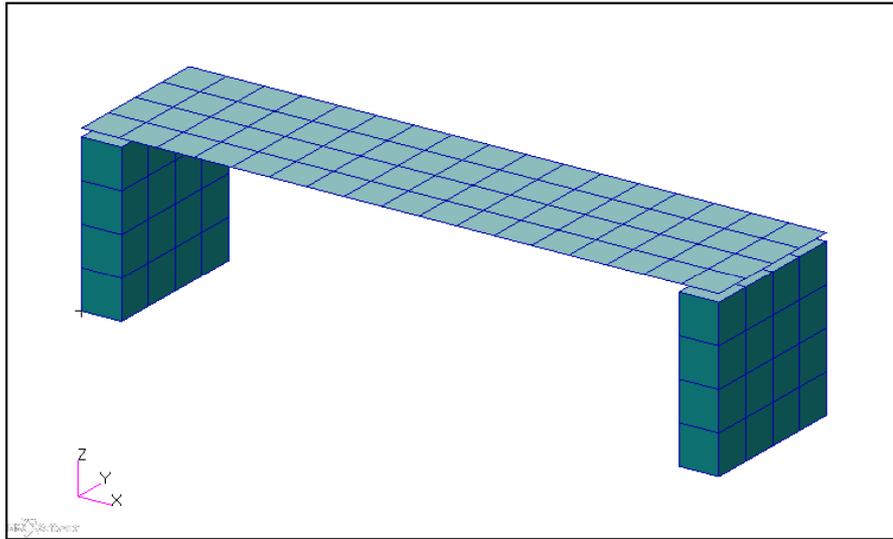
GLUED CONTACT

GLUED CONTACT

- **SOL 400 and 101 support the general glued contact capability**

  - Simulates a glued joint

  - Bodies don't have to be initially in contact. They can come in contact during the analysis and become glued.

  - After being glued together, bodies can separate again or stay glued based on user-specified criteria.

  - Just like touching contact, the general glued contact utilizes the nonlinear solver which is an incremental and iterative process.

GLUED CONTACT (Cont.)

- **Permanent Glued Contact is a special case of glued contact**

  - Designed to help users quickly assemble components with dissimilar meshes

  - Available in SOL 101, 103, 105, 107, 108, 109, 110, 111, 112, and 200.

  - A linear solution. Permanent contact constraint MPC equations are used. No nonlinear increments or iterations involved.

PERMANENT GLUED CONTACT EXAMPLES
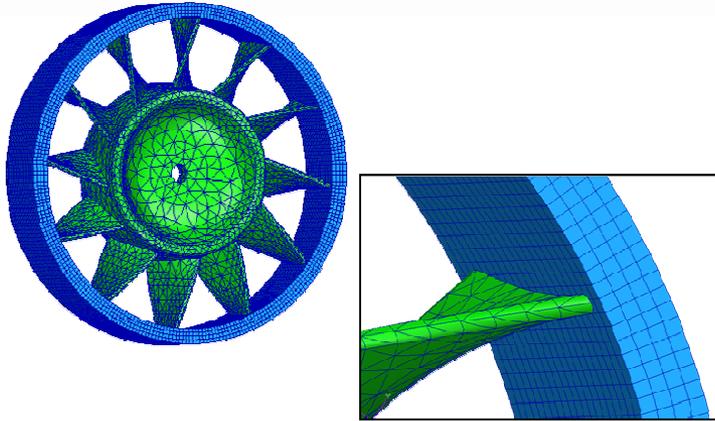
Solid Face-to-Face
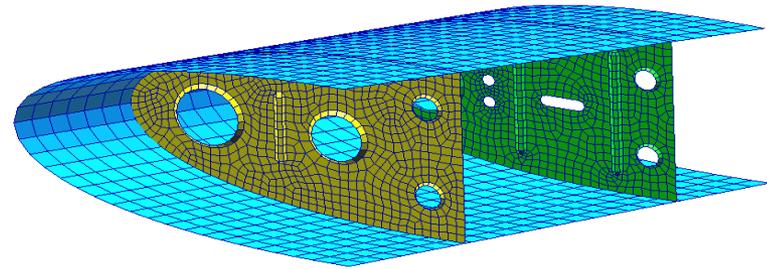
Shell Edge-to-Face
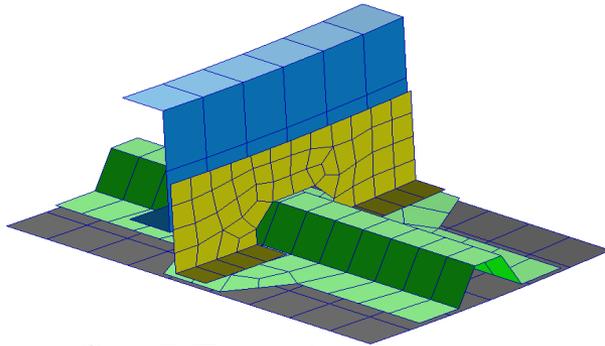
Shell Face-to-Face
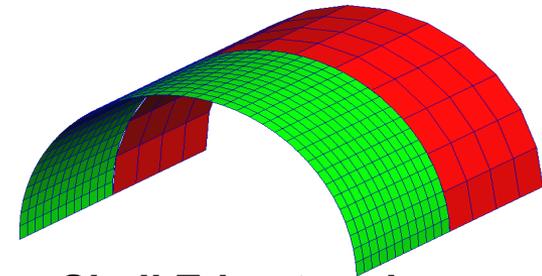
Shell Edge-to-edge

SETTING UP GLUED CONTACT

- **BCTABLE – IGLUE parameter**

1	2	3	4	5	6	7	8	9	10
BCTABLE	ID	IDSLAVE	IDMAST	NGROUP	COPTS	COPTM			
	"SLAVE"	IDSLA1	ERROR	FNTOL	FRIC	CINTERF	IGLUE		
		ISEARCH	ICOORD	JGLUE	TOLID	DQNEAR	DISTID		
		"FBSH"	FRLIM	BIAS	SLIDE	HARDS	COPTS1	COPTM1	
		"BKGL"	BGST	BGSN	BGM	BGN			
		"HHHB"	HCT	HCV	HNC	BNC	EMISS	HBL	
		FK	EXP	METHOD	ADAPT	THICK	THICKOF	PENV	
		FACT	TSTART	TEND	MAXPAR	PENCHK	ESE	VSE	

SETTING UP GLUED CONTACT (Cont.)

- **BCTABLE – IGLUE parameter**

  - **0** – no gluing

  - **1** - Activates the glue option. In the glue option, all degrees-of- freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. The node will be projected onto the contact body.

  - **2** - Activates a special glue option to insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. To maintain an initial gap, ERROR should be set to a value slightly larger than the physical gap.

  - **3** - Insures **full moment carrying glue** when shells contact. The node will be projected onto the contacted body.

  - **4** - Insures **full moment carrying glue** when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body.

SETTING UP GLUED CONTACT (Cont.)

- **In SOLs 101 and 400, if contact is initially not true set NLGLUE on BCPARA to 1**
- **For SOL 400 with a mixture of glued and non-glued bodies, BCPARA,0,NLGLUE,1 must be used**

MSC Software®

CASE STUDY 2

SHELL EDGE-TO-EDGE GLUED CONTACT

- **First define the contact bodies**

SHELL EDGE-TO-EDGE GLUED CONTACT (Cont.)

- **The contact bodies are located in the same plane with the same thickness**

Shell thickness display
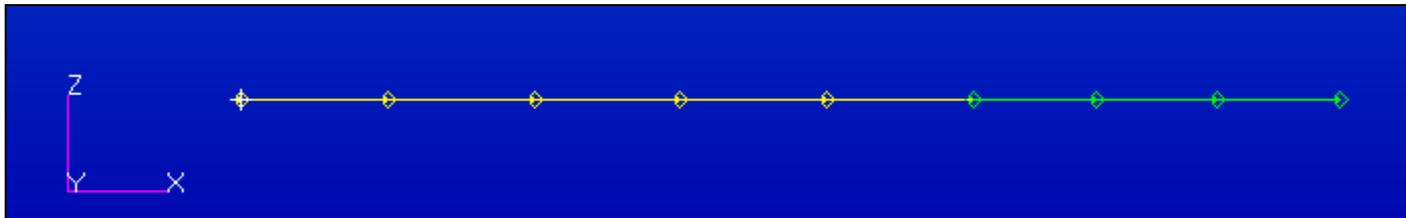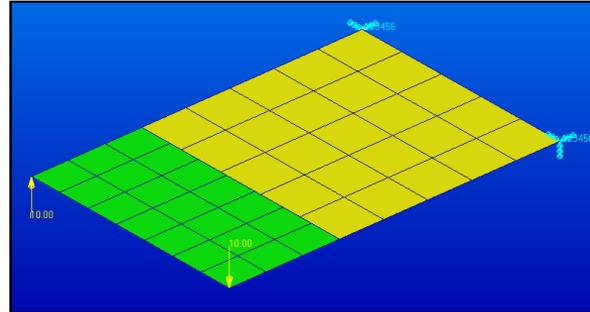
SHELL EDGE-TO-EDGE GLUED CONTACT (Cont.)

- **Next define the contact table**
  - Set contact detection to 2nd $\rightarrow$ 1st

MSC Software
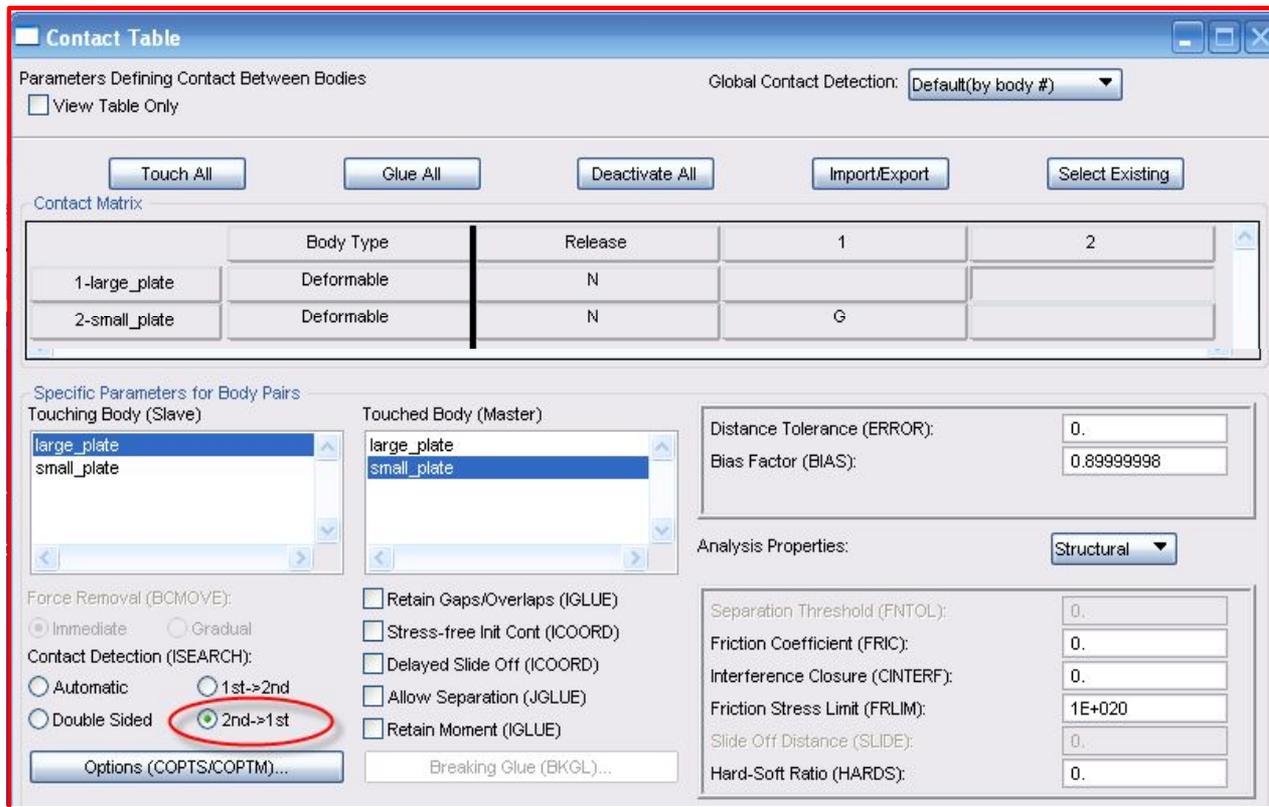
SHELL EDGE-TO-EDGE GLUED CONTACT (Cont.)

- **Next define the contact table**
  - Important step: Set contact option to ignore shell thickness for both bodies. This tells MSC Nastran to ignore the top and bottom faces of the shell elements and simply glue the mid-planes of the shell bodies together.
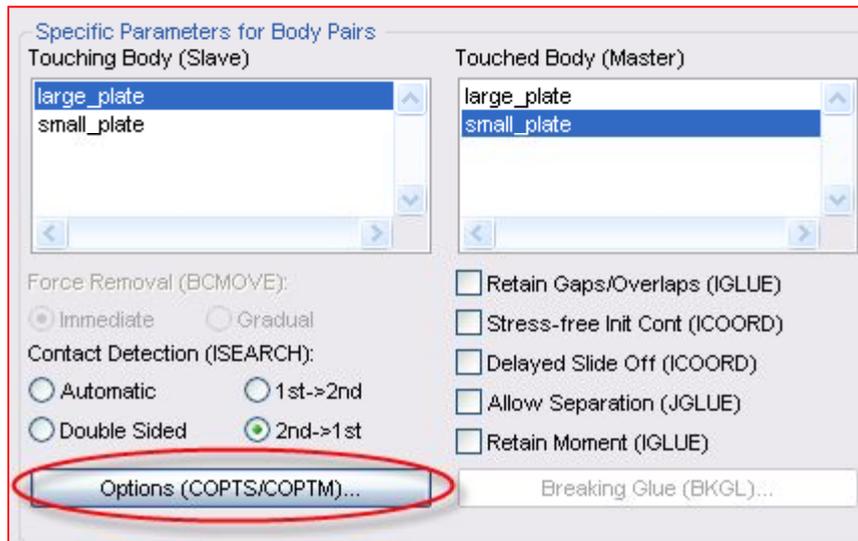
SHELL EDGE-TO-EDGE GLUED CONTACT (Cont.)

- ## Next define the contact table
  - Important step: Turn on moment carrying capability to enable the glued joint to transfer moments
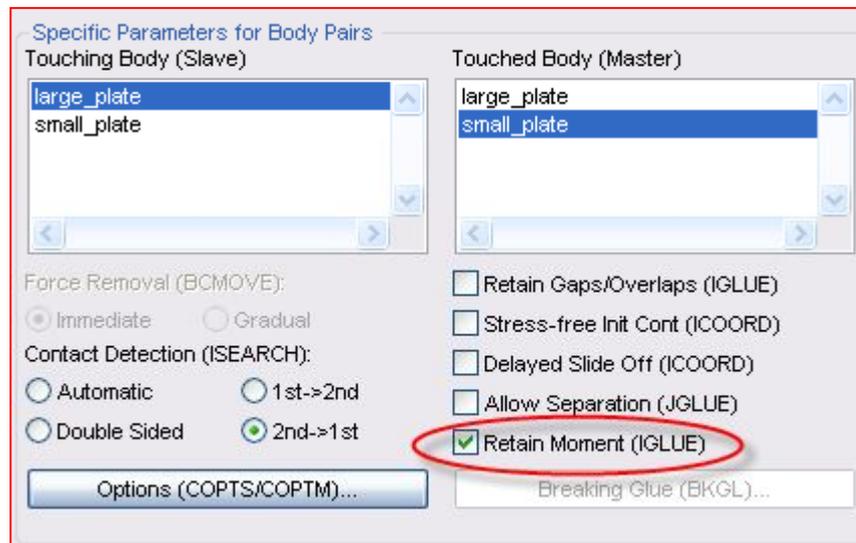
SHELL EDGE-TO-EDGE GLUED CONTACT (Cont.)

- **What the MSC Nastran entry looks like**

SHELL EDGE-TO-EDGE GLUED CONTACT (Cont.)

- **Plot the deformed shape**
  - The two plates appear to deform as one assembly
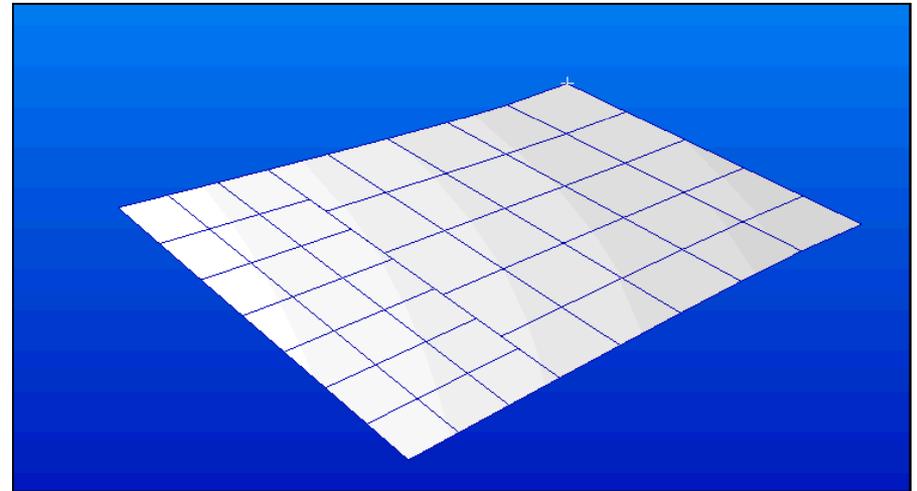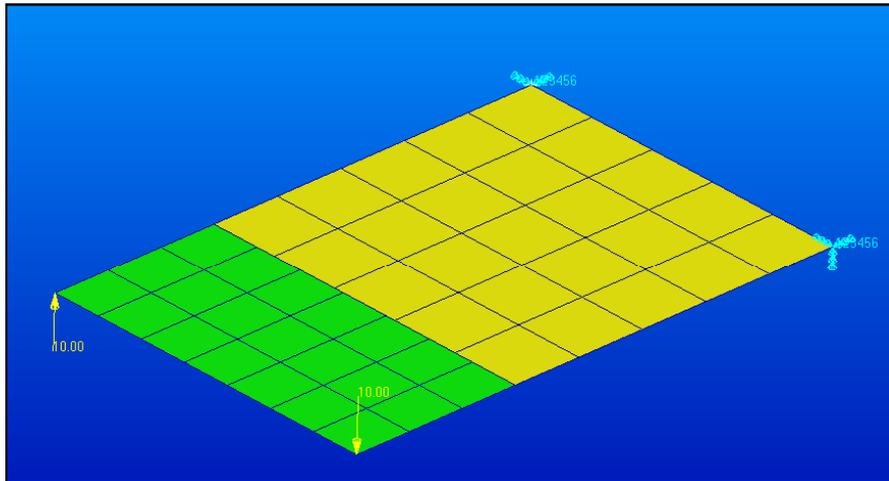
PLOT THE CONTACT STATUS

- **Plotting the Contact Status**
  - 0 indicates a retained node (master)
  - 1 indicates a tied node (slave)
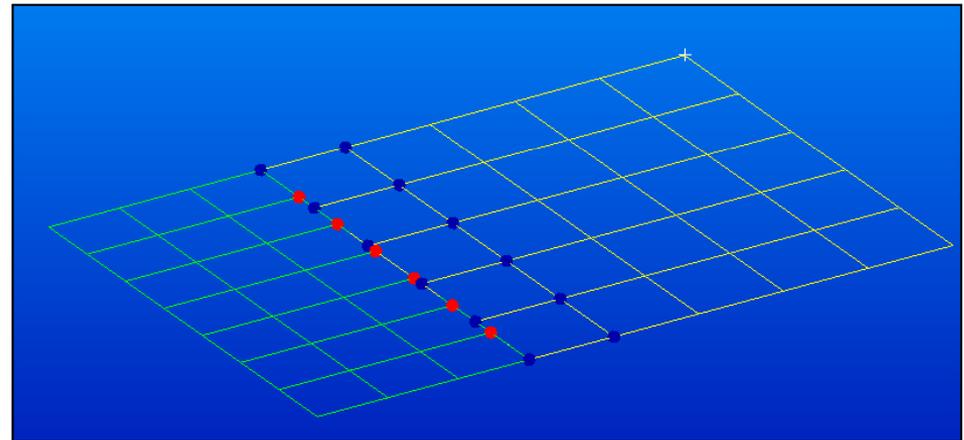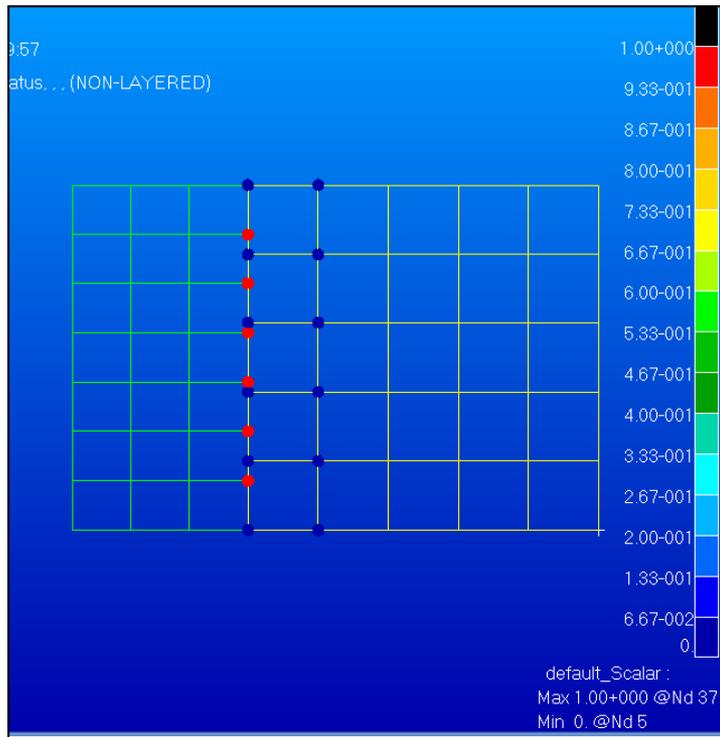
PLOT THE GLUE MPC EQUATIONS

- **Plot the MPC equations**
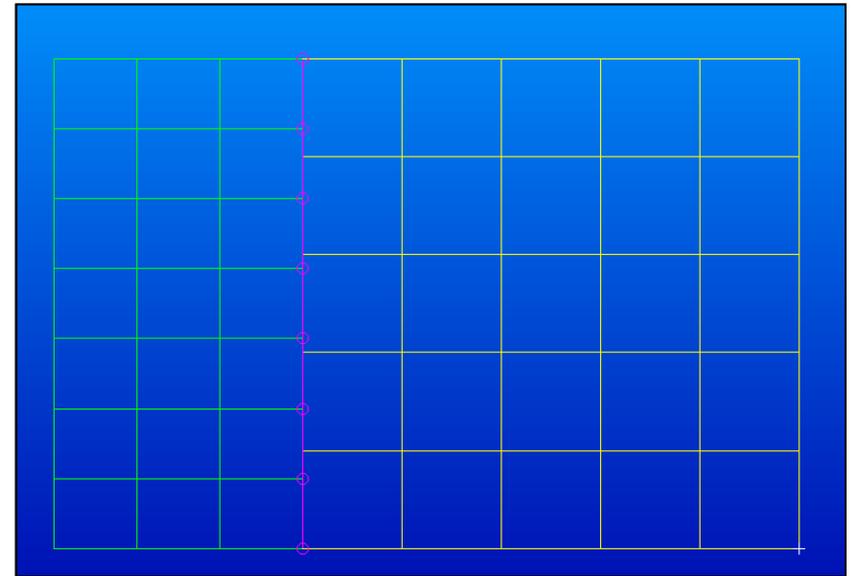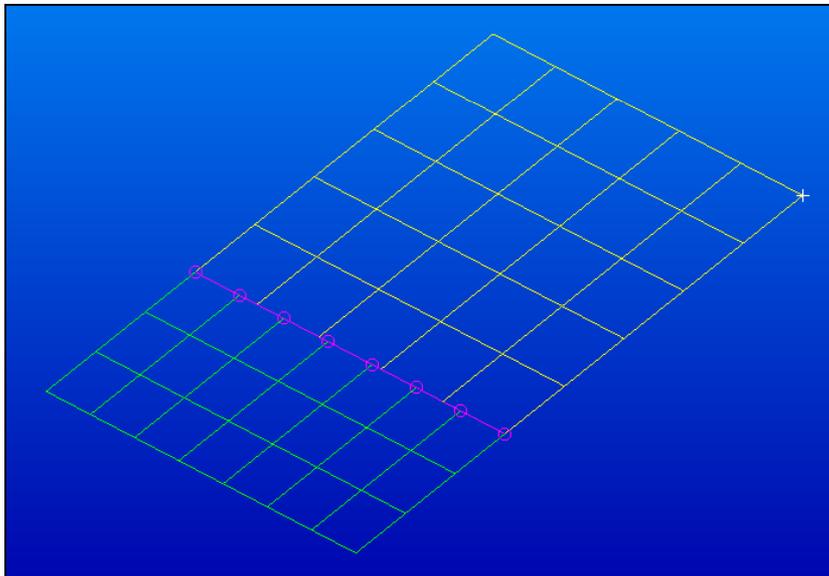
CASE STUDY 3

