Implicit Nonlinear Analysis using MSC Nastran and Patran

NAS400 Course Notes

January 2015

NA*V2013.1*