SHELL EDGE-TO-EDGE GLUED CONTACT WITH IN-PLANE GAP

- **Two shell bodies lie in the same plan**

- **An in-plane gap exists**

0.030 gap

0.060 gap

0.090 gap
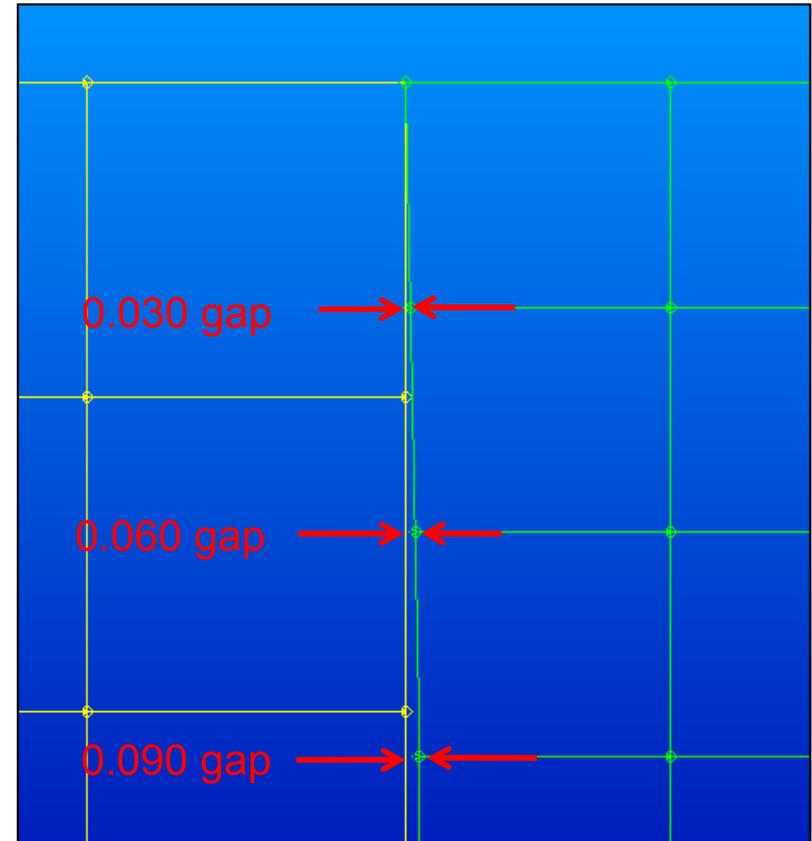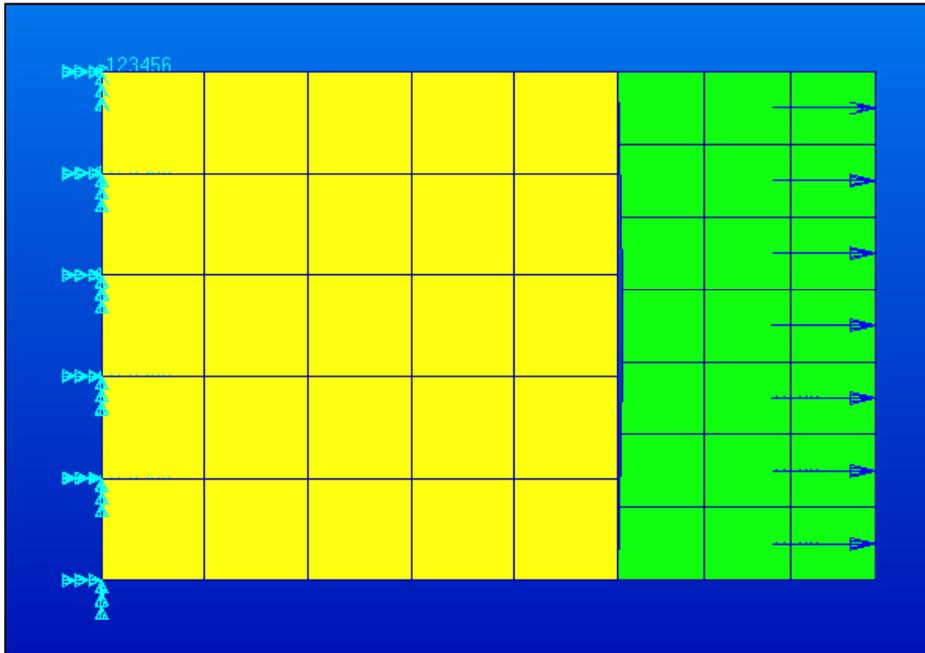
SHELL EDGE-TO-EDGE GLUED CONTACT WITH IN-PLANE GAP (Cont.)

- **Glued the two plates**

- **Plot the deformed shape**

- **The plate edges are partially glued**

MSC Software
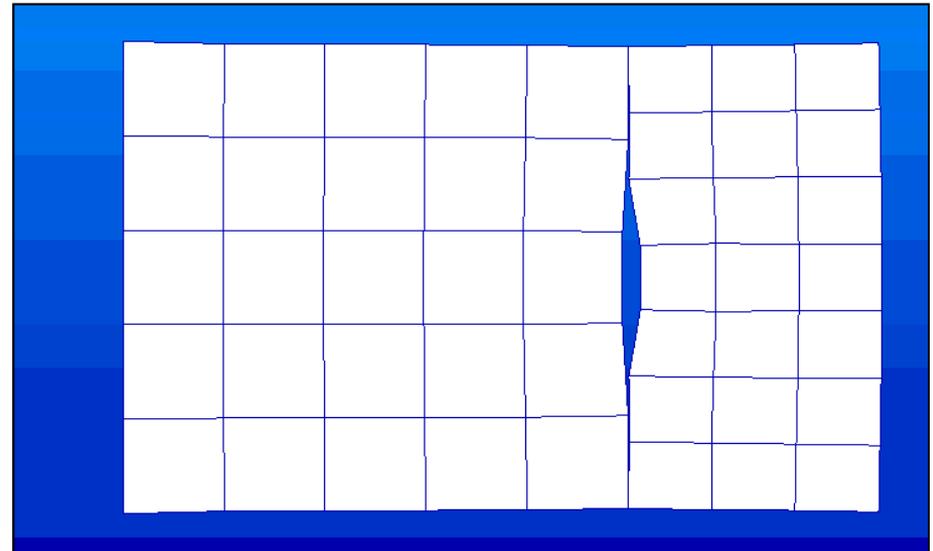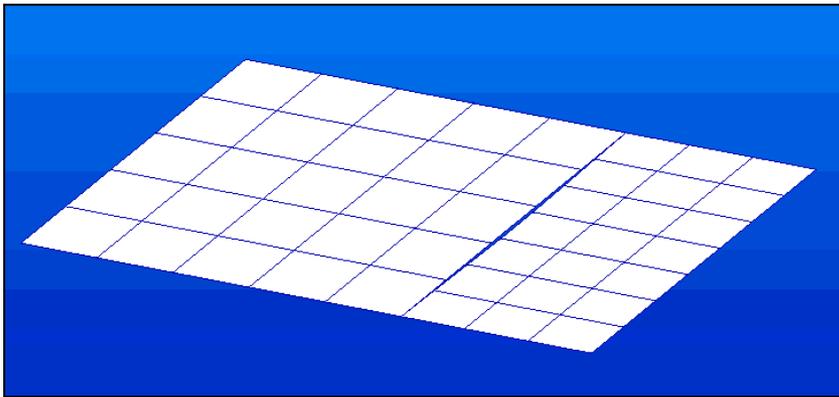
SHELL EDGE-TO-EDGE GLUED CONTACT WITH IN-PLANE GAP (Cont.)

- **Plot the contact status**
  - 2 nodes did not get glued

Missed 2 nodes
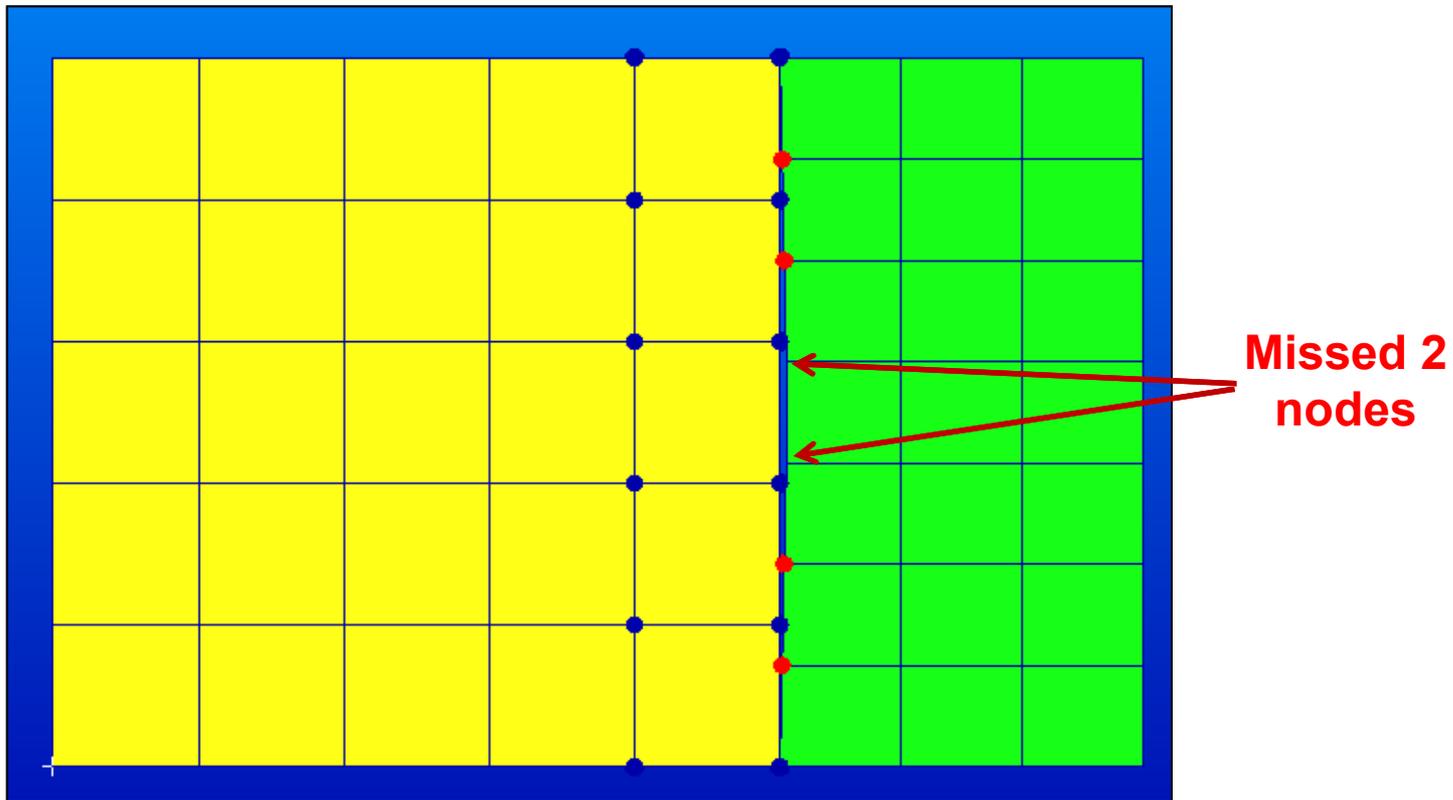
OUTPUT EQUIVALENT MPCS

- **Option to generate the equivalent MPC entries**

- **Add the following case control to the input file**

  NLOPRM MPCPCH=BEGN

- **A punch file (file_name.pch) is generated that can be brought into a pre-processing for plotting**

MSC Software

SHELL EDGE-TO-EDGE GLUED CONTACT WITH IN-PLANE GAP (Cont.)

- **Plot the MPC equations**
  - 2 nodes did not get glued

Missed 2 nodes
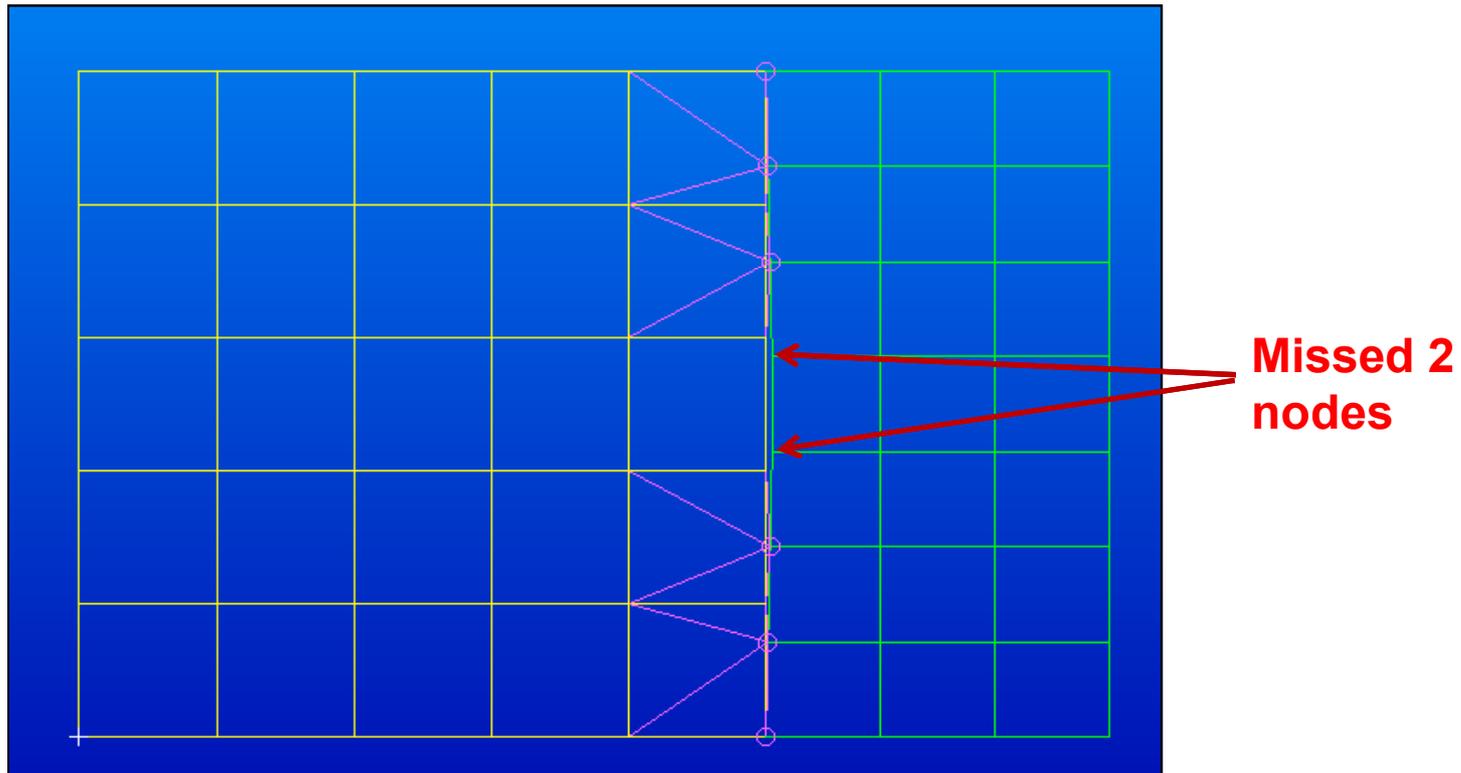
SHELL EDGE-TO-EDGE GLUED CONTACT WITH IN-PLANE GAP (Cont.)

- **Examine the default contact tolerance**
  - The default value is 0.0625
  - The largest in-plane gap is 0.090
  - Let's increase the contact tolerance to 0.100

MSC Software

SHELL EDGE-TO-EDGE GLUED CONTACT WITH IN-PLANE GAP (Cont.)

- **Analysis results for contact tolerance of 0.100**
  - Same results as before. No improvement.
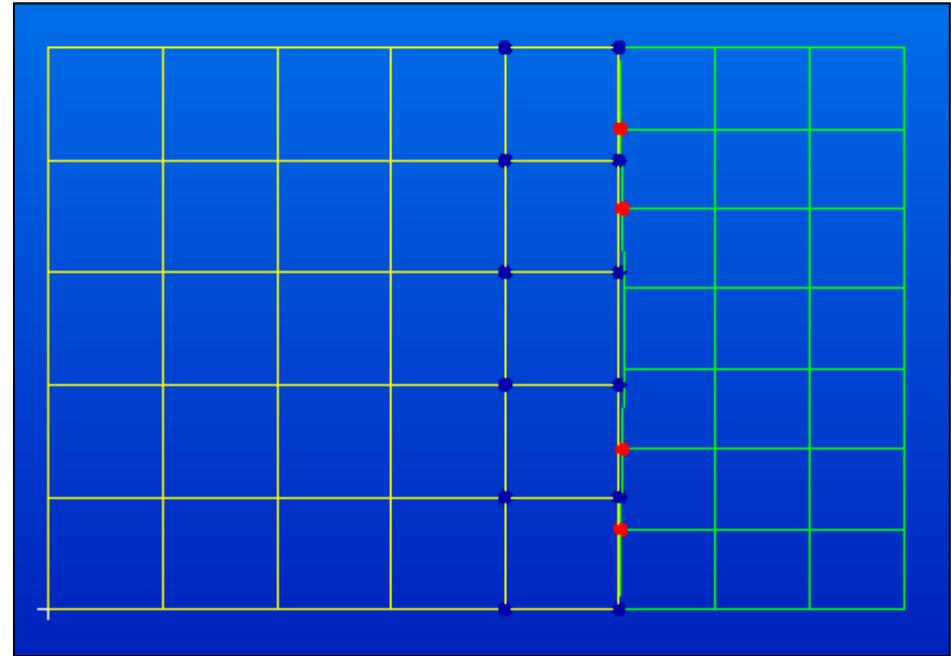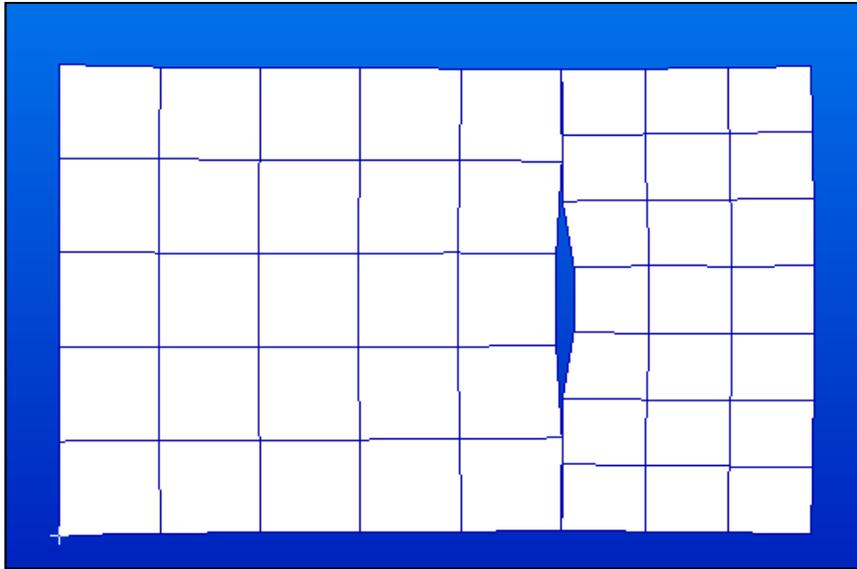
SHELL EDGE-TO-EDGE GLUED CONTACT WITH IN-PLANE GAP (Cont.)

- **How to handle in-plane gaps**
  - The contact tolerance (ERROR) is measured normal to the Master element face. That's why increasing this value did not help bridging across the in-plane gap.
  - The delayed slide off tolerance (**SLIDE**) can be used to bridge the in-plane gap
    - By default, this slide off tolerance is equal to the default contact tolerance. It extends the Master element edge by this amount.
    - If this default tolerance is not enough, then the user can set a specific slide off tolerance by
      - Set **ICOORD=2** to activate the SLIDE parameter
      - and set **SLIDE** to the desired value

MSC Software

SHELL EDGE-TO-EDGE GLUED CONTACT WITH IN-PLANE GAP (Cont.)
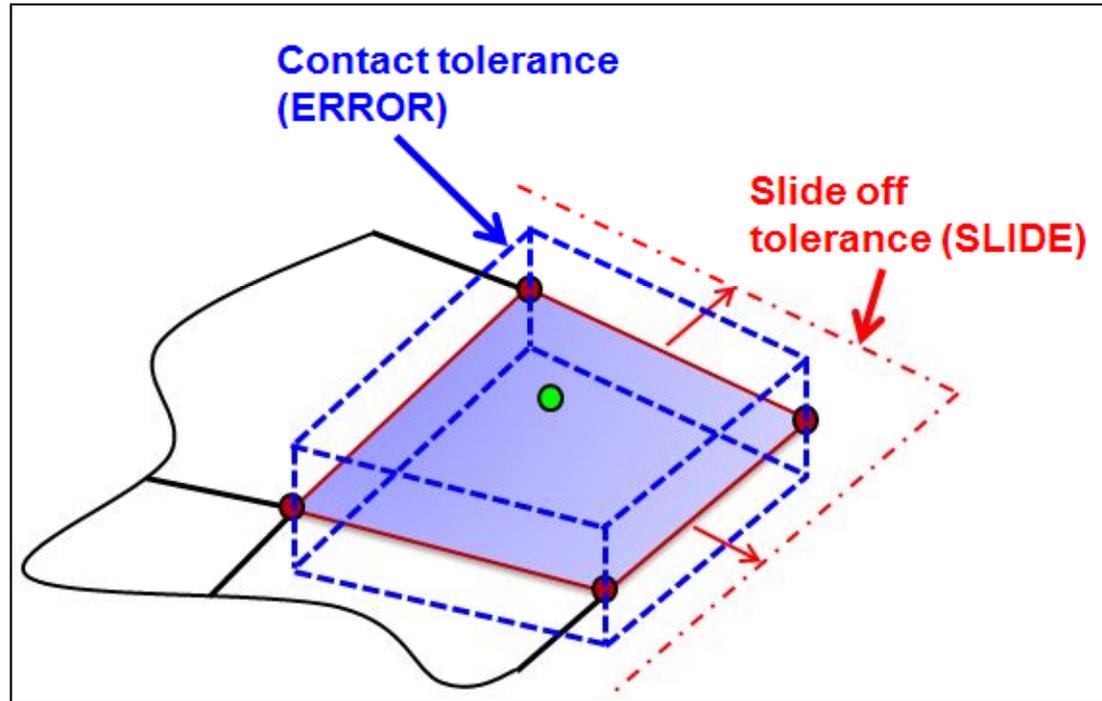
SHELL EDGE-TO-EDGE GLUED CONTACT WITH IN-PLANE GAP (Cont.)

- **How to handle in-plane gaps**
  - Set ICOORD=2
  - Set SLIDE=0.100

SHELL EDGE-TO-EDGE GLUED CONTACT WITH IN-PLANE GAP (Cont.)

- **What the MSC Nastran entries looks like**

SHELL EDGE-TO-EDGE GLUED CONTACT WITH IN-PLANE GAP (Cont.)

- **Analysis results for new ICOORD and SLIDE settings**
  - Deformed shape plot shows that both plates appear to be fully connected

SHELL EDGE-TO-EDGE GLUED CONTACT WITH IN-PLANE GAP (Cont.)

- **Analysis results for new ICOORD and SLIDE settings**
  - Plot contact status
  - All nodes are connected
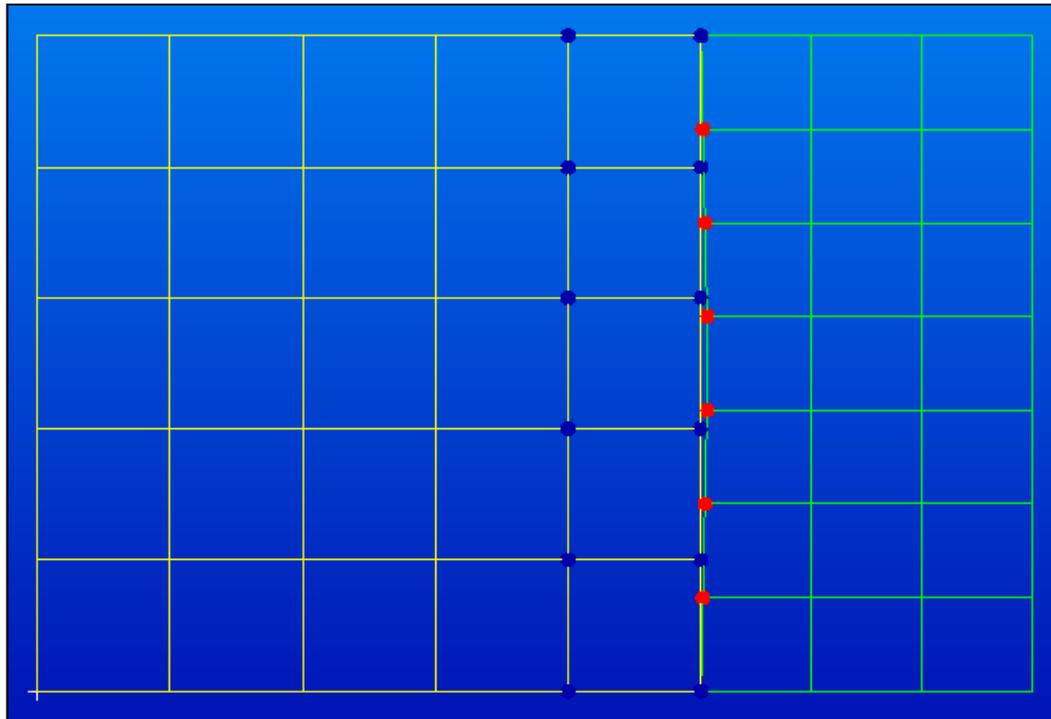
SHELL EDGE-TO-EDGE GLUED CONTACT WITH IN-PLANE GAP (Cont.)

- **Analysis results for new ICOORD and SLIDE settings**
  - Plot MPC equations
  - All nodes are connected
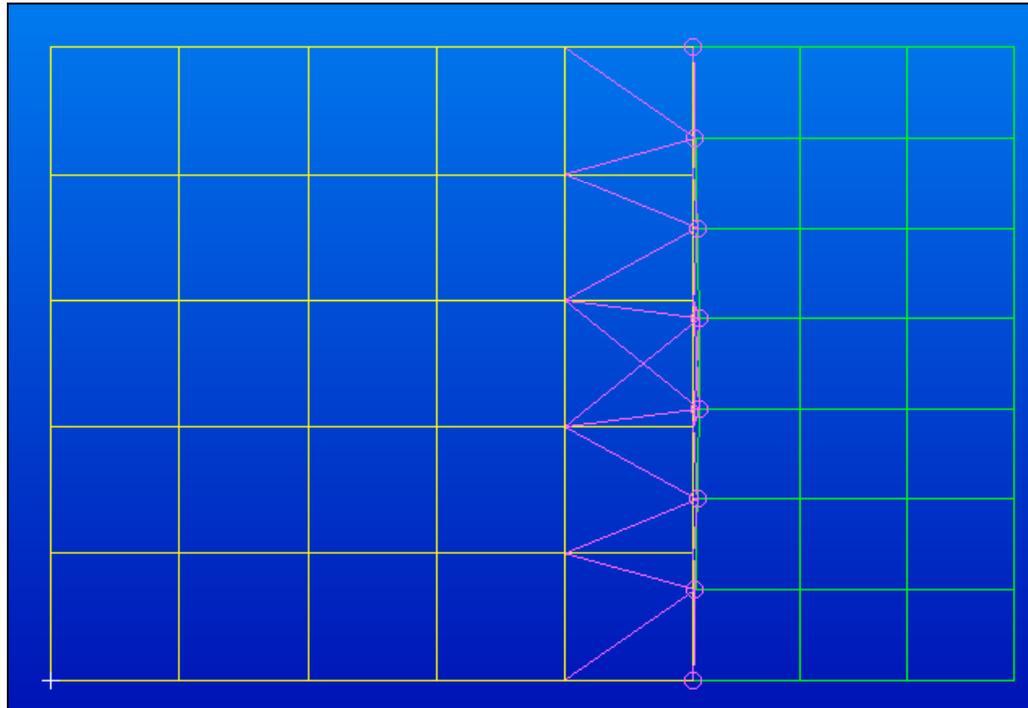
WORKSHOP 5: SOLID GLUE CONTACT

- **Please go to the Seminar Workbook where you will find step-by-step instructions for this workshop**

MSC Software®
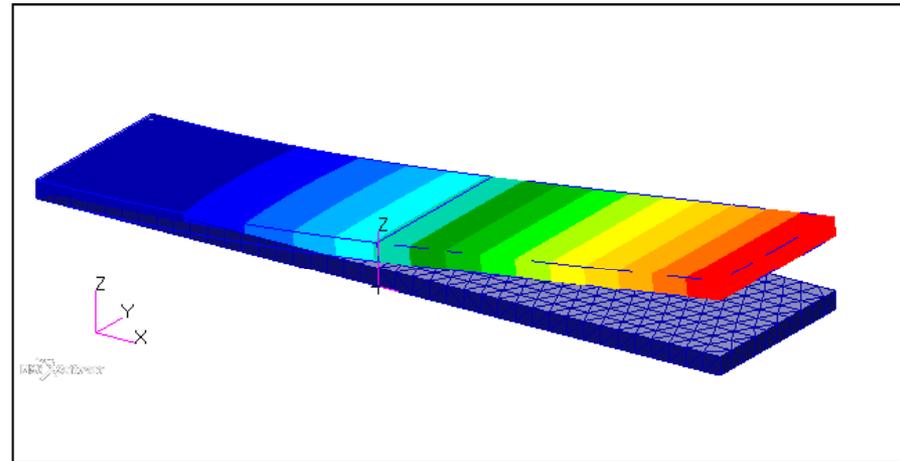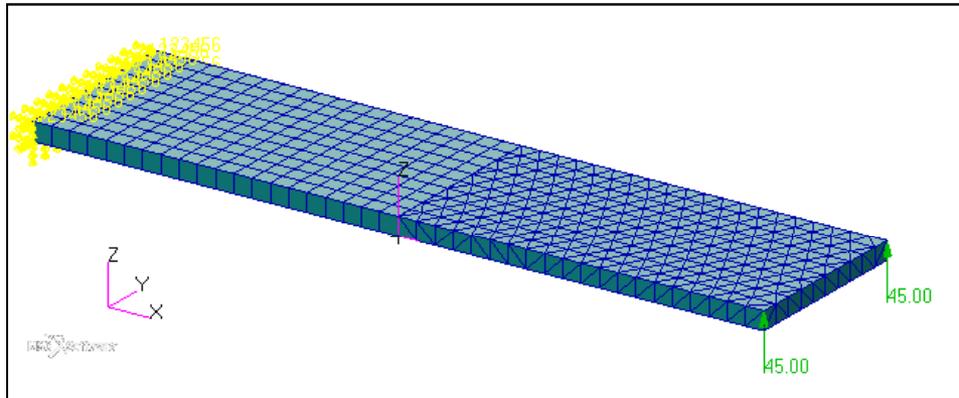
SECTION 6

MODEL CHECKOUT

CONTENTS

CONTENTS

CONTENTS

MSC Software

COMMON TYPES OF ERRORS

- **Mistakes in engineering judgment**

- **Approximations to physical behavior**
  - Engineering theory
  - Finite element theory
  - Finite element implementation
  - Modeling
    - Bolted connection
    - Welded connection
    - Corners
    - Transitions

- **Modeling errors**
  - Connections
    - Beam to plate
    - Beam to solid
    - Plate to solid
  - Beam orientation
  - Beam releases
  - Loading (how well do you know the loading yourself?)

MSC Software®

COMMON TYPES OF ERRORS (Cont.)

- **Finite element error**

- **Round-off error (can be disastrous when it occurs)**
  - Computers use binary arithmetic (If you enter .1, internally it may be .099999998)

- **Program bugs (nobody's perfect)**
  - A list of known errors is maintained and distributed

Eternal Vigilance is the Price of a Good Analysis

COMMON MODELING ERRORS

- **Plates not lining up = zipper**
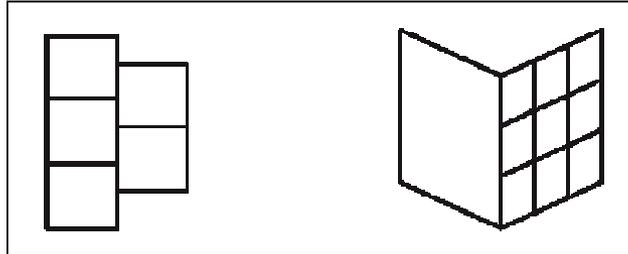
- **Any connections depending on in-plane rotational stiffness of plates, or any rotational stiffness on solids**

- **Instabilities – for example, releasing both ends of a beam in torsion**

- **Offsets of elements in wrong coordinate system (should be in the output coordinate systems of the grid points for Bars and Beams)**

- **Member properties wrong (beam orientation) – also plates – membrane only – left out bending**

- **Beam end releases – are they local or global**

- **Element force output is normally in element coordinate system**

DIAGNOSIS OF A NEW MODEL - PARAMs

PARAM	Operation
AUTOSPC, EPPRT, MAXRATIO	Check relative magnitudes of matrix terms
FIXEDB	Solve superelements individually
	Statics = fixed-boundary solution
	Dynamics = calculated component modes
IRES	Load Error
GRDPNT	Check mass, CG, M. Moment of Inertia
USETPRT	Print set tables
SESEF	Strain energy fractions (superelements – SOL 103)
TINY	Minimum percentage value of element strain energy for printout (Values not printed are not available for post-processing)

MSC Software

DIAGNOSIS OF A NEW MODEL - DIAGs

DIAG	Operation
8	Print matrix trailers
14	Print DMAP listing
15	Print table trailers
56	List Qualifier changes as the solution progresses – also, list all DMAP statements executed on the .f04 file (normally only modules are listed)

- **MSC NASTRAN DATA BLOCK NAME CONVENTION FOR MATRICES**

$$\mathbf{KYIJ} = \left[\mathbf{K}_{IJ}^{Y} \right]$$

where

Y = type: A, D, 4		I,J = col, row sets	
K = stiffness		M = mass	
B = viscous damping		G = transformation	
D = rigid body transformation		U = displacement	
P = load		Q = force of constraint	

F04 OUTPUT: TIME LOG AND DMAP TRACE FORMAT

- **Prints matrix trailers as the matrices are created**

Day_Time Elapsed I/O_Mb Del_Mb CPU_Sec Del_CPU SubDMAP Line (S)
SubDMAP/Module

- Module Name
- DMAP Sequence ID
- subDMAP
- Elapsed Time for Job (used for "time" limit)
- File Operations
- Wall Clock – Elapsed Seconds
- Time of Day
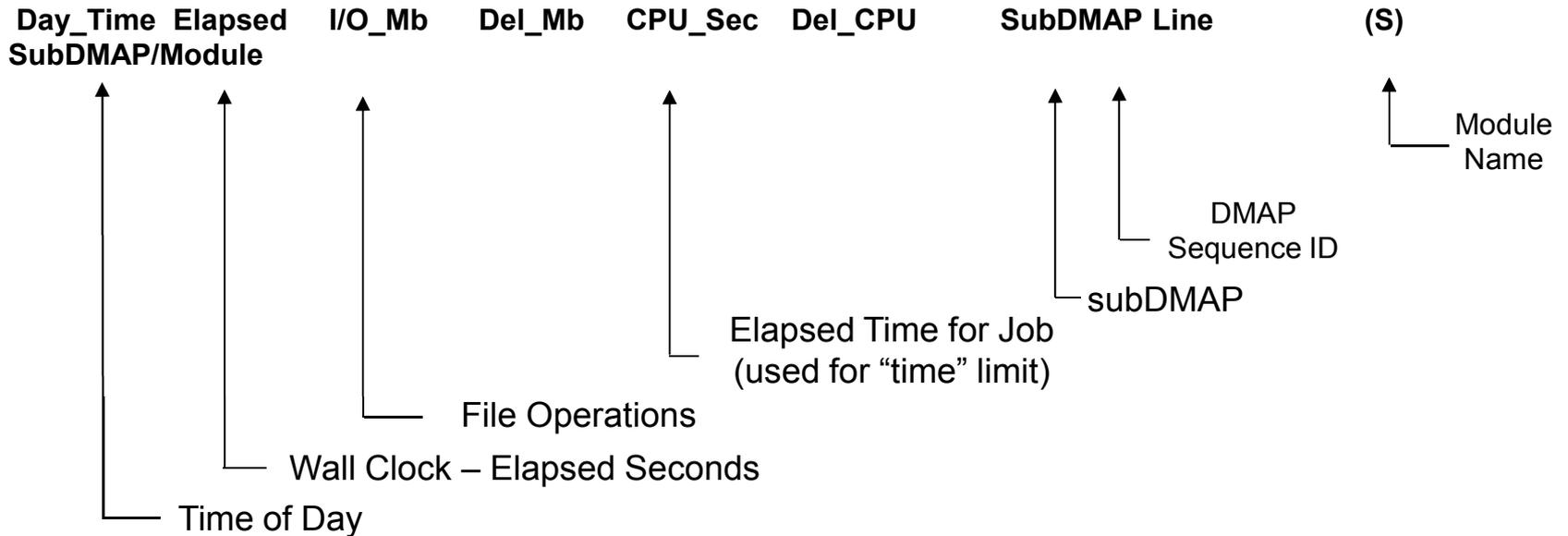
Day_Time	Elapsed	I/O_Mb	Del_Mb	CPU_Sec	Del_CPU	SubDMAP	Line	(S)SubDMAP/Module
18:04:40	0:00	152.0	1.0	0.1	0.0	SEMODES	56	(S)IFPL BEGN *
18:04:41	0:01	271.0	11.0	0.8	0.0	SEKR	18	UPARTN END

MSC Software

DIAG 8 F04 OUTPUT – Matrix Trailers

- **Sample printout using DIAG 8**

```
DAY TIME      ELAPSED      I/O MB       DEL_MB         CPU SEC       DEL_CPU      SUB_DMAP/DMAP_MODULE MESSAGES
14:16:23      0:16         18.6             .0           3.5             .0       SEMG     28 EMG      BEGN
*8**  Module DMAP Matrix      Cols  Rows  F  T  NzWds       Density BlockT StrL NbrStr  EndAvg  BndMax  NulCol
         EMG             28   KELM          1   300 2  2   600  1.00000D+00     3   300      1    300     300    0 *8**
14:16:24      0:17         18.6             .0           3.5             .0       SEMG    111 EMG      BEGN
*8**  Module DMAP Matrix      Cols  Rows  F  T  NzWds       Density BlockT StrL NbrStr  EndAvg  BndMax  NulCol
         EMG            111   KJJM         48    48 6  2    48  2.50000D+00     3   3      192     22      45   24 *8**
```

- **Definitions**

F(orm)	1=square, 2=rectangle, 3=diagonal, 6=symmetric, etc.
T(ype)	1=single precision real, 2=double precision real, 3=single precision complex, 4=double precision complex
Nzwds	Largest Number of nonzero words among all columns
Den	Density, (number of nonzero terms) ÷ (Rows x Columns))
BlockT	Number of GINO blocks (1 block = "buffsize -1" words)

MINIMUM RECOMMENDED MODEL CHECKS

Pre-Analysis

- **Understand the structure and the elements**
  - Make small models – understand the problem
  - Use pilot models in areas of uncertainty
  - If you are not familiar with using the element type or SOLution you expect to use, make simple models and compare the answers to theoretical results (with a simple model, you should be able to obtain excellent correlation with theoretical results).

- **Model checks before the analysis**
  - Geometry
    - Pre-processor (or Undeformed plots)
      - Look at connections between different element types
        » Based on knowledge of elements
        » Based on loads
        » Look at corners (QUAD plates)
      - Shrink plots

MSC Software

MINIMUM RECOMMENDED MODEL CHECKS (Cont.)

- **Elements**
  - Beam and bar
    - Check that I1 and I2 have proper orientation and values
    - Check all end releases (in member coordinates)
    - Verify all offsets (in output coordinate system of GRIDs)
    - Material – need E, ν (or G), and ρ
  - Plates and Shells
    - Check aspect ratios, taper, and warpage
    - Check orientation – Z, surfaces consistent
    - Check attachments – especially any depending on in-plane rotational stiffness, any corners, and "shells"
    - Verify any offsets (in element coordinate system)
    - Material – need E, ν (or G), and ρ
    - Property entry – be sure to get the correct properties. (One of the most commonly made errors is <u>not</u> specifying MID2 for "bending" plates

MINIMUM RECOMMENDED MODEL CHECKS (Cont.)

- **Solids**
  - Check aspect ratios
  - Check taper
  - Check attachments. If any attachments depend on rotational stiffness, special modeling effort is required
  - Material – need E, ν (or G), and ρ

- **Mass properties**
  - Check ρ on MATi entries
  - Check NSM on property entries
    - Bars, beams = mass/unit length
    - Plates = mass/unit area
  - Submit with PARAM, GRDPNT, xxxx

    where xxxx = ID of GRID point to calculate mass properties about
    - Always check center of gravity and total weight (mass) versus known values

MSC Software®

MINIMUM RECOMMENDED MODEL CHECKS (Cont.)

- **Loadings:**
  - Verify they are correct (OLOAD RESULTANT)

- **Constraints:**
  - Verify that they are defined (often they are forgotten)
  - Verify they are correct (location and orientation – in output coordinate system of the GRID points)
  - Verify that they are applied (SPC CASE CONTROL command)

- **Static Checks – ALWAYS RUN STATICS FIRST!!!**
  - Apply 1–g in X, Y, and Z directions independently
    - Check load paths (GPFORCE)
    - Check reactions (SPC FORCE)
      - Does total = applied load?
      - Are the reactions at the correct locations and do they have the correct orientation?
    - In Dynamics, approximate frequency: $f = \dfrac{1}{2\pi}\sqrt{\dfrac{g}{d}}$

    where d = center of gravity displacement in direction of applied g-load

    g = acceleration due to gravity

MSC Software

MINIMUM RECOMMENDED MODEL CHECKS (Cont.)

- **Equilibrium check – verify model is not over-constrained**
  - Run free-free. Remove known constraints and check for unconstrained motion under applied loads or imposed displacements.

    or
  - Use the Case Control Command GROUNDCHECK, to check for over-constrained systems.
  - Thermal equilibrium check – if thermal loads are to be considered.
  - Check α on MATi entries
  - Check for unconstrained thermal expansion – on a copy of your model
    - Apply a determinate set of constraints
    - Use the same α for all materials
    - Apply a uniform ΔT to the structure. It should expand "freely," that is, with no reactions, element forces, or stresses

MSC Software

MINIMUM RECOMMENDED MODEL CHECKS (Cont.)

After the Analysis

- **Statics**
  - Check EPSILON and MAXRATIO
    - Epsilon > 10^{-9} may indicate trouble
    - MAXRATIO > 10^7 may indicate trouble
  - Check reactions. Do they equal the applied loads (Σ applied loads are printed as "OLOAD RESULTANT" in superelement solutions)?
  - Check load paths – use grid point force balance to "trace" loads
    - Check stress contours for "consistency"
      - "Sharp" corners indicate bad modeling
      - Use different options (i.e., topological and geometric) and compare results
      - Check stress discontinuities
      - Compare values to "hand calc" or small model results

MINIMUM RECOMMENDED MODEL CHECKS (Cont.)

- **Dynamics – normal modes**
  - Check frequencies. Are they in the expected range?

    (Did you forget WTMASS???)
    - If free-free, are there six "rigid-body" (f=0.0) modes?
    - Are there any mechanisms (f=0.0)?
      - More than six "rigid-body" modes in free-free?
      - Any "rigid-body" modes in constrained modes?
  - Check mode shapes, and Identify modes
    - Plots and/or animation
    - Effective weight and kinetic energy
      - (Case Control Commands **MEFFMASS** and **EKE**) help to identify "significant" modes

STIFFNESS MATRIX CHECKS

- **The model (stiffness and mass matrices) should be checked to verify that the elements are not (obviously) bad and that the model is not over-constrained**

  - Sample – CELASi between non coincident points or CELASi to ground

- **This check can be performed at various stages during the analysis – at each stage, a potential problem is checked**

  - G-set – at this stage of the solution, the elements (including GENELs and K2GG) are checked for grounding

  - N-set – at this stage, the MPC equations are checked

  - A-set – (free-free only) check that the SPC's do not over-constrain the structure

- **Use the Case Control Command GROUNDCHECK**

STIFFNESS MATRIX CHECKS (Cont.)

- **Sample Model 1 – Cantilever Beam**
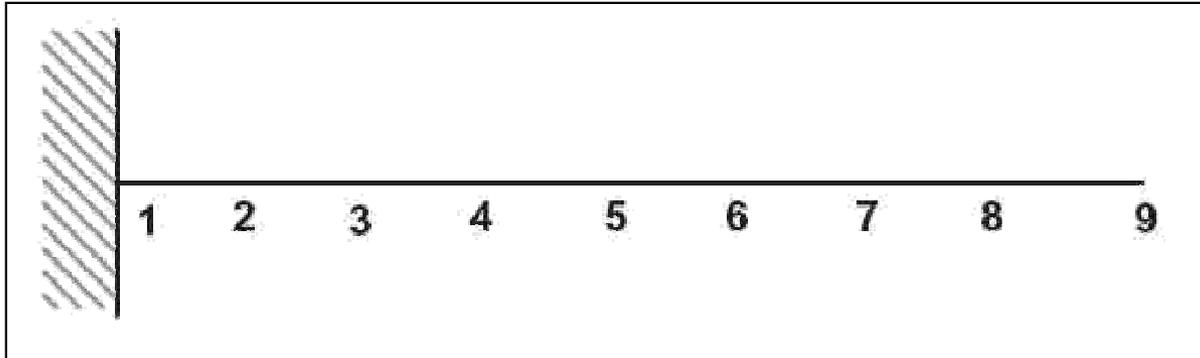
- – Properties:
  - **I1 = 10**
  - **I2 = 10**
  - **J = 5**
  - **A = 1**
  - **E = 10,000,000.**
  - **ν = .3**
  - **ρ = .1**
  - **WTMASS = .002588**

MSC Software®

STIFFNESS (AND MASS) CHECKS

- **Input File: ground_check_1a.dat**

```
SOL 103
CEND
TITLE = Cantilever Beam Modeled with 8 CBAR elements
GROUNDCHECK=YES
SUBCASE 1
    SUBTITLE=Default
    LABEL = Perform Model Checks
    METHOD = 1
    SPC = 1
    VECTOR(SORT1,REAL)=ALL
BEGIN BULK
MAT1      1          1.+7              .3        .1
PBAR, 1,  1,    1., 10., 10., 5.
CBAR, 1,  1,  1, 2,  0., 1., 0.
CBAR, 2,  1,  2, 3,  0., 1., 0.
CBAR, 3,  1,  3, 4,  0., 1., 0.
CBAR, 4,  1,  4, 5,  0., 1., 0.
CBAR, 5,  1,  5, 6,  0., 1., 0.
CBAR, 6,  1,  6, 7,  0., 1., 0.
CBAR, 7,  1,  7, 8,  0., 1., 0.
CBAR, 8,  1,  8, 9,  0., 1., 0.
GRID      1                 0.00    0.      0.
GRID      2                 1.25    0.      0.
GRID      3                 2.50    0.      0.
GRID      4                 3.75    0.      0.
GRID      5                 5.00    0.      0.
GRID      6                 6.25    0.      0.
GRID      7                 7.50    0.      0.
GRID      8                 8.75    0.      0.
GRID      9                10.00    0.      0.
PARAM     GRDPNT  0
PARAM     WTMASS  .002588
PARAM     AUTOSPC YES
SPC1      1       123456  1
EIGRL     1                         5
ENDDATA
```

MSC Software

OUTPUT FROM ground_check_1a

```
0                      M O D E L   S U M M A R Y        BULK = 0
                     ENTRY NAME          NUMBER OF ENTRIES
                     ----------          -----------------
                        CBAR                      8
                        EIGRL                     1
                        GRID                      9
                        MAT1                      1
                        PARAM                     3
                        PBAR                      1
                        SPC1                      1

 ^^^
 ^^^  >>> IFP OPERATIONS COMPLETE <<<
 ^^^
1     CANTILEVER BEAM MODELED WITH 8 CBAR ELEMENTS                      MARCH   5, 2012  MSC.NASTRAN 11/25/11    PAGE     6
0
                 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
0                                        REFERENCE POINT =       0
                                                 M O
          *   1.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00 *
          *   0.000000E+00   1.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   5.000000E+00 *
          *   0.000000E+00   0.000000E+00   1.000000E+00   0.000000E+00  -5.000000E+00   0.000000E+00 *
          *   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00 *
          *   0.000000E+00   0.000000E+00  -5.000000E+00   0.000000E+00   3.359375E+01   0.000000E+00 *
          *   0.000000E+00   5.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   3.359375E+01 *
                                                  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 *
```

MSC Software®

OUTPUT FROM ground_check_1a (Cont.)

```
         DIRECTION
MASS AXIS SYSTEM (S)        MASS                X-C.G.         Y-C.G.         Z-C.G.
         X            1.000000E+00        0.000000E+00   0.000000E+00   0.000000E+00
         Y            1.000000E+00        5.000000E+00   0.000000E+00   0.000000E+00
         Z            1.000000E+00        5.000000E+00   0.000000E+00   0.000000E+00
                                              I(S)
               *   0.000000E+00   0.000000E+00   0.000000E+00  *
               *   0.000000E+00   8.593750E+00   0.000000E+00  *
               *   0.000000E+00   0.000000E+00   8.593750E+00  *
                                              I(Q)
               *   0.000000E+00                               *
               *                  8.593750E+00                *
               *                                 8.593750E+00 *
                                               Q
               *   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  *
```

OUTPUT FROM ground_check_1a (Contd.)

```
*** USER INFORMATION MESSAGE 7570 (GPWG1D)
    RESULTS OF RIGID BODY CHECKS OF MATRIX KGG       (G-SET)  FOLLOW:
    PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF   1.228800E-01
         DIRECTION          STRAIN ENERGY          PASS/FAIL
         ---------          -------------          ---------
             1               0.000000E+00             PASS
             2               0.000000E+00             PASS
             3               0.000000E+00             PASS
             4               0.000000E+00             PASS
             5               0.000000E+00             PASS
             6               0.000000E+00             PASS

    SOME POSSIBLE REASONS MAY LEAD TO THE FAILURE:
      1. CELAS1 ELEMENTS CONNECTING TO ONLY ONE GRID POINT;
      2. CELAS1 ELEMENTS CONNECTING TO NON-COINCIDENT POINTS;
      3. CELAS1 ELEMENTS CONNECTING TO NON-COLINEAR DOF;
      4. IMPROPERLY DEFINED DMIG MATRICES;

1     CANTILEVER BEAM MODELED WITH 8 CBAR ELEMENTS
```

OUTPUT FROM ground_check_1a (Cont.)

```
*** USER INFORMATION MESSAGE 5010 (LNCILD)
    STURM SEQUENCE DATA FOR EIGENVALUE EXTRACTION.
    TRIAL EIGENVALUE = 8.697013D+08, CYCLES = 4.693590D+03 NUMBER OF EIGENVALUES BELOW THIS VALUE = 2
*** USER INFORMATION MESSAGE 5010 (LNCILD)
    STURM SEQUENCE DATA FOR EIGENVALUE EXTRACTION.
    TRIAL EIGENVALUE = 1.786807D+10, CYCLES = 2.127448D+04 NUMBER OF EIGENVALUES BELOW THIS VALUE = 6
```

```
                                  R E A L   E I G E N V A L U E S
MODE    EXTRACTION    EIGENVALUE        RADIANS          CYCLES        GENERALIZED      GENERALIZED
NO.       ORDER                                                          MASS            STIFFNESS
  1          1       4.709041E+08     2.170032E+04     3.453714E+03     1.000000E+00     4.709041E+08
  2          2       4.709041E+08     2.170032E+04     3.453714E+03     1.000000E+00     4.709041E+08
  3          3       9.503416E+08     3.082761E+04     4.906367E+03     1.000000E+00     9.503416E+08
  4          4       8.335352E+09     9.129815E+04     1.453055E+04     1.000000E+00     8.335352E+09
  5          5       1.786391E+10     1.336559E+05     2.127200E+04     1.000000E+00     1.786391E+10
```

DESCRIPTION OF ground_check_1a OUTPUT

- **Grid Point Weight Output (GPWG module)**
  - The scale factor entered with parameter WTMASS is applied to the assembled element mass before GPWG. The GPWG module, however, converts mass back to the original input units that existed prior to the scaling effect of the parameter WTMASS
  - GPWG is performed on the *g*-size mass matrix, which is the mass matrix prior to the processing of the rigid elements, MPCs, and SPCs
  - Any masses at scalar points and fluid-related masses are not included in the GPWG calculation
  - GPWG for a superelement does not include the mass form upstream superelements. Therefore, GPWG for the residual structure includes only the mass of the residual (not any upstream superelements). The center of gravity location is also based on the mass of the current superelement only
  - The output from the GPWG is for information purposes only and is not used in the analysis
  - The rigid-body mass matrix [*MO*] is computed with respect to the reference grid point in the basic coordinate system. The Grid point to be used is specified using PARAM, GNDPNT
  - For further information see the *MSC NASTRAN Linear Static Analysis User's Guide*

DESCRIPTION OF ground_check_1a OUTPUT (Cont.)

- **Stiffness Check Output**
  - These checks are performed by multiplying the stiffness matrix by a set of rigid-body vectors(R_b) which are based on the geometry (calculated about PARAM, GRDPNT)
  - The rigid-body strain energy checks are calculated as (note that the factor of ½ is not included in the calculation)

$$R_b^T K R_b = CHKKii$$

  - This check is performed on the G-, N-, A-set matrices (i in CHKii is the set being checked)
  - If any term in the resulting "CHK" matrix exceeds the value of PARAM, CHECKTOL (default value is calculated based in the stiffness of your model), the results of the check are printed
  - "Reaction forces" are calculated, normalized to a minimum of 1.0, filtered, and printed (if CHECKTOL is exceeded)

$$KR_b = REACi$$

DESCRIPTION OF ground_check_1a OUTPUT (Cont.)

- **Stiffness Check Output (Cont.)**
  - Note that "full" data recovery is not performed, and if a DOF which does not belong to the remaining set is constrained, the nearest point (by connection) in the remaining set is indicated. See results for CHKKAA—point 1 is constrained, but does not belong to the A-set, therefore, the constraint shows up at point 2

- **Mass Check Output**
  - These checks are performed by multiplying the mass matrix by a set of rigid-body vectors(R_b) which are based on the geometry (calculated about PARAM, GRDPNT)
  - The calculation is similar to that performed on the stiffness matrix
  - The results at the G-set should match Grid Point Weight Generator
  - The checks at the N- and A-set check if MPCs and constraints remove (or re-distribute) mass

MSC Software

Ground_check_1b – MODEL WITH A BAD ELEMENT

- **Same model as before, only now connect a CELAS2 element between DOF 2 at Grid Points 2 and 3 (this will cause "grounding"), as the direction of the stiffness terms is not along the line connecting the GRID points)**

- **Samples of CELASi elements which cause "grounding"**

Connected to "Ground"
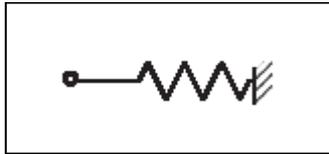
Geometric mismatch K_θ to ground
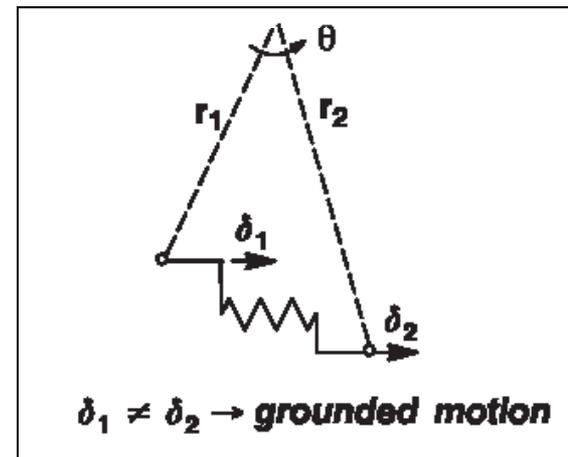
STIFFNESS CHECKS (CONT)

- **Input File – ground_check_1b.dat**

```
SOL 103
CEND
TITLE = Cantilever Beam with 8 CBAR + 1 CELAS2
GROUNDCHECK=YES
SUBCASE 1
    SUBTITLE=Default
    LABEL = Perform Model Checks
    METHOD = 1
    SPC = 1
    VECTOR(SORT1,REAL)=ALL
BEGIN BULK
MAT1        1          1.+7              .3        .1
PBAR, 1,  1,    1., 10., 10., 5.
CBAR, 1, 1,   1, 2,   0., 1., 0.
CBAR, 2, 1,   2, 3,   0., 1., 0.
.
.
.
CBAR, 8, 1,   8, 9,   0., 1., 0.
$ Add a CELAS2 incorrectly specified
CELAS2, 99, 1000., 2, 2,   3, 2
GRID       1                  0.00    0.        0.
GRID       2                  1.25    0.        0.
.
.
.
GRID       9                 10.00    0.        0.
PARAM      GRDPNT   0
PARAM      WTMASS   .002588
PARAM      AUTOSPC  YES
SPC1       1         123456  1
EIGRL      1                          5
ENDDATA
```

OUTPUT FROM ground_check_1b

```
*** USER INFORMATION MESSAGE 7570 (GPWG1D)
    RESULTS OF RIGID BODY CHECKS OF MATRIX KGG       (G-SET)  FOLLOW:
    PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF   1.228801E-01
        DIRECTION        STRAIN ENERGY         PASS/FAIL
        ---------        -------------         ---------
            1            0.000000E+00             PASS
            2            0.000000E+00             PASS
            3            0.000000E+00             PASS
            4            0.000000E+00             PASS
            5            0.000000E+00             PASS
            6            7.812500E+02             FAIL

    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;
1    CANTILEVER BEAM WITH 8 CBAR + 1 CELAS2                          MARCH   5, 2012  MSC.NASTRAN 11/25/11   PAGE     9
```

```
0      PERFORM MODEL CHECKS                                                                              SUBCASE 1

                                     R E A L   E I G E N V A L U E S
    MODE    EXTRACTION      EIGENVALUE          RADIANS            CYCLES         GENERALIZED        GENERALIZED
    NO.       ORDER                                                                  MASS            STIFFNESS
      1          1          4.709041E+08      2.170032E+04      3.453714E+03      1.000000E+00      4.709041E+08
      2          2          4.709118E+08      2.170050E+04      3.453742E+03      1.000000E+00      4.709118E+08
      3          3          9.503416E+08      3.082761E+04      4.906367E+03      1.000000E+00      9.503416E+08
      4          4          8.335352E+09      9.129815E+04      1.453055E+04      1.000000E+00      8.335352E+09
      5          5          1.786391E+10      1.336559E+05      2.127200E+04      1.000000E+00      1.786391E+10
1    CANTILEVER BEAM WITH 8 CBAR + 1 CELAS2                          MARCH   5, 2012  MSC.NASTRAN 11/25/11   PAGE    12
```

MSC Software

RESULTS OF ground_check_1b

- **At the G-set, the structural matrices are grounded when the alter attempts to rotate the model about the z-axis**

- **This is indicated by the large term in the CHKKGG matrix for DOF 6**

- **By looking at the REACGNRM matrix – this matrix represents the forces (normalized to a maximum of 1.0) preventing the model from moving as a rigid body. The column associated with DOF 6 (z-rotation) contains terms for DOF 2 of grid points 2 and 3, indicating that a modeling error exists in that area**

- **This is the location of the CELAS2**

Ground_check_1c – MODEL WITH A BAD MPC

- **Same model as before, only now connect an MPC between DOF 2 at Grid Points 2 and 3 (since the points are not coincident, this will cause "grounding")**

- **The MPC states that the y-direction translation of the Grid Point 2 must equal the y-direction translation of Grid Point 3**

Ground_check_1c – IMPROPER MPC

- **Input File: ground_check_1c.dat**

```
SOL 103
CEND
TITLE = Cantilever Beam with 8 CBAR, and 1 MPC
GROUNDCHECK(SET=(G,N))=YES
SUBCASE 1
    SUBTITLE=Default
    LABEL = Perform Model Checks
    METHOD = 1
    MPC = 1
    SPC = 1
    VECTOR(SORT1,REAL)=ALL
BEGIN BULK
MAT1      1        1.+7                    .3       .1
PBAR, 1,  1,    1., 10.,  10.,  5.
CBAR, 1, 1,   1, 2,   0., 1.,  0.
CBAR, 2, 1,   2, 3,   0., 1.,  0.
CBAR, 3, 1,   3, 4,   0., 1.,  0.
CBAR, 4, 1,   4, 5,   0., 1.,  0.
CBAR, 5, 1,   5, 6,   0., 1.,  0.
CBAR, 6, 1,   6, 7,   0., 1.,  0.
CBAR, 7, 1,   7, 8,   0., 1.,  0.
CBAR, 8, 1,   8, 9,   0., 1.,  0.
$ Add an MPC Equation (incorrectly specified)
MPC, 1,   2,2,1.,    3, 2, -1.
GRID      1                    0.00    0.      0.
GRID      2                    1.25    0.      0.
GRID      3                    2.50    0.      0.
GRID      4                    3.75    0.      0.
GRID      5                    5.00    0.      0.
GRID      6                    6.25    0.      0.
GRID      7                    7.50    0.      0.
GRID      8                    8.75    0.      0.
GRID      9                   10.00    0.      0.
PARAM    GRDPNT   0
PARAM    WTMASS   .002588
PARAM    AUTOSPC  YES
SPC1     1        123456  1
EIGRL    1                              5
ENDDATA
```

MSC Software

OUTPUT FORM ground_check_1c

```
*** USER INFORMATION MESSAGE 7570 (GPWG1D)
    RESULTS OF RIGID BODY CHECKS OF MATRIX KGG      (G-SET)  FOLLOW:
    PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF   1.228800E-01
            DIRECTION           STRAIN ENERGY           PASS/FAIL
           ----------          --------------          ---------
                1               0.000000E+00              PASS
                2               0.000000E+00              PASS
                3               0.000000E+00              PASS
                4               0.000000E+00              PASS
                5               0.000000E+00              PASS
                6               0.000000E+00              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;

1  CANTILEVER BEAM WITH 8 CBAR, AND 1 MPC                    MARCH   5, 2012  MSC.NASTRAN 11/25/11   PAGE    9
```

OUTPUT FORM ground_check_1c (Cont.)

```
*** USER INFORMATION MESSAGE 7570 (GPWG1D)
    RESULTS OF RIGID BODY CHECKS OF MATRIX KNN        (N-SET)  FOLLOW:
    PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF   1.228800E-01
          DIRECTION             STRAIN ENERGY        PASS/FAIL
          ---------             -------------        ---------
             1                  0.000000E+00            PASS
             2                  0.000000E+00            PASS
             3                  0.000000E+00            PASS
             4                  0.000000E+00            PASS
             5                  0.000000E+00            PASS
             6                  9.600000E+08            FAIL

    SOME POSSIBLE REASONS MAY LEAD TO THE FAILURE:
      1. MULTIPOINT CONSTRAINT EQUATIONS WHICH DO NOT SATISFY RIGID-BODY MOTION;
      2. RBE3 ELEMENTS FOR WHICH THE INDEPENDENT DEGREE-OF-FREEDOM CANNOT DESCRIBE
         ALL POSSIBLE RIGID-BODY MOTIONS.
```

OUTPUT FORM ground_check_1c (Cont.)

```
0     PERFORM MODEL CHECKS                                                            SUBCASE 1

                              R E A L   E I G E N V A L U E S
     MODE    EXTRACTION       EIGENVALUE           RADIANS            CYCLES        GENERALIZED        GENERALIZED
     NO.       ORDER                                                                  MASS             STIFFNESS
        1         1          4.709041E+08       2.170032E+04      3.453714E+03      1.000000E+00      4.709041E+08
        2         2          9.503416E+08       3.082761E+04      4.906367E+03      1.000000E+00      9.503416E+08
        3         3          1.233946E+09       3.512757E+04      5.590726E+03      1.000000E+00      1.233946E+09
        4         4          8.335352E+09       9.129815E+04      1.453055E+04      1.000000E+00      8.335352E+09
        5         5          1.786391E+10       1.336559E+05      2.127200E+04      1.000000E+00      1.786391E+10
1     CANTILEVER BEAM WITH 8 CBAR, AND 1 MPC                      MARCH   5, 2012  MSC.NASTRAN 11/25/11    PAGE    13
```

MSC Software

Discussion of ground_check_1c Results

- At the G-set, the structural matrices pas the rigid-body tests, since the CELAS2 which caused the problem in ground_check_1b has been removed.

- Matrix KNN fails the rigid-body test due to the incorrectly-specified MPC equation.   This is indicated by the large term in the CHKKNN matrix at DOF 6.

- By looking at the REACNCOL matrix – this matrix represents the forces (normalized to a maximum of 1.0) preventing the model from moving as a rigid body. The 6th column contains terms for GRID points 1 and 3, indicating that a modeling error exists in that area.

- This is the location of MPC (NOTE – since the test is performed on the N-set, GRID 2 DOF 2 no longer exists, since it is in the M-set and has been removed).

MSC Software

Ground_check_2a – MODEL ERROR

- **Question: What is wrong with this rod model?**
  - Input File: ground_check_2a.dat

```
SOL 101
CEND
TITLE = Groundcheck for an Inclined Rod
ECHO = SORT
GROUNDCHECK(GRID=1, SET=(G,N+AUTOSPC))=YES
SUBCASE 1
    SUBTITLE=Default
    SPC  =  1
    LOAD = 1
    DISPLACEMENT(SORT1,REAL)=ALL
    SPCFORCES(SORT1,REAL)=ALL
    STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
BEGIN BULK
MAT1       1        1.+7
PROD       1        1        1.
CROD       1        1        1        2
CROD       2        1        2        3
FORCE, 1,  3, 0, 1000., 0.866025, 0.5, 0.
GRID       1                 0.       0.        0.
GRID       2                 .866025 0.5       0.
GRID       3                 1.732051 1.        0.
PARAM      AUTOSPC  YES
PARAM      GRDPNT   0
SPC1       1        123456   1
SPC1       1        3456     2        3
ENDDATA
```
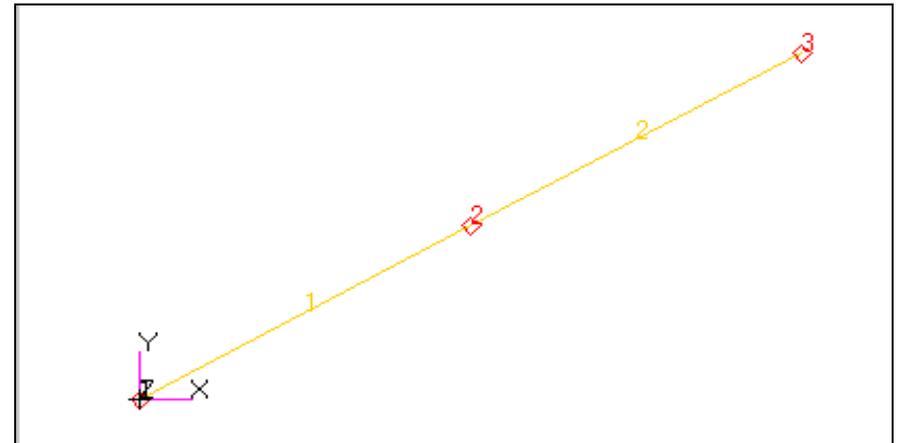
HOW TO AVOID SERIOUS MODELING MISTAKES

- Take the time to understand the structure and how it behaves under load. Perform hand calculation or use a simple model first.

- Take the time to understand MSC NASTRAN (particularly the elements). Run small samples each time you try something new.

- Use independent checks (if available).

- Estimate the cost (labor and computer costs) before you start.

MSC Software

CHECK FOR BAD MODES

- **Identify your modes using one or more of the following:**
  - Plot your eigenvectors (either using the MSC NASTRAN plotter or MSC PATRAN) and identify them
  - Try setting NORM=MAX on EIGRL entry and look at modal masses. Small values may indicate singularities or local modes (not recommended).
  - Use Case Control Commands EKE, and MEFFMASS to print kinetic energy and modal effective mass .

- **Watch for warnings on orthogonality checks**

- **Look for extraneous low frequency modes – these often indicate incorrect modeling (for example plate elements without MID2 on the PSHELL entry)**

SAMPLE OF SHRINK PLOTS

- **Stiffened Plate with Error in Modeling**
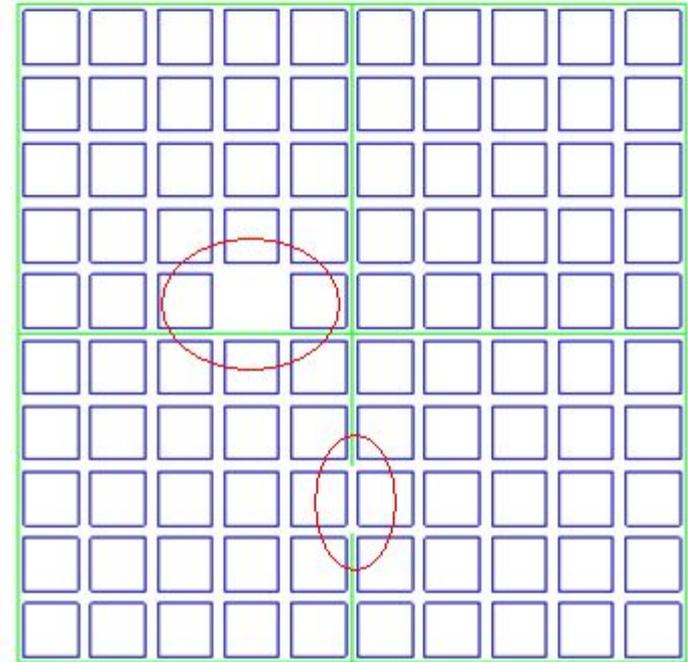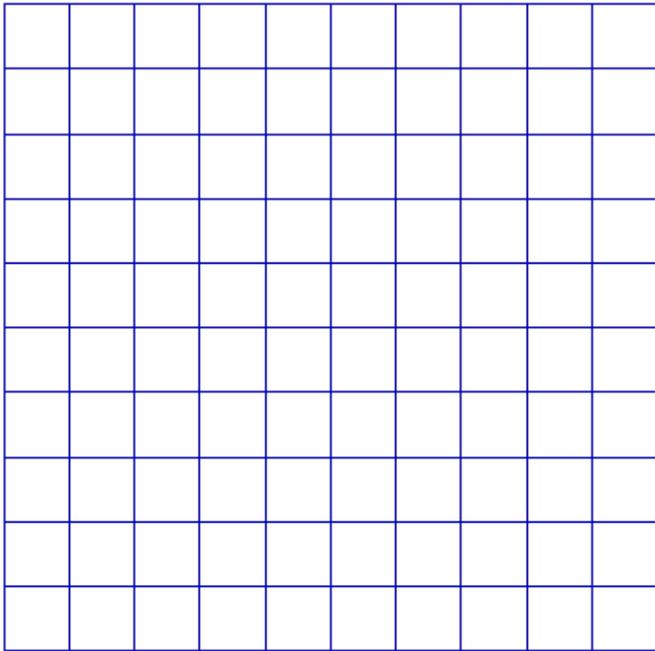
SOME RECOMMENDATIONS

- **Understand the important things BEFORE you get into trouble!!!**
  - Understand your structure and how you expect it to perform
  - Understand your loading
  - Understand your model
  - Understand how to use the program
  - Understand the limitations of the method
  - Use simple sample problems (preferably with known solutions) to understand the MSC Nastran solution.

- **ALWAYS perform a static solution first, then progress to the more complicated solutions.**

MORE TRICKS

ELEMENT FORCE DISTRIBUTIONS FOR UNIFORM STRESS

- **Pressure loadings on elements are replaced by forces at the corners.**

- **For a simple element, such as the QUAD4 or TRIA3, the force is simply the total force divided by the number of connecting GRID points.**

- **For higher order elements, the forces are found based on the element formulation, and may not agree with user's physical intuition or common sense.**

- **Example**
  - Grid point loads equivalent to a uniform pressure
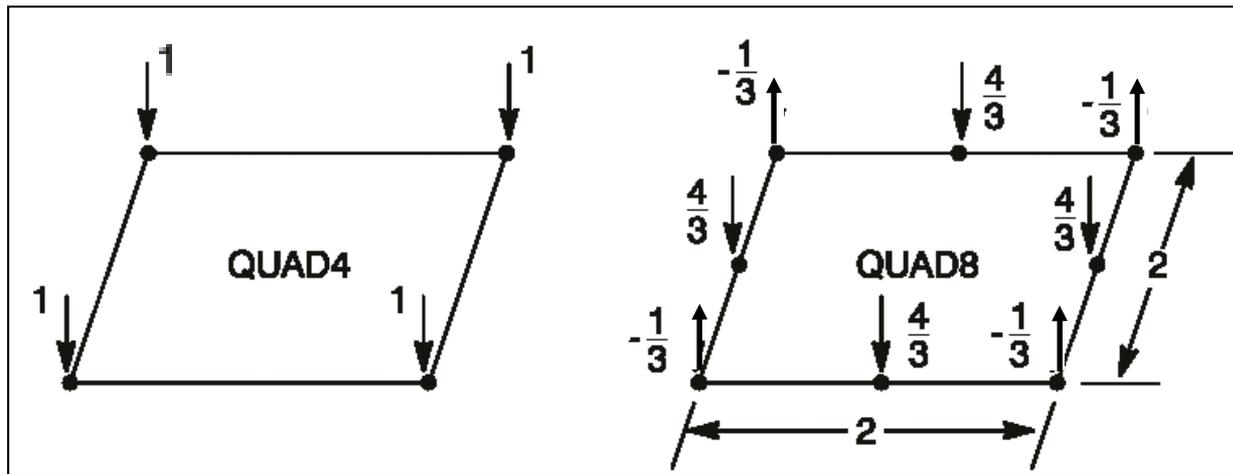
ELEMENT FORCE DISTRIBUTIONS FOR UNIFORM STRESS (Cont.)

- **Different element types have different order polynomials used for their displacement fields.**

- **As a result, it is possible that local discontinuities will occur when you mix different element types.**

MSC Software
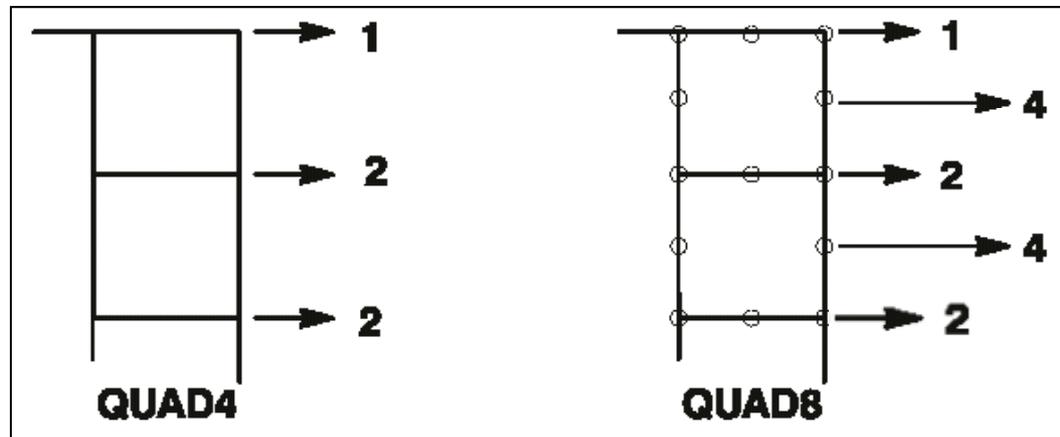
ELEMENT FORCE DISTRIBUTIONS FOR UNIFORM STRESS (Cont.)

- **Method of Testing**
  - Put SPCs on boundary, to induce uniform stress.
  - Measure SPCF output.
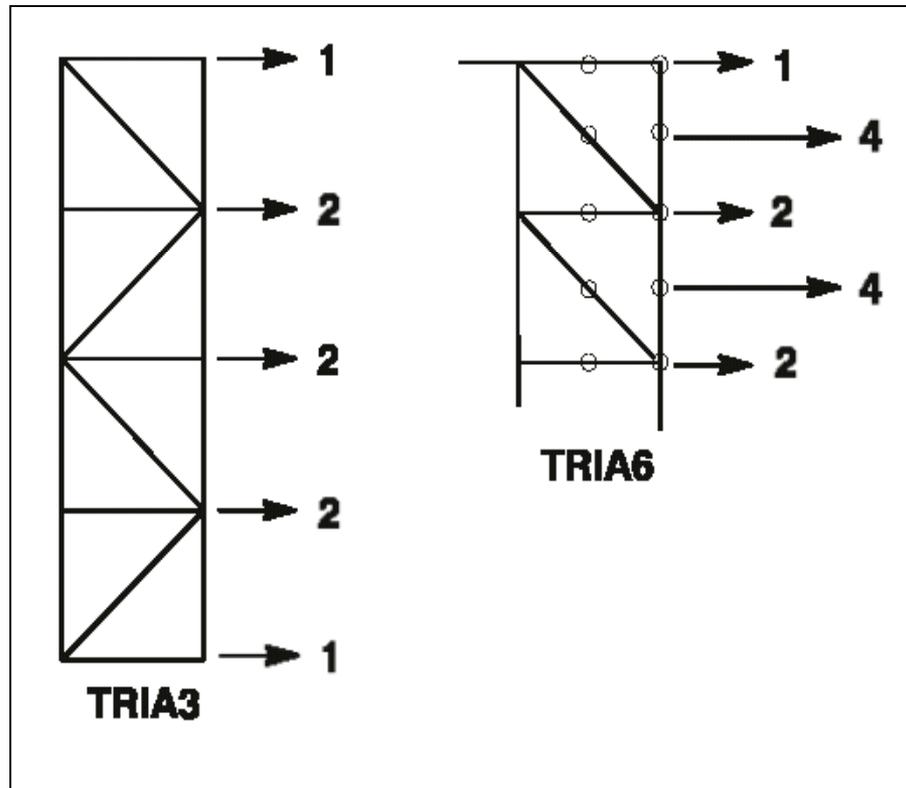
DISPLACEMENT COORDINATE SYSTEMS

- When you define a GRID entry in MSC NASTRAN, there are two fields used to select coordinate systems.

Format:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	

Example:

GRID	2	3	1.0	-2.0	3.0		316		

- CP is the "position" coordinate system, or the system in which X1, X2, and X3 are measured.

- CD is the "displacement" coordinate system. This defines the coordinate system which is used to measure the displacements at the GRID point. All constraints, MPCs, SPCs, and the BAR/BEAM offsets and orientation vector use this coordinate system.

- Coordinate systems may be rectangular (X,Y,Z), cylindrical (R, θ, Z), or

  spherical (R, θ, Φ).

OFFSETS ON BARS, BEAMS AND PLATES
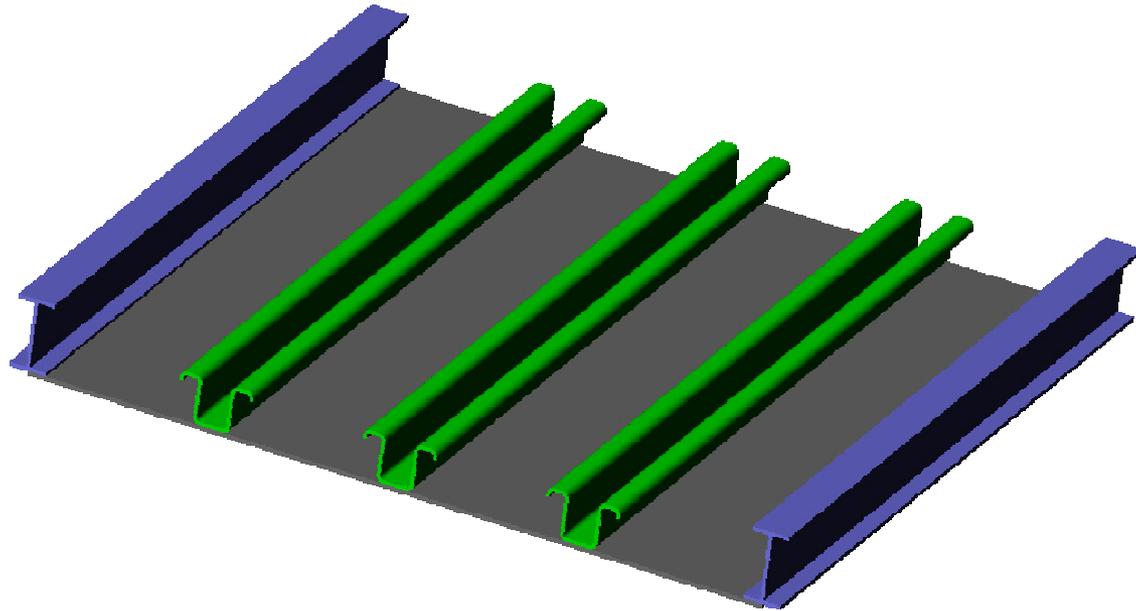
OFFSETS ON BARS, BEAMS AND PLATES (Cont.)

- **GRID points connecting BAR/BEAM elements may NOT lie on the beam neutral axis.**

- **GRID points connecting plate elements may NOT lie on the plate mid-surface.**

- **Shear center of Beam sections may NOT lie on the beam neutral axis.**

MSC Software

HOW ARE OFFSETS IMPLEMENTED ON BARS AND BEAMS?

- **Bar and Beam elements may be offset from the connecting GRID points.**

- **The offsets are entered on the continuation of the CBAR or CBEAM entry and <u>are in the displacement coordinate system of the GRID points</u>**

- **<u>Offset Bar Sample – Axial Load</u>**
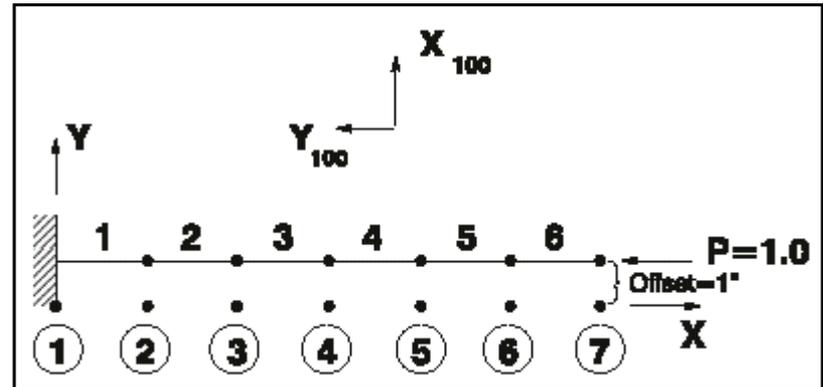
- **Let us look at this problem two different ways**
  - **Case 1**: GRID points using the basic coordinate system as the displacement coordinate system.
  - **Case 2**: Define, and Use a coordinate system 100 as the displacement coordinate system on the GRID points

HOW ARE OFFSETS IMPLEMENTED ON BARS AND BEAMS? (Cont.)

- **First problem – using the basic coordinate system**

```
ID OFFSET, BAR MODEL
SOL 101
CEND
TITLE = TEST OF OFFSET BAR MODEL – FILE offbar.dat
SUBT = SHOW WHERE ELEMENT FORCE OUTPUT IS
LOAD = 1
DISP = ALL
ELFORCE = ALL
GPFORCE = ALL
set 999 – 7
oload = 999
BEGIN BULK
$
$ DEFINE MODEL
$
GRID   1        0.    0.    0.        123456
GRID   2        2.    0.    0.
GRID   3        4.    0.    0.
GRID   4        6.    0.    0.
GRID   5        8.    0.    0.
GRID   6        10.   0.    0.
GRID   7        12.   0.    0.
```

```
$ OFFSET 1 UNIT IN THE Y DIRECTION – IN GRID
OUTPUT COORD
CBAR  1    1    1    2    0.    1.    0.        +CB1
+CB1            0.    1.    0.    0.    1.    0.
CBAR  2    1    2    3    0.    1.    0.        +CB2
+CB2            0.    1.    0.    0.    1.    0.
CBAR  3    1    3    4    0.    1.    0.        +CB3
+CB3            0.    1.    0.    0.    1.    0.
CBAR  4    1    4    5    0.    1.    0.        +CB4
+CB4            0.    1.    0.    0.    1.    0.
CBAR  5    1    5    6    0.    1.    0.        +CB5
+CB5            0.    1.    0.    0.    1.    0.
CBAR  6    1    6    7    0.    1.    0.        +CB6
+CB6            0.    1.    0.    0.    1.    0.
$
PBAR  1    1    1.    12.   12.   24.
MAT1  1    30.+6         .3
pload1        1    6    FX    FR    1.    -1.
ENDDATA
```

MSC Software®

SAMPLE OF OFFSET BAR AXIAL LOAD

```
        ID OFFSET, BAR MODEL
        SOL 101
        CEND
1   TEST OF OFFSET BAR MODEL | FILE OFFBAR.DAT                        MARCH   5, 2012  MSC.NASTRAN 11/25/11    PAGE    3
      SHOW WHERE ELEMENT FORCE OUTPUT IS
0
0                                        C A S E    C O N T R O L    E C H O
             COMMAND
             COUNT
               1        TITLE = TEST OF OFFSET BAR MODEL | FILE OFFBAR.DAT
               2        SUBT = SHOW WHERE ELEMENT FORCE OUTPUT IS
               3        LOAD = 1
               4        DISP = ALL
               5        ELFORCE = ALL
               6        GPFORCE = ALL
               7        SET 999 | 7
               8        OLOAD = 999
               9        BEGIN BULK
1   TEST OF OFFSET BAR MODEL | FILE OFFBAR.DAT                        MARCH   5, 2012  MSC.NASTRAN 11/25/11    PAGE    4
      SHOW WHERE ELEMENT FORCE OUTPUT IS
0
                                      S O R T E D   B U L K   D A T A   E C H O
             ENTRY
             COUNT          .   1  ..   2  ..   3  ..   4  ..   5  ..   6  ..   7  ..   8  ..   9  ..  10  .
               1-      CBAR    1      1      1      2      0.     1.     0.                  +
               2-      +                     0.     1.     0.     0.     1.     0.
               3-      CBAR    2      1      2      3      0.     1.     0.                  +
               4-      +                     0.     1.     0.     0.     1.     0.
               5-      CBAR    3      1      3      4      0.     1.     0.     0.           +
               6-      +                     0.     1.     0.     0.     1.     0.
               7-      CBAR    4      1      4      5      0.     1.     0.                  +
               8-      +                     0.     1.     0.     0.     1.     0.
               9-      CBAR    5      1      5      6      0.     1.     0.                  +
              10-      +                     0.     1.     0.     0.     1.     0.
              11-      CBAR    6      1      6      7      0.     1.     0.                  +
              12-      +                     0.     1.     0.     0.     1.  123456  0.
              13-      GRID    1             0.     0.     0.
              14-      GRID    2             2.     0.     0.
              15-      GRID    3             4.     0.     0.
              16-      GRID    4             6.     0.     0.
              17-      GRID    5             8.     0.     0.
              18-      GRID    6            10.     0.     0.
              19-      GRID    7            12.     0.     0.
              20-      MAT1    1     3E+07         .3
              21-      PBAR    1      1      1.    12.    12.    24.
              22-      PLOAD1  1      6     FX     FR     1.    -1.
                       ENDDATA
                 TOTAL COUNT= 23
                       INPUT BULK DATA ENTRY COUNT = 28
```

SAMPLE OF OFFSET BAR AXIAL LOAD (Cont.)

```
1     TEST OF OFFSET BAR MODEL | FILE OFFBAR.DAT                    MARCH   5, 2012  MSC.NASTRAN 11/25/11    PAGE       5
      SHOW WHERE ELEMENT FORCE OUTPUT IS
0
                              M O D E L   S U M M A R Y           BULK = 0
                              ENTRY NAME          NUMBER OF ENTRIES
                              ----------          -----------------
                                  CBAR                        6
                                  GRID                        7
                                  MAT1                        1
                                  PBAR                        1
                                PLOAD1                        1

  ^^^
  ^^^  >>> IFP OPERATIONS COMPLETE <<<
  ^^^
```

MSC Software

SAMPLE OF OFFSET BAR AXIAL LOAD (Cont.)

```
 *** USER INFORMATION MESSAGE 7310 (VECPRN)
     ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
     RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.
0                                                        OLOAD     RESULTANT
   SUBCASE/    LOAD
   DAREA ID    TYPE       T1            T2            T3            R1            R2            R3
0        1      FX   -1.000000E+00     ----          ----          ----      0.000000E+00  0.000000E+00
                FY       ----      0.000000E+00      ----      0.000000E+00      ----      0.000000E+00
                FZ       ----          ----      0.000000E+00  0.000000E+00  0.000000E+00      ----
                MX       ----          ----          ----      0.000000E+00      ----          ----
                MY       ----          ----          ----          ----      0.000000E+00      ----
                MZ       ----          ----          ----          ----          ----      1.000000E+00
             TOTALS  -1.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  1.000000E+00
1     TEST OF OFFSET BAR MODEL | FILE OFFBAR.DAT                    MARCH   5, 2012  MSC.NASTRAN 11/25/11   PAGE    7
         SHOW WHERE ELEMENT FORCE OUTPUT IS
```

```
 *** USER INFORMATION MESSAGE 7310 (VECPRN)
     ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
     RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.
0                                                        SPCFORCE  RESULTANT
   SUBCASE/    LOAD
   DAREA ID    TYPE       T1            T2            T3            R1            R2            R3
0        1      FX    1.000000E+00     ----          ----          ----      0.000000E+00  0.000000E+00
                FY       ----      0.000000E+00      ----      0.000000E+00      ----      0.000000E+00
                FZ       ----          ----      0.000000E+00  0.000000E+00  0.000000E+00      ----
                MX       ----          ----          ----      0.000000E+00      ----          ----
                MY       ----          ----          ----          ----      0.000000E+00      ----
                MZ       ----          ----          ----          ----          ----     -1.000000E+00
             TOTALS   1.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00  0.000000E+00 -1.000000E+00
1     TEST OF OFFSET BAR MODEL | FILE OFFBAR.DAT                    MARCH   5, 2012  MSC.NASTRAN 11/25/11   PAGE    9
         SHOW WHERE ELEMENT FORCE OUTPUT IS
```

SAMPLE OF OFFSET BAR AXIAL LOAD (Cont.)

```
                                    L O A D   V E C T O R

      POINT ID.   TYPE        T1              T2            T3           R1           R2           R3
          7        G     -1.000000E+00    0.0          0.0          0.0          0.0        1.000000E+00
1    TEST OF OFFSET BAR MODEL | FILE OFFBAR.DAT                    MARCH   5, 2012  MSC.NASTRAN 11/25/11   PAGE   12
0      SHOW WHERE ELEMENT FORCE OUTPUT IS


0                        F O R C E   D I S T R I B U T I O N   I N   B A R   E L E M E N T S      ( C B A R )
0    ELEMENT  STATION        BEND-MOMENT                    SHEAR FORCE                  AXIAL
       ID.    (PCT)     PLANE 1       PLANE 2          PLANE 1       PLANE 2            FORCE            TORQUE
          1   0.000    0.0           0.0              0.0           0.0           -1.000000E+00        0.0
          1   1.000    0.0           0.0              0.0           0.0           -1.000000E+00        0.0
          2   0.000    0.0           0.0              0.0           0.0           -1.000000E+00        0.0
          2   1.000    0.0           0.0              0.0           0.0           -1.000000E+00        0.0
          3   0.000    0.0           0.0              0.0           0.0           -1.000000E+00        0.0
          3   1.000    0.0           0.0              0.0           0.0           -1.000000E+00        0.0
          4   0.000    0.0           0.0              0.0           0.0           -1.000000E+00        0.0
          4   1.000    0.0           0.0              0.0           0.0           -1.000000E+00        0.0
          5   0.000    0.0           0.0              0.0           0.0           -1.000000E+00        0.0
          5   1.000    0.0           0.0              0.0           0.0           -1.000000E+00        0.0
          6   0.000    0.0           0.0              0.0           0.0           -1.000000E+00        0.0
          6   1.000    0.0           0.0              0.0           0.0            0.0                 0.0
1    TEST OF OFFSET BAR MODEL | FILE OFFBAR.DAT                    MARCH   5, 2012  MSC.NASTRAN 11/25/11   PAGE   13
0      SHOW WHERE ELEMENT FORCE OUTPUT IS
0
```

MSC Software

SAMPLE OF OFFSET BAR AXIAL LOAD (Cont.)

```
                         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    1.000000E+00     0.0             0.0             0.0             0.0            -1.000000E+00
           1           1     BAR        -1.000000E+00     0.0             0.0             0.0             0.0             1.000000E+00
           1                 *TOTALS*    0.0              0.0             0.0             0.0             0.0             0.0
0          2           1     BAR         1.000000E+00     0.0             0.0             0.0             0.0            -1.000000E+00
           2           2     BAR        -1.000000E+00     0.0             0.0             0.0             0.0             1.000000E+00
           2                 *TOTALS*    0.0              0.0             0.0             0.0             0.0             0.0
0          3           2     BAR         1.000000E+00     0.0             0.0             0.0             0.0            -1.000000E+00
           3           3     BAR        -1.000000E+00     0.0             0.0             0.0             0.0             1.000000E+00
           3                 *TOTALS*    2.220446E-16     0.0             0.0             0.0             0.0            -2.220446E-16
0          4           3     BAR         1.000000E+00     0.0             0.0             0.0             0.0            -1.000000E+00
           4           4     BAR        -1.000000E+00     0.0             0.0             0.0             0.0             1.000000E+00
           4                 *TOTALS*   -4.440892E-16     0.0             0.0             0.0             0.0             4.440892E-16
0          5           4     BAR         1.000000E+00     0.0             0.0             0.0             0.0            -1.000000E+00
           5           5     BAR        -1.000000E+00     0.0             0.0             0.0             0.0             1.000000E+00
           5                 *TOTALS*    1.332268E-15     0.0             0.0             0.0             0.0            -1.332268E-15
0          6           5     BAR         1.000000E+00     0.0             0.0             0.0             0.0            -1.000000E+00
           6           6     BAR        -1.000000E+00     0.0             0.0             0.0             0.0             1.000000E+00
           6                 *TOTALS*   -8.881784E-16     0.0             0.0             0.0             0.0             8.881784E-16
0          7                 APP-LOAD   -1.000000E+00     0.0             0.0             0.0             0.0             1.000000E+00
           7           6     BAR         1.000000E+00     0.0             0.0             0.0             0.0            -1.000000E+00
           7                 *TOTALS*    0.0              0.0             0.0             0.0             0.0             0.0
1   TEST OF OFFSET BAR MODEL | FILE OFFBAR.DAT                            MARCH   5, 2012   MSC.NASTRAN 11/25/11   PAGE    14
    SHOW WHERE ELEMENT FORCE OUTPUT IS
```

MSC Software

HOW ARE OFFSETS IMPLEMENTED ON BARS AND BEAMS? (Cont.)

- **The following summarizes the preceding output**

- **OLOAD RESULTANT – this is the summation of all applied loads about PARAM, GRDPNT (about the BASIC origin if GRDPNT is not specified) in the basic coordinate system.**

  - In this case, it verifies that the applied load was –1.0 units in the X–direction and is offset by 1.0 units (M_z = 1.0 = the load multiplied by the offset)

- **ALWAYS VERIFY THAT THE OLOAD RESULTANT IS CORRECT!!!**

- **SPCFORCE RESULTANT – Similar output for the reaction forces. This should be equal and opposite to the OLOAD RESULTANT. If not, Use GROUNDCHECK Case Control Command.**

- **DISPLACEMENT VECTOR – displacement at the GRID point in the displacement coordinates.**

  - These displacements are in the BASIC coordinate system and verify that the loading is an axial load (there is only axial displacement). Notice that the OLOAD RESULTANT correctly shows a resultant moment about the origin, but the element is offset by 1.0 units.

MSC Software

HOW ARE OFFSETS IMPLEMENTED ON BARS AND BEAMS? (Cont.)

- **LOAD VECTOR – is the loading at the GRID points in the displacement coordinate system.**
  - In this case, we can now see both the axial component (–1.0 in the X–direction) and the moment due to the offset.

- **FORCE DISTRIBUTION IN BAR ELEMENTS – forces in the elements (in the element coordinate system)**
  - Once again, this verifies that the loading is an axial loading only, with the element output showing only axial force.
  - NOTE – the PLOAD1 is a loading applied on the element. In MSC NASTRAN, the element is considered to begin and end at the offset locations. Therefore, the applied load is along the axis of the element.

- **GRID POINT FORCE BALANCE – Where is the moment?**
  - Remember that this output is in the displacement coordinate system of the grids. Each element is offset from the GRID points by 1.0 units (internally a rigid offset), and the elements transfer the axial force and the resulting moment to the GRID points.

– **Case 2**: GRID points using the coordinate system 100 as the displacement coordinate system.

```
ID OFFSET,  BAR MODEL
SOL 101
CEND
TITLE = TEST OF OFFSET BAR MODEL – FILE offbar100.dat
SUBT = USE DISPLACEMENT COORDINATE SYSTEM
LOAD = 1
DISP = ALL
ELFORCE = ALL
GPFORCE = ALL
set 999 – 7
oload = 999
BEGIN BULK
$
CORD2R  100        0.    0.    0.    0.    0.    1.
+000001
++0000010.    1.    0.                            +000002
GRID   1          0.    0.    0.    100    123456
GRID   2          2.    0.    0.    100
GRID   3          4.    0.    0.    100
GRID   4          6.    0.    0.    100
GRID   5          8.    0.    0.    100
GRID   6         10.    0.    0.    100
GRID   7         12.    0.    0.    100
```

```
$ OFFSET 1 UNIT IN THE BASIC Y DIRECTION –
$ THIS IS IN THE +X IN GRID OUTPUT COORD
CBAR  1    1    1    2    1.    0.    0.        +CB1
+CB1            1.    0.    0.    1.    0.    0.
CBAR  2    1    2    3    1.    0.    0.        +CB2
+CB2            1.    0.    0.    1.    0.    0.
CBAR  3    1    3    4    1.    0.    0.        +CB3
+CB3            1.    0.    0.    1.    0.    0.
CBAR  4    1    4    5    1.    0.    0.        +CB4
+CB4            1.    0.    0.    1.    0.    0.
CBAR  5    1    5    6    1.    0.    0.        +CB5
+CB5            1.    0.    0.    1.    0.    0.
CBAR  6    1    6    7    1.    0.    0.        +CB6
+CB6            1.    0.    0.    1.    0.    0.
PBAR  1    1    1.    12.    12.    24.
MAT1  1    30.+6       .3
pload1         1    6    FX    FR    1.    -1.
ENDDATA
```

HOW ARE OFFSETS IMPLEMENTED ON BARS AND BEAMS? (Cont.)

- **Since coordinate system 100 is the displacement coordinate system, the element offsets will be in the x-direction.**

- **NOTE – For this example, orientation vectors of**

  < 1, 0, 0> are used for the BAR elements. Although this appears to be parallel to the elements, it is not, because the orientation vector of BAR and BEAM elements is defined using the displacement coordinate system of the first GRID it connects to.

MSC Software®

OFFSET BAR USING DISPLACEMENT COORDINATES

```
0                                        S O R T E D   B U L K   D A T A   E C H O
          ENTRY
          COUNT           .    1  . .   2   . .   3   . .   4   . .   5   . .   6   . .   7   . .   8   . .   9   . .  10   .
            1-     CBAR    1        1        1        2        1.       0.       0.                         +
            2-     +                         1.       0.       0.       1.       0.       0.
            3-     CBAR    2        1        2        3        1.       0.       0.                         +
            4-     +                         1.       0.       0.       1.       0.       0.
            5-     CBAR    3        1        3        4        1.       0.       0.                         +
            6-     +                         1.       0.       0.       1.       0.
            7-     CBAR    4        1        4        5        1.       0.       0.                         +
            8-     +                         1.       0.       0.       1.       0.       0.
            9-     CBAR    5        1        5        6        1.       0.       0.                         +
           10-     +                         1.       0.       0.       1.       0.       0.
           11-     CBAR    6        1        6        7        1.       0.       0.                         +
           12-     +                         1.       0.       0.       1.       0.       0.
           13-     CORD2R  100                0.       0.       0.       0.       0.       1.               +
           14-     +       0.       1.       0.
           15-     GRID    1                 0.       0.       0.      100     123456
           16-     GRID    2                 2.       0.       0.      100
           17-     GRID    3                 4.       0.       0.      100
           18-     GRID    4                 6.       0.       0.      100
           19-     GRID    5                 8.       0.       0.      100
           20-     GRID    6                10.       0.       0.      100
           21-     GRID    7                12.       0.       0.      100
           22-     MAT1    1       3E+07             .3
           23-     PBAR    1        1        1.      12.      12.      24.
           24-     PLOAD1  1        6       FX       FR        1.      -1.
                   ENDDATA
           TOTAL COUNT= 25
                   INPUT BULK DATA ENTRY COUNT = 30
1  TEST OF OFFSET BAR MODEL | FILE OFFBAR100.DAT                        MARCH   5, 2012  MSC.NASTRAN 11/25/11   PAGE     5
   USE DISPLACEMENT COORDINATE SYSTEM
0
                           M O D E L   S U M M A R Y           BULK = 0
                   ENTRY NAME           NUMBER OF ENTRIES
                   ----------           -----------------
                      CBAR                      6
                     CORD2R                     1
                      GRID                      7
                      MAT1                      1
                      PBAR                      1
                     PLOAD1                     1
```

MSC Software

OFFSET BAR USING DISPLACEMENT COORDINATES (Cont.)

```
*** USER INFORMATION MESSAGE 7310 (VECPRN)
    ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
    RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.
0                                                    OLOAD      RESULTANT
  SUBCASE/    LOAD
  DAREA ID    TYPE       T1             T2             T3             R1             R2             R3
0        1     FX   -1.000000E+00     ----           ----           ----        0.000000E+00   0.000000E+00
               FY       ----      0.000000E+00       ----        0.000000E+00      ----        0.000000E+00
               FZ       ----          ----       0.000000E+00   0.000000E+00   0.000000E+00      ----
               MX       ----          ----           ----        0.000000E+00      ----           ----
               MY       ----          ----           ----           ----        0.000000E+00      ----
               MZ       ----          ----           ----           ----           ----        1.000000E+00
             TOTALS -1.000000E+00  0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   1.000000E+00
1   TEST OF OFFSET BAR MODEL | FILE OFFBAR100.DAT                    MARCH   5, 2012  MSC.NASTRAN 11/25/11   PAGE      7
      USE DISPLACEMENT COORDINATE SYSTEM
```

```
*** USER INFORMATION MESSAGE 7310 (VECPRN)
    ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM WILL BE USED AS REFERENCE LOCATION.
    RESULTANTS ABOUT ORIGIN OF SUPERELEMENT BASIC COORDINATE SYSTEM IN SUPERELEMENT BASIC SYSTEM COORDINATES.
0                                                    SPCFORCE   RESULTANT
  SUBCASE/    LOAD
  DAREA ID    TYPE       T1             T2             T3             R1             R2             R3
0        1     FX    1.000000E+00     ----           ----           ----        0.000000E+00   0.000000E+00
               FY       ----      0.000000E+00       ----        0.000000E+00      ----        0.000000E+00
               FZ       ----          ----       0.000000E+00   0.000000E+00   0.000000E+00      ----
               MX       ----          ----           ----        0.000000E+00      ----           ----
               MY       ----          ----           ----           ----        0.000000E+00      ----
               MZ       ----          ----           ----           ----           ----       -1.000000E+00
             TOTALS  1.000000E+00  0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00  -1.000000E+00
1   TEST OF OFFSET BAR MODEL | FILE OFFBAR100.DAT                    MARCH   5, 2012  MSC.NASTRAN 11/25/11   PAGE      9
      USE DISPLACEMENT COORDINATE SYSTEM
```

MSC Software

# OFFSET BAR USING DISPLACEMENT COORDINATES (Cont.)

```
                        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              0.0              0.0              0.0
          2        G          0.0          6.666667E-08         0.0              0.0              0.0              0.0
          3        G          0.0          1.333333E-07         0.0              0.0              0.0              0.0
          4        G          0.0          2.000000E-07         0.0              0.0              0.0              0.0
          5        G          0.0          2.666667E-07         0.0              0.0              0.0              0.0
          6        G          0.0          3.333333E-07         0.0              0.0              0.0              0.0
          7        G          0.0          4.000000E-07         0.0              0.0              0.0              0.0
1     TEST OF OFFSET BAR MODEL | FILE OFFBAR100.DAT                      MARCH    5, 2012  MSC.NASTRAN 11/25/11    PAGE    11
      USE DISPLACEMENT COORDINATE SYSTEM
```

```
                    F O R C E   D I S T R I B U T I O N   I N   B A R   E L E M E N T S        ( C B A R )
0   ELEMENT  STATION          BEND-MOMENT                    SHEAR FORCE                   AXIAL
     ID.     (PCT)     PLANE 1        PLANE 2        PLANE 1        PLANE 2          FORCE              TORQUE
       1     0.000    0.0            0.0            0.0            0.0            -1.000000E+00         0.0
       1     1.000    0.0            0.0            0.0            0.0            -1.000000E+00         0.0
       2     0.000    0.0            0.0            0.0            0.0            -1.000000E+00         0.0
       2     1.000    0.0            0.0            0.0            0.0            -1.000000E+00         0.0
       3     0.000    0.0            0.0            0.0            0.0            -1.000000E+00         0.0
       3     1.000    0.0            0.0            0.0            0.0            -1.000000E+00         0.0
       4     0.000    0.0            0.0            0.0            0.0            -1.000000E+00         0.0
       4     1.000    0.0            0.0            0.0            0.0            -1.000000E+00         0.0
       5     0.000    0.0            0.0            0.0            0.0            -1.000000E+00         0.0
       5     1.000    0.0            0.0            0.0            0.0            -1.000000E+00         0.0
       6     0.000    0.0            0.0            0.0            0.0            -1.000000E+00         0.0
       6     1.000    0.0            0.0            0.0            0.0             0.0                  0.0
1     TEST OF OFFSET BAR MODEL | FILE OFFBAR100.DAT                      MARCH    5, 2012  MSC.NASTRAN 11/25/11    PAGE    13
      USE DISPLACEMENT COORDINATE SYSTEM
```

OFFSET BAR USING DISPLACEMENT COORDINATES (Cont.)

```
                          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      0.0      -1.000000E+00    0.0          0.0          0.0      -1.000000E+00
        1          1        BAR           0.0       1.000000E+00    0.0          0.0          0.0       1.000000E+00
        1                   *TOTALS*      0.0       0.0             0.0          0.0          0.0       0.0
0       2          1        BAR           0.0      -1.000000E+00    0.0          0.0          0.0      -1.000000E+00
        2          2        BAR           0.0       1.000000E+00    0.0          0.0          0.0       1.000000E+00
        2                   *TOTALS*      0.0       0.0             0.0          0.0          0.0       0.0
0       3          2        BAR           0.0      -1.000000E+00    0.0          0.0          0.0      -1.000000E+00
        3          3        BAR           0.0       1.000000E+00    0.0          0.0          0.0       1.000000E+00
        3                   *TOTALS*      0.0      -2.220446E-16    0.0          0.0          0.0      -2.220446E-16
0       4          3        BAR           0.0      -1.000000E+00    0.0          0.0          0.0      -1.000000E+00
        4          4        BAR           0.0       1.000000E+00    0.0          0.0          0.0       1.000000E+00
        4                   *TOTALS*      0.0       4.440892E-16    0.0          0.0          0.0       4.440892E-16
0       5          4        BAR           0.0      -1.000000E+00    0.0          0.0          0.0      -1.000000E+00
        5          5        BAR           0.0       1.000000E+00    0.0          0.0          0.0       1.000000E+00
        5                   *TOTALS*      0.0      -1.332268E-15    0.0          0.0          0.0      -1.332268E-15
0       6          5        BAR           0.0      -1.000000E+00    0.0          0.0          0.0      -1.000000E+00
        6          6        BAR           0.0       1.000000E+00    0.0          0.0          0.0       1.000000E+00
        6                   *TOTALS*      0.0       8.881784E-16    0.0          0.0          0.0       8.881784E-16
0       7                   APP-LOAD      0.0       1.000000E+00    0.0          0.0          0.0       1.000000E+00
        7          6        BAR           0.0      -1.000000E+00    0.0          0.0          0.0      -1.000000E+00
        7                   *TOTALS*      0.0       0.0             0.0          0.0          0.0       0.0
1    TEST OF OFFSET BAR MODEL | FILE OFFBAR100.DAT                        MARCH   5, 2012  MSC.NASTRAN 11/25/11    PAGE    14
     USE DISPLACEMENT COORDINATE SYSTEM
```

MSC Software

HOW ARE OFFSETS IMPLEMENTED ON BARS AND BEAMS?

- **The preceding output is similar to that obtained in the original run.**

- **The OLOAD RESULTANT, SPCFORCE RESULTANT, and ELEMENT FORCES are identical.**
  - The RESULTANTs are in BASIC, and the ELEMENT FORCES are in the element coordinate system.

- **The DISPLACEMENTs are identical, however, they are transformed into the displacement coordinate system (100), so they now show up as Y–translations.**

- **The LOAD VECTOR is also in the displacement coordinate system**
  - We still see the 1.0 unit load and the moment due to the offset, but now they are in system 100.

- **GRID POINT FORCE BALANCE – now we see the identical results as before, but they are in the displacement coordinate system (system 100).**

MSC Software

SHEAR CENTER – BEAMS AND BARS

- One of the most common modeling errors is to ignore the offset between the shear center and the neutral axis on a BAR or BEAM element when the cross-section is not doubly-symmetric.

- Most users use the BAR element. The BAR element assumes that the shear center and the neutral axis are coincident.

CBAR Element Geometry with Offsets.

CBAR Element Geometry without Offsets.
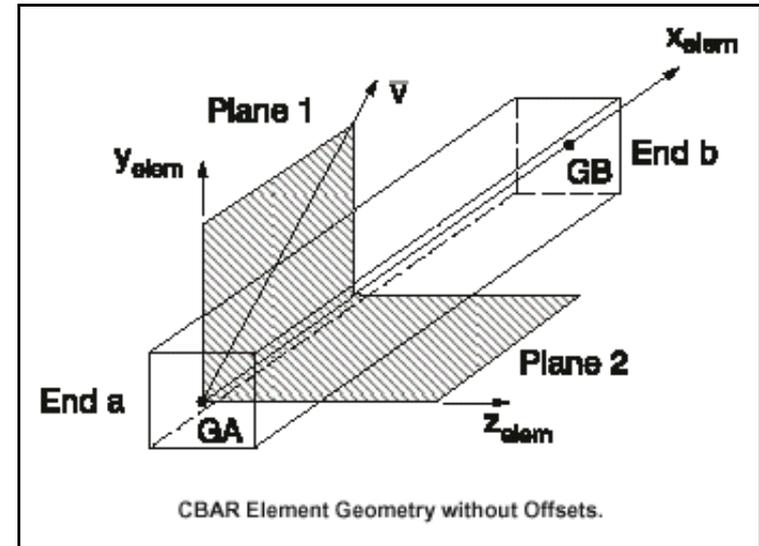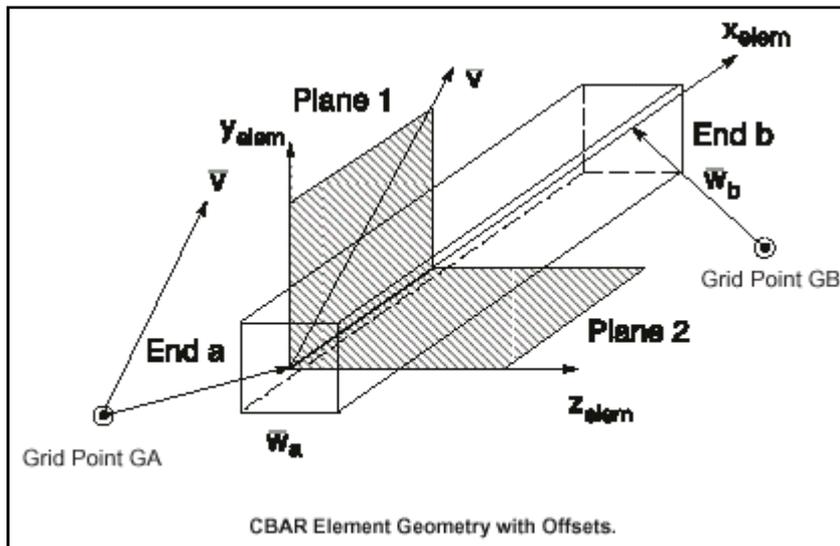
SHEAR CENTER – BEAMS AND BARS (Cont.)

- The BEAM element allows the shear center and the neutral axis to be offset from each other.

- The offset on the CBEAM entry is from the GRID points to the shear center. The PBEAM entries (N1A, N2A, N1B,N2B) are the offset from the shear center to the neutral axis.

- NOTE – the CI, DI, EI, and FI (stress recovery location) are relative to the shear center –they are NOT relative to the neutral axis.

- Element loads located at the shear center; not at the neutral axis.

- Once again, if the section is doubly-symmetric, or if there is no load effecting the offset, BAR will work fine in linear analysis. Otherwise use BEAM.

CBEAM Element Geometry System. (Non p-adaptive)

MSC Software
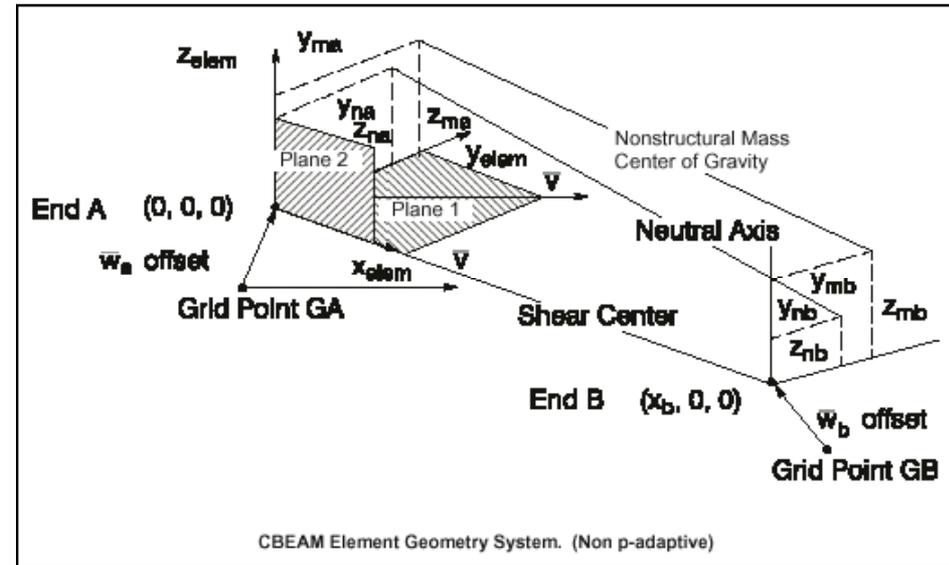
BEAM Vs. BAR

FEATURES	CBEAM	CBAR
Variable X-Section	YES	NO
Warping X-Section	YES	NO
Shear Relief	YES	NO
Shear Center Offset	YES	NO
Mass Moment of Inertia	YES	NO
Geometric Nonlinear	YES	NO
Plastic Hinges	YES	NO

- **In Summary,**
  - Bar assume symmetric cross section, beam does not.
  - Beam allows nonlinear analysis. Bar is linear only.
  - Bar doesn't allow for taper.

MSC Software®

HOW ARE OFFSETS IMPLEMENTED ON PLATES

- **GRID points connecting plate elements may NOT lie on the plate mid-surface.**

- **The plate offsets (ZOFFS) are entered on the CQUADi, and CTRIAi cards. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. Material matrices and stress fiber locations are relative to the reference plane.**

MSC Software

HANDY HINTS

- **Element strain energy (ESE) is a good tool for determining where to make changes to obtain maximum benefits.**

- **Example – two springs in series**

  Which is the best one to stiffen to reduce the tip deflection?
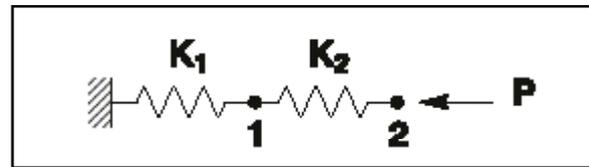
where K_1 = 10 $ESE = \dfrac{1}{2} K \Delta^2$

 K_2 = 1

 P   = 1 $for\ \mathbf{K}_1, ESE_1 = \dfrac{1}{2} K_1 \Delta_1^2 = \dfrac{1}{2} \dfrac{P^2}{K_1} = \dfrac{1}{20}$

$$where\ \Delta_1 = \dfrac{P}{K_1}$$

HANDY HINTS (Cont.)

$$for\ \mathbf{K_2}, ESE_2 = \frac{1}{2} K_2 (\Delta_2 - \Delta_1)^2 = \frac{1}{2} K_2 \left(\frac{P}{K_1} + \frac{P}{K_2} - \frac{P}{K_1} \right)^2$$

$$= \frac{1}{2} K_2 \left(\frac{P}{K_2} \right)^2 = \frac{1}{2}$$

$$where\ \Delta_1 = \frac{P}{K_1} \qquad \Delta_2 = \frac{P}{K_1} + \frac{P}{K_2}$$

Most of the ESE is in the smaller spring. Therefore, stiffening it is the most efficient way to reduce the tip deflection.

- **Normal modes analysis is similar in that changing the stiffness of the elements with the most ESE in a mode is often the most efficient way to shift the frequency of that mode.**

MSC Software

HANDY HINTS (Cont.)

- **Determine mesh size based on behavior – in areas of high stress variation, place extra elements**
  - $.61\sqrt{rt}$ from stiffener on a stiffened cylinder – normally 3⁺ elem

Plot of Moment versus Distance

for a Pressurized Stiffened Cylindrical Shell
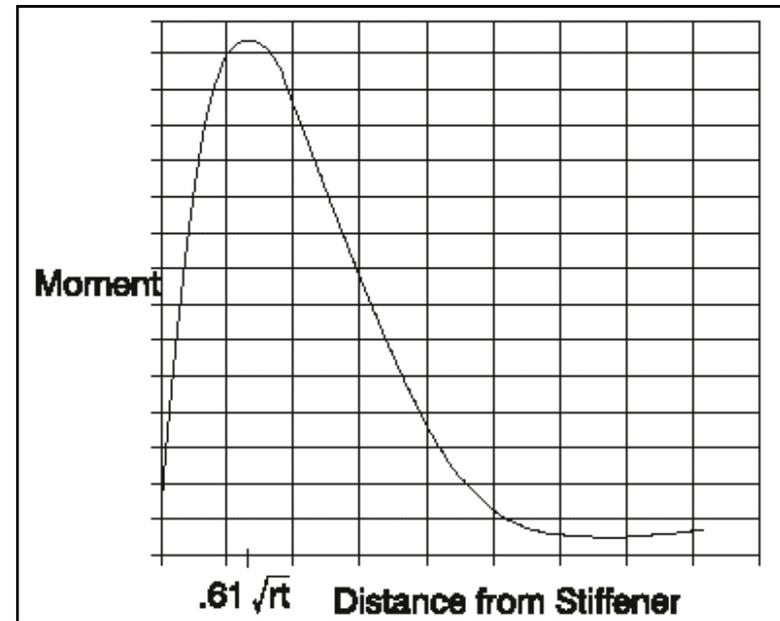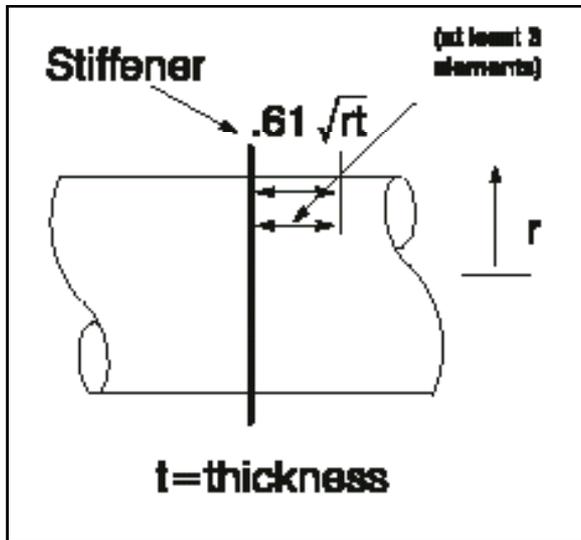
HANDY HINTS (Cont.)

- **Curved Shells**

    - Normally QUAD4 can cover a 5–10° angle on a cylindrical surface.

    - QUAD8 can cover a 10–25° angle.

    - QUAD8 without midside points uses linear interpolation. This is much less accurate than QUAD4.

    - A simple rule – for buckling and normal modes, there should be at least 5 GRID points per half sine wave of the deformed shape. .

MORE HANDY HINTS

- **Plates should have same orientation for stress output.**
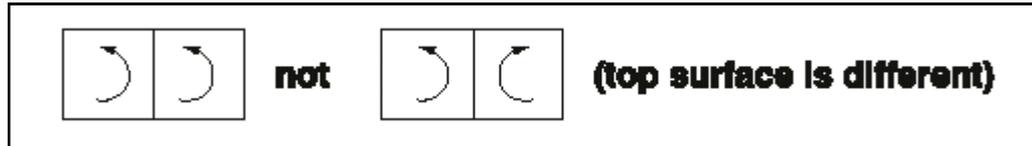
- **Plate output is usually in the element coordinate system.**

- **Pressure loads on plate are applied as point loads 1/4 at each corner on a QUAD4.**
  - The direction is based on plate orientation. A positive pressure acts in the positive element Z direction.

- **Plates at a corner are much softer than the actual structure**
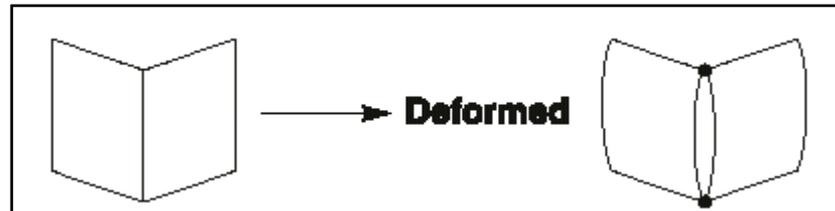
- **Each one does not have in-plane rotational stiffness. The model has stability, but does not properly transfer loads.**

MSC Software

COMPATIBILITIES

Plate Beam
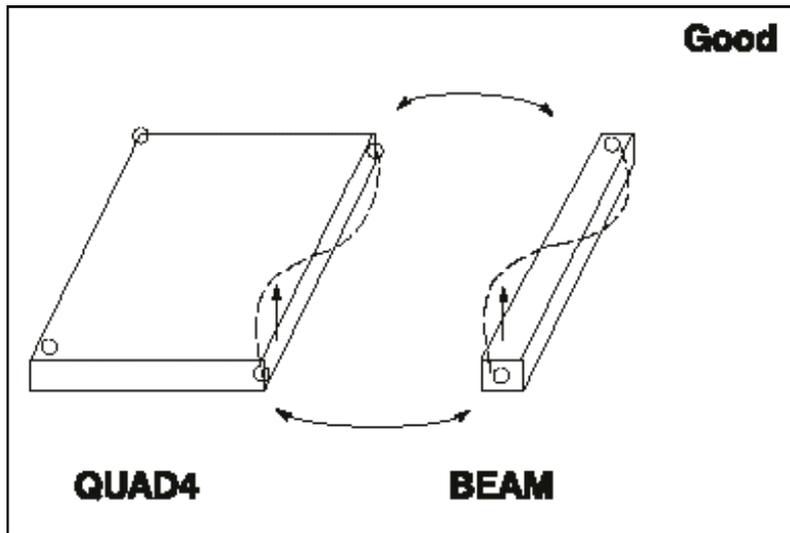
Membrane – Beam
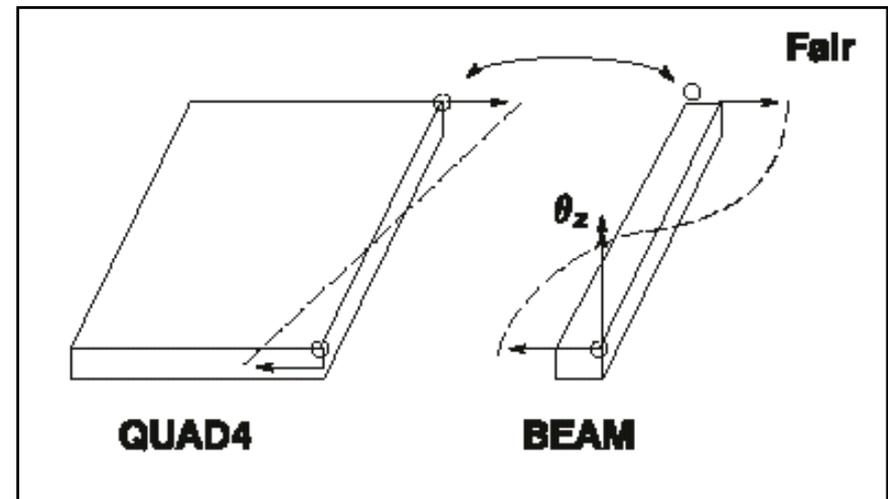
(To improve the compatibility use MPC on θ_z.)

COMPATIBILITIES (Cont.)

Corners
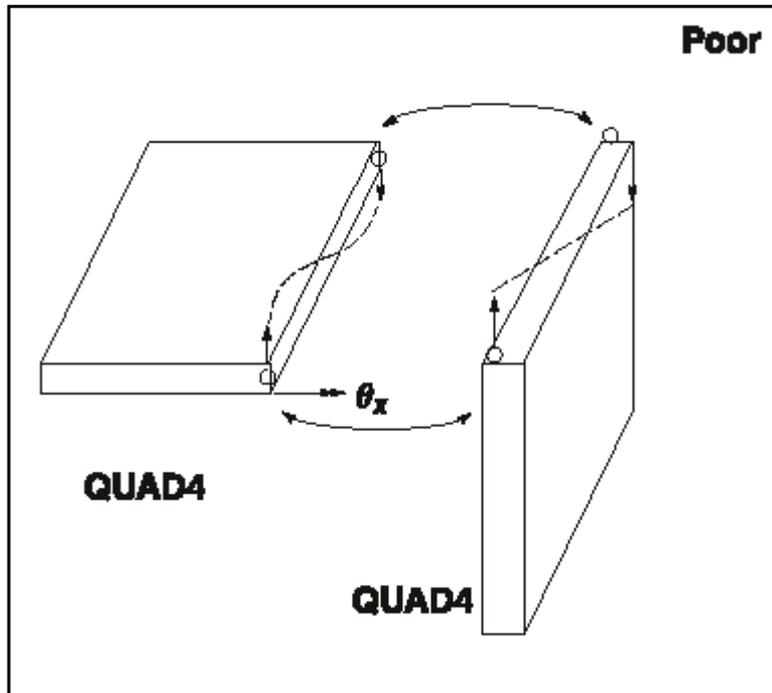
MODELING CORNERS WITH PLATES AND BEAMS

Problem – **A corner is much stiffer than is represented by two plate elements coming together**
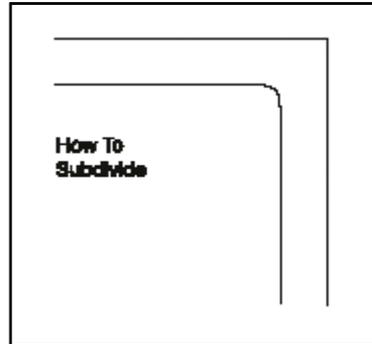
Poor Solution
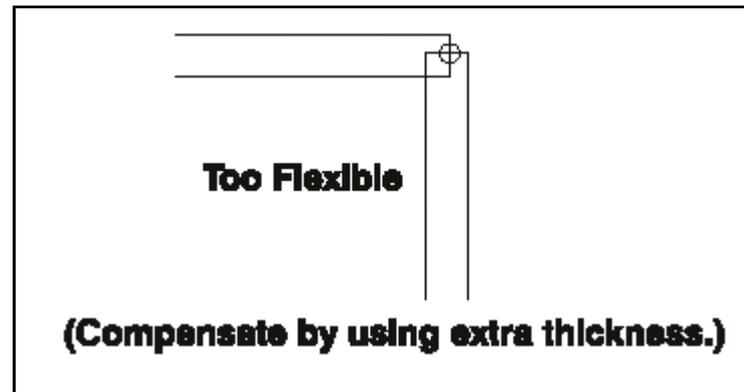
MODELING CORNERS WITH PLATES AND BEAMS (Cont.)

Good Solutions
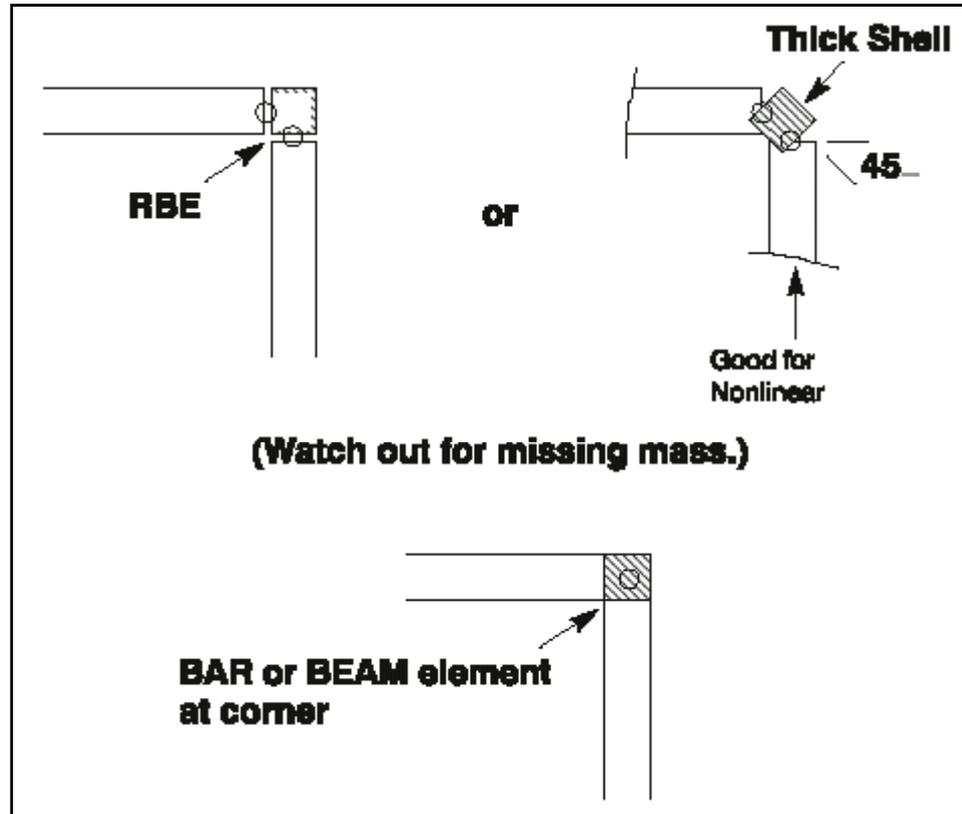
(recommended properties: calculated based on ½ of each attached plate elements – use only I1 ad I2 on the PBAR or PBEAM)

EXAMPLE OF POOR PRACTICE

Reinforcing web does not have constant strain.

TRIA3

Web

Beams
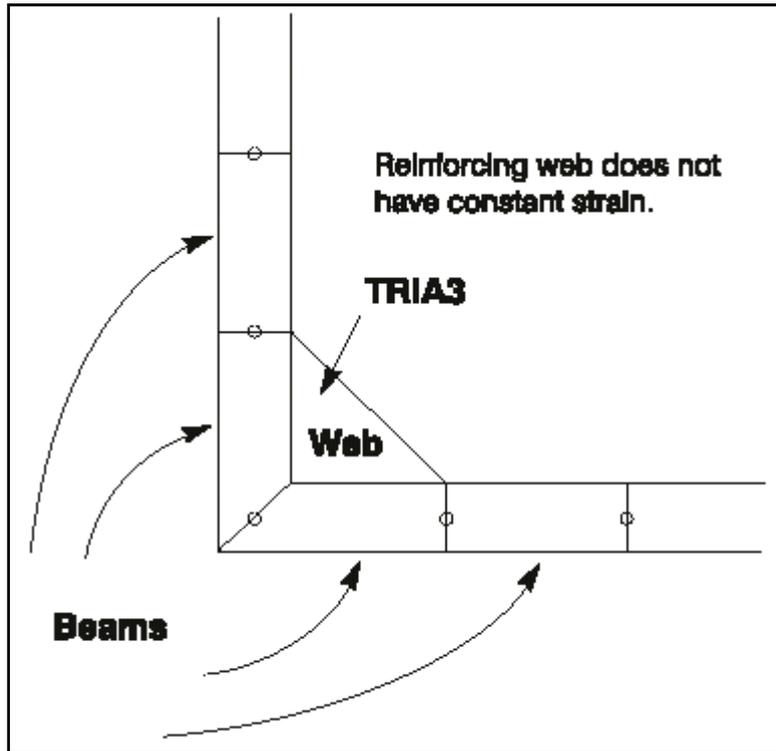
Preferred Practice

Rod
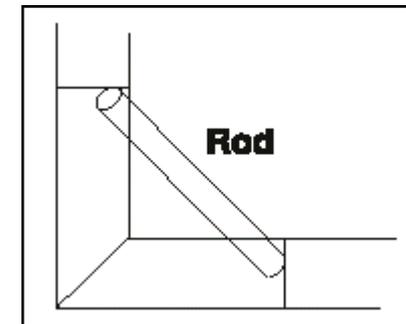
EQUIVALENT ROD

Assume
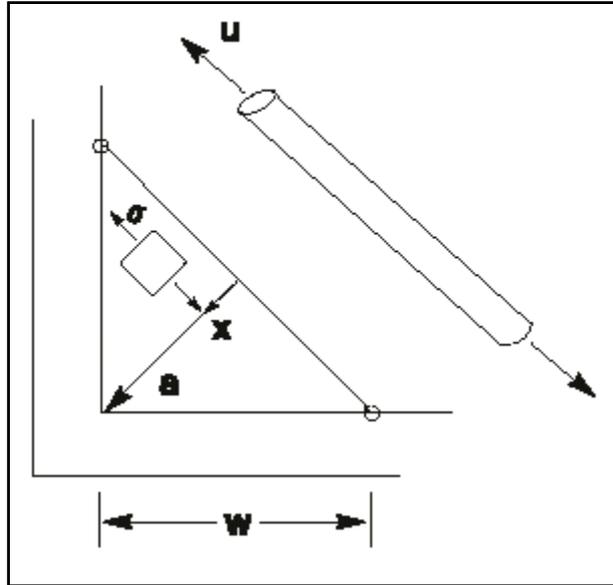
Length of Rod (L) = w

Area of Rod (A) = ?

$$K_{eq} = \frac{AE}{\left(W\sqrt{2}\right)} = \frac{AE}{2a}$$

$$\sigma = \sigma_o\left(1 - \frac{x}{a}\right)$$

$$\varepsilon = \varepsilon_o\left(1 - \frac{x}{a}\right)$$

$$\varepsilon_o = \frac{u}{2a}$$

EQUIVALENT ROD (Cont.)

Energy

$$dU = \frac{t}{2} E \, \varepsilon^2 \, dA$$

$$dA = 2a\left(1 - \frac{x}{a}\right)dx$$

$$U = Et\varepsilon_0^2 \int_0^a \left(1 - \frac{x}{a}\right)^3 \, dx = \frac{Eta^2}{4}\varepsilon_0^2$$

Stiffness

$$K = \frac{\partial^2 U}{\partial u^2} = \frac{Et}{8}$$

To match this stiffness by a rod of length $w\sqrt{2}$, and area A: $EA/(w\sqrt{2}) = Et/8$

$$A = wt\sqrt{2}/8$$

MSC Software®

EQUIVALENT ROD (Cont.)

Compare with constant strain results

$$\varepsilon = \varepsilon_0 = \frac{u}{2a} \quad \mathbf{Area} = a^2$$

$$U = \frac{t}{2} E \, \varepsilon_0^2 \, a^2$$

$$= \frac{Eta^2}{2}\left(\frac{u}{2a}\right)^2$$

$$K = \frac{\partial^2 U}{\partial u^2} = \frac{Et}{4}$$

Factor of 2 difference!

MSC Software

EQUIVALENT ROD EXAMPLE RESULTS
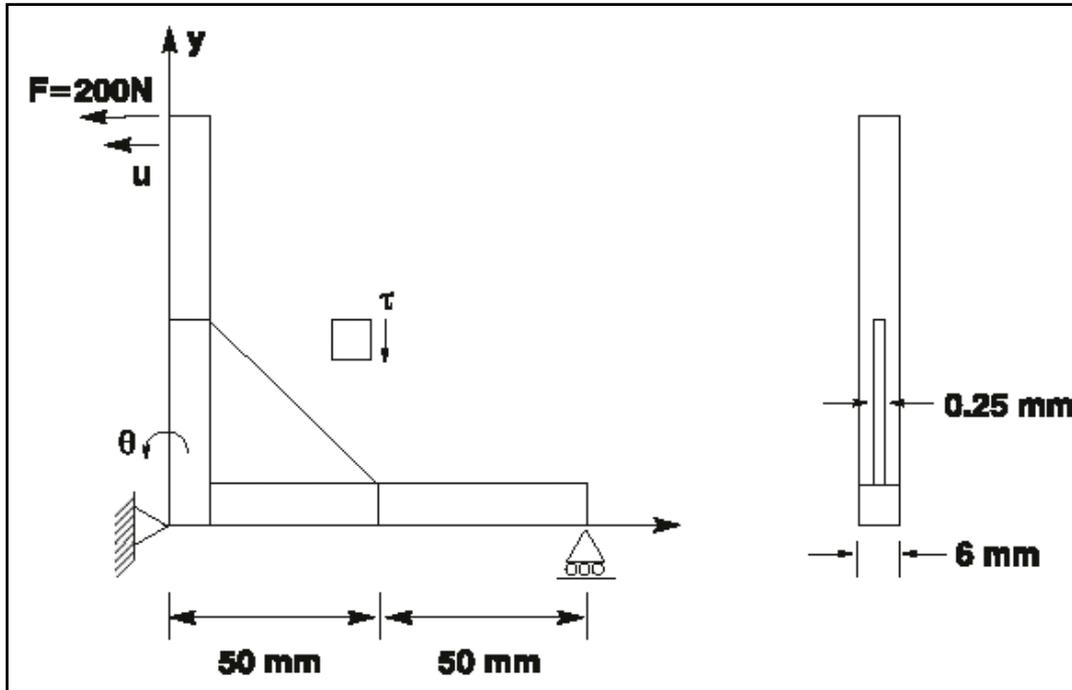
$E = 6.9 \times 10^4$

$K_{EQ} = 2156.25 \text{ N / mm}$

$A = 2.21$

Results

Case	U	τ_{max}	θ
1 TRIA	4.29	77.85	.0214
16 TRIAs	5.25	319.3	.0263
1 ROD	5.03	294.5	.0251

MSC Software

MESH TRANSITIONS

- **In general, mesh transitions are handled by modern pre–processors such as Patran, and are not as much of a concern as they were in the past.**

- **General rules for mesh transitions**
  - Keep transitions away from areas of interest.
  - Try to use compatible elements.
  - If compatible elements cannot be used, use "R"-type elements to approximate the dominant behavior.
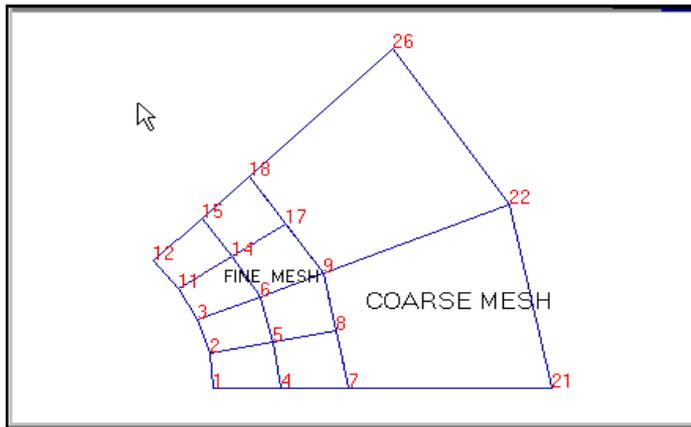
Coarse and Fine Mesh
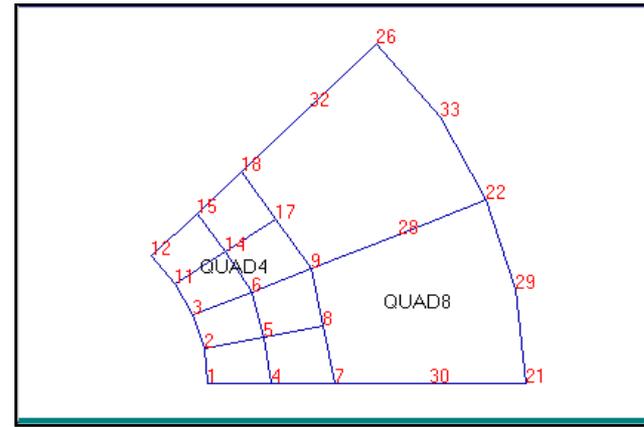
Elements of Different Types

MSC Software

MESH TRANSITIONS (Cont.)
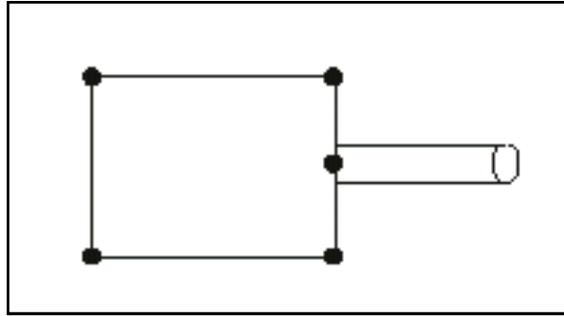
Non-Conforming Element Types
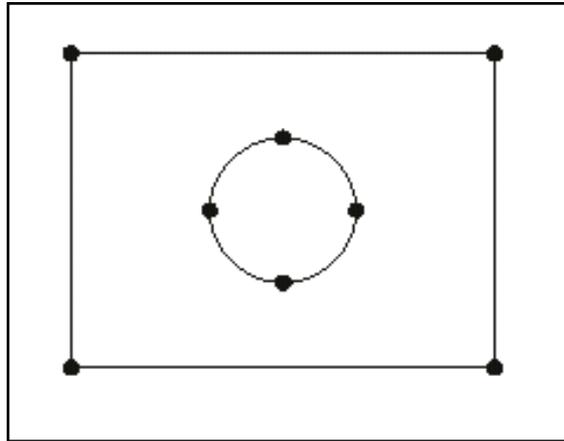
Mismatched Shapes

BEAM TO PLATE ELEMENTS

- **Situation: Your pre–processors may not handle beam–to–plate, beam–to–solid, or plate–to–solid connections automatically.**

- **If you have any of these connections in your model, they require special modeling efforts.**

- **Example:**
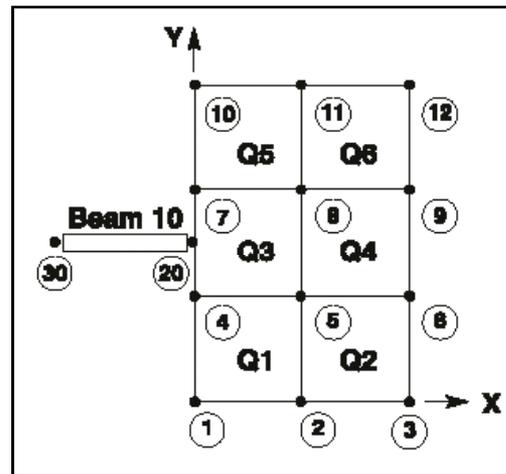
- Have existing Grid Points 1 through 12 and QUADs Q1 through Q6. It is desired to attach beam 10 to QUAD mesh using Grid Point 20. Several solutions are discussed here.

  (Note: DOF 6 of Grid Points 1 through 12 might be constrained since QUAD plates do not have stiffness in this direction.)

MSC Software

BEAM TO PLATE ELEMENTS (Cont.)

Option 1

- **Grid 20 is not added. Use offsets for BAR.**

- **Beam 10 goes from Grid 30 to Grid 4 with offset from Grid 4 to Beam 10 center line.**

- **Problems**
    - Unrealistic moment in plates is due to the beam offset.
    - The in-plane rotation must be handled. Otherwise, it is a "pinned connection" for that DOF.
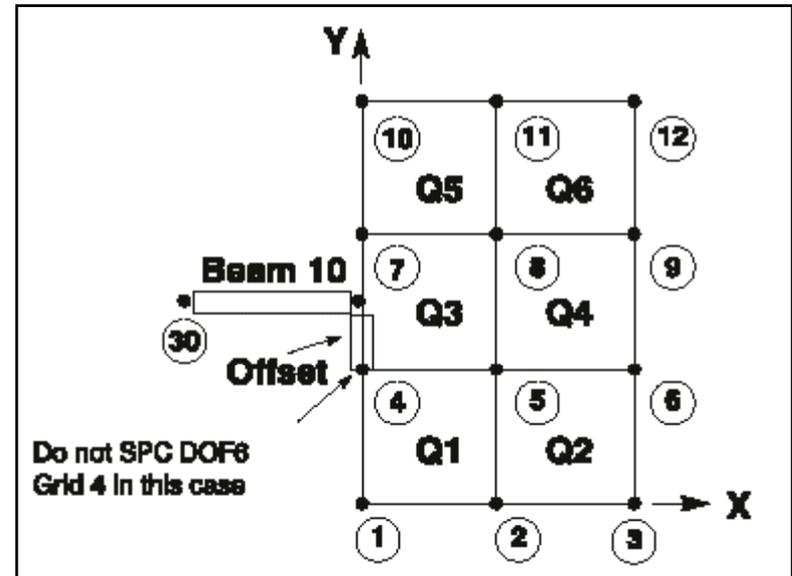
BEAM TO PLATE ELEMENTS (Cont.)

Option 2

- **Add a grid and two beams**

- **Beam properties approximated by section of a QUAD half width and its thickness**

- **Problems**
  - May have added extra stiffness at edge due to beams
  - May lose some local effects where the beam attaches to the plates

Note: DOFs 1 through 6 refer to XYZ coordinate system as defined here. In applying these solutions to another problem, note which DOFs are the out-of-plane and in-plane stiffness.
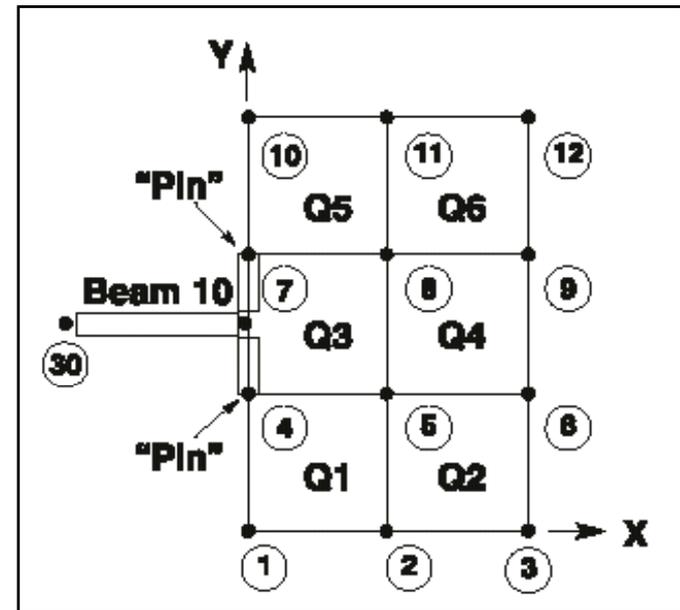
BEAM TO PLATE ELEMENTS (Cont.)

Option 3

- **Add a grid and three triangles**

- **Problems**

  - Need to add RBE3 DOF 6 from Grid Point 20 to Grid Points 4, 5, 7, and 8 in DOFs 1 and 2

  - Must add two more elastic elements and one rigid element

  - Be careful not to constrain DOF 6 at GRID point 20

MSC Software
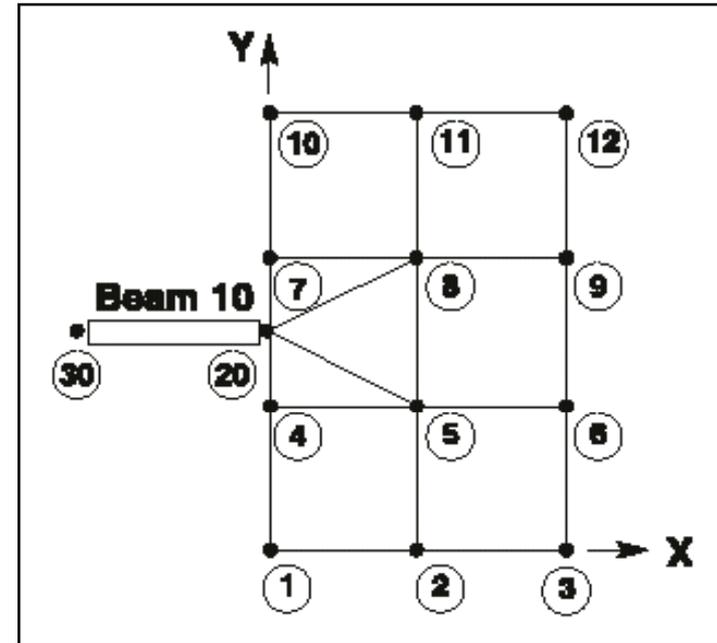
BEAM TO PLATE ELEMENTS (Cont.)

Option 4

- **Add a grid and an RBE3.**

- **RBE3 DOF 1 through 6 at Grid Point 20 to DOFs 1, 2, 3, and 5 of Grid Points 4, 5, 7, 8**

- **Problems**
  - Lose some local effects near the beam connection
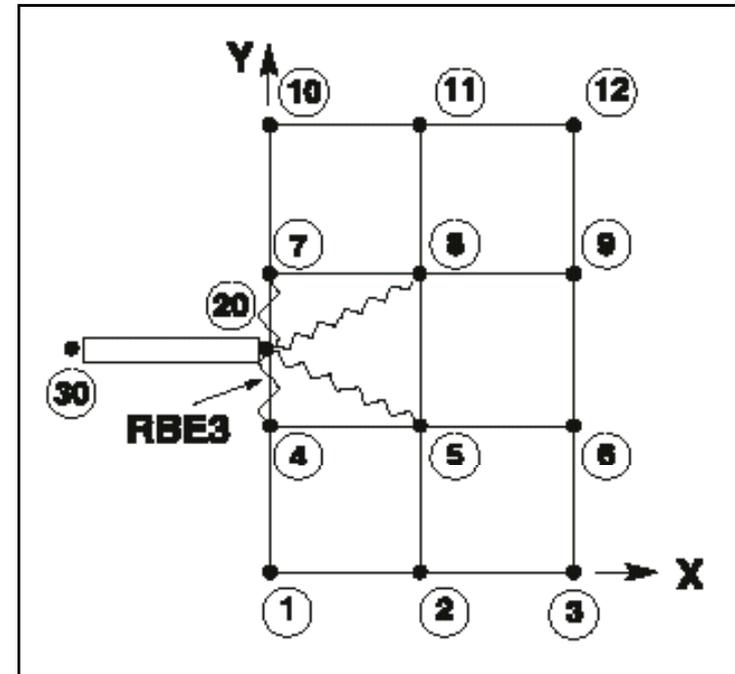
BEAM TO PLATE ELEMENTS (Cont.)

- **Situation where the beam attaches to an existing grid**

- **Beam 10 extends from Grid Point 30 to Grid Point 7**

- **RBE3 Grid Point 7 DOF 6 to DOF 1, 2, and 3 of Points 4, 8, and 10**

- **Do not SPC DOF 6 at Grid Point 7.**

- **Problems**
  - Handling the in-plane rotation

MSC Software
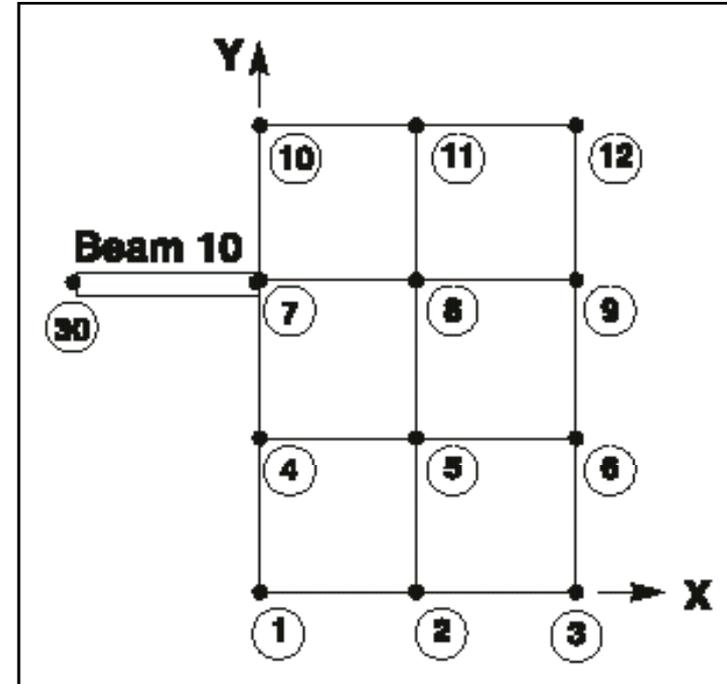
MORE MESH TRANSITIONS

Solid Plate*
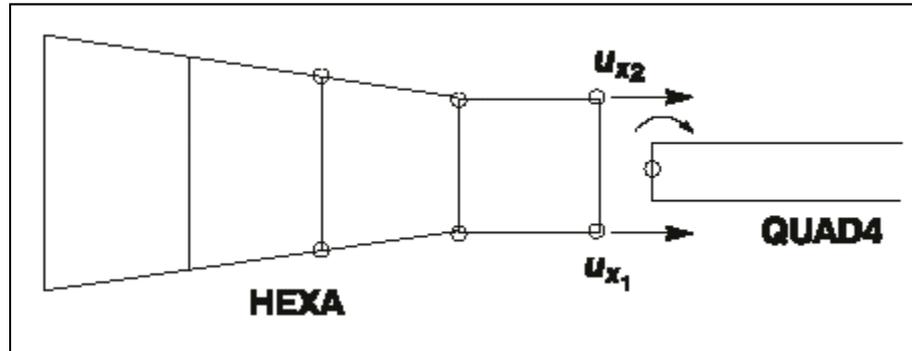
Use MPCs or RBEs

RBE2 = enforce plate theory at transition

RSSCON = easy way to make the connection, especially if your
preprocessor supports it.

MORE MESH TRANSITIONS (Cont.)

Higher Order – Lower Order
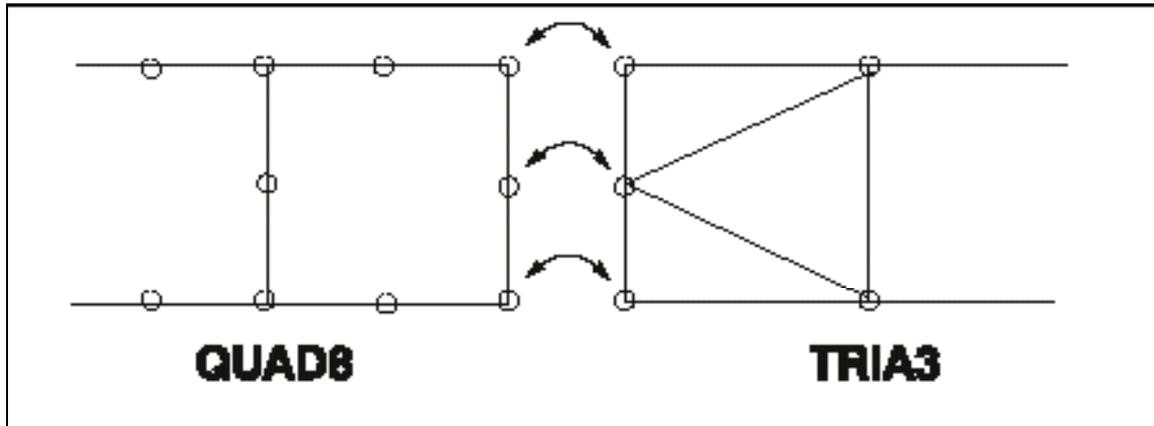
If you do this, always do it away from areas of interest.

SOME POSSIBLE PLATE-TO-SOLID TRANSITION

Split Plate
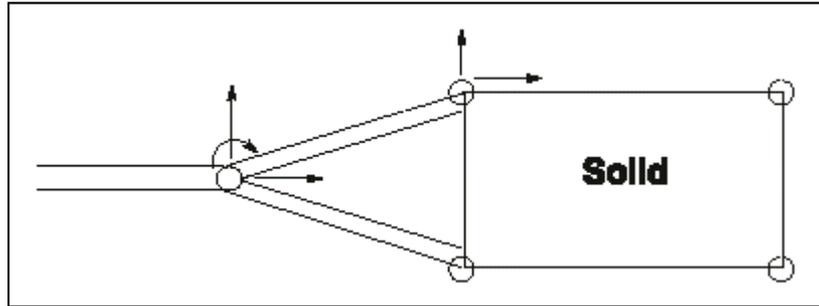
Extra Element
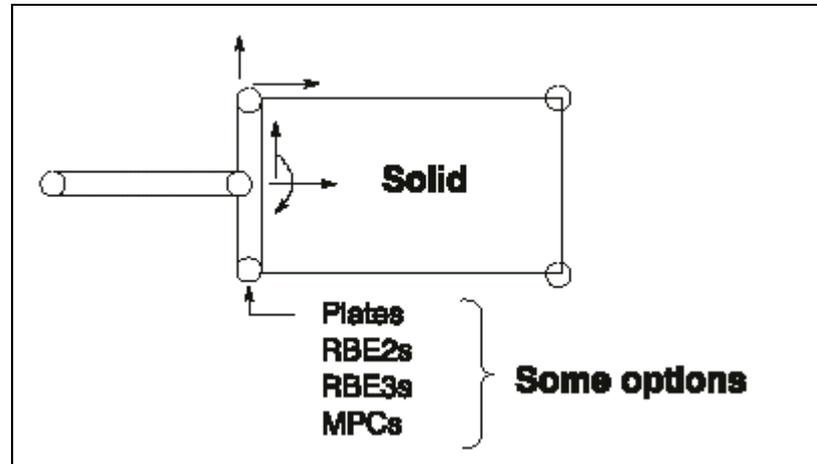
SOME POSSIBLE PLATE-TO-SOLID TRANSITION (Cont.)

<u>Split Solid</u>
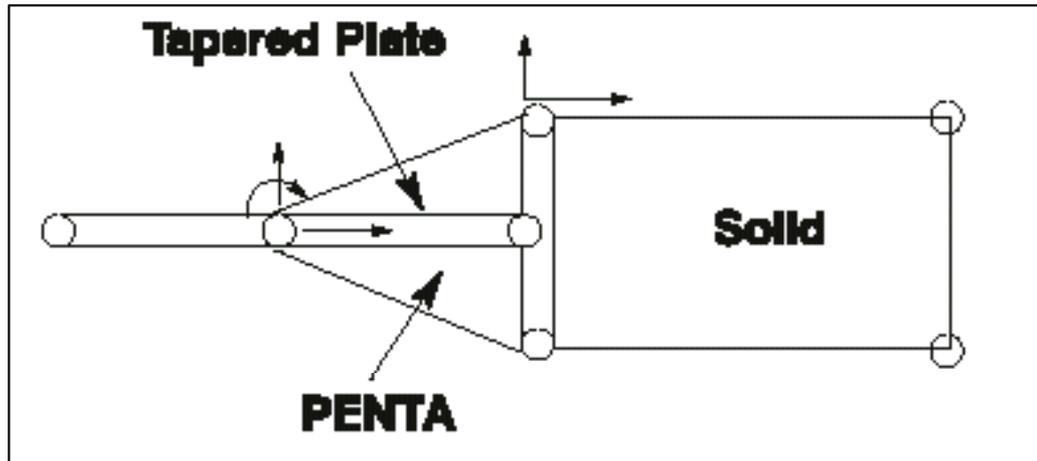
<u>Note:</u> The plate may be the same thickness as the solid.

APPENDIX A

DOCUMENTATION

DOCUMENTATION

- **This section briefly describes the MSC Nastran documentation and how to access it**

    – MSC Nastran Quick Reference Guide

      - Contains complete description of the MSC NASTRAN statements, File Management statements, Executive Control statements, Case Control commands, Bulk Data entries and parameters. This book is necessary for all MSC Nastran users.

    – MSC Nastran Release Guide

      - Describes version-dependent capabilities and presents illustrative examples. This document is required to use the new capabilities in each new version of MSC Nastran.

MSC Software

DOCUMENTATION

- **This section briefly describes MSC Nastran documentation and how to access it**

SIMCOMPANION

http://simcompanion.mscsoftware.com

- **Product Info and Docs**
  - Access to all Product Documentation

DOCUMENTATION (Cont.)

- MSC Nastran Common Questions and Answers
  - Contains answers to commonly asked questions on a wide variety of analysis topics.
- MSC Nastran Reference Manual
  - Contains material that is primarily reference oriented, non-solution sequence dependent material and is highly subdivided for ease of use. This manual is independent of machine type or operating system. This document is typically not required for the day-to-day analysis activities of most analysts and can, therefore, be shared by a group or department.
- MSC Nastran Installation and Operations Guide
  - Contains machine-dependent information for installing, customizing, and using MSC Nastran.

DOCUMENTATION

- MSC Nastran DMAP Programmer's Guide
  - Replaces Section 5 of the old MSC Nastran User's Manual. This manual is required if you are using MSC Nastran DMAP features.
- MSC Nastran User's Guides and Other Books
  - User's Guides describe in detail all aspects of MSC Nastran input, output, and modeling guidelines used for specific analysis capabilities. These documents are recommended for the various capabilities that you use most often. The following books are available:

DOCUMENTATION (Cont.)

- **Getting Started with MSC Nastran User's Guide**

- **MSC Nastran Linear Static Analysis User's Guide**

- **MSC Nastran Dynamic Analysis User's Guide**

- **MSC Nastran Design Sensitivity and Optimization User's Guide**

- **MSC Nastran Thermal Analysis User's Guide**

- **MSC Nastran Superelement User's Guide**

- **MSC Demonstration Problems – Application Examples**

- **Explicit Nonlinear (SOL 700) User's Guide**

- **Implicit Nonlinear (SOL 600) User's Guide**

- **MSC Nastran Design Sensitivity and Optimization User's Guide**

- **DMAP Programmer's Guide**

MSC Software

DOCUMENTATION (Cont.)

- **MSC Nastran Numerical Methods User's Guide**

- **MSC Nastran Aeroelastic Analysis User's Guide**

- **MSC Nastran Bibliography**
  - Lists approximately 1900 technical papers pertaining to MSC Nastran arranged by author and topic.

DOCUMENTATION

- **Much of the MSC Nastran documentation is delivered with the product.**

  *<install-directory>***\Doc\pdf_nastran\nastran_library.pdf**

MSC Software

DOCUMENTATION

Older User Documentation Available

- **MSC Nastran Handbook for Superelement Analysis**

- **MSC Nastran Handbook for Nonlinear Analysis**

- **MSC Nastran Verification Problem Manual**
  Provides examples for which there are known solutions and compares MSC Nastran output with theory. Examples using most of the Unstructured Solution Sequences are included.

MSC Software®

DOCUMENTATION (Cont.)

- **The MSC NASTRAN Theoretical Manual**
  - Explains the theoretical basis of the elements and algorithms. The numerical algorithm material is superseded by the MSC Nastran Handbook for Numerical Methods and the MSC Nastran Numerical Methods User's Guide.

- **MSC Nastran Programmer's Manual**
  - Describes the input and output for the modules. This manual is not recommended for general use but is useful for advanced DMAP development.

ADDITIONAL DOCUMENTATION

- **MSC Corporate Web Site**
  - The MSC Software Corporation's Website provides several sources of information that can assist you in running MSC Nastran and our other products. MSC's Web address is

    http://www.mscsoftware.com

  - Here, you can find out what is new with MSC.Software, read white papers on the use of MSC products, download technical papers from previous User's Conference, obtain a schedule of training courses, share feedback and suggestions interactively with other users, and even download software patches and utilities. The sssalter library and error list are also posted on the Web.

MSC Software®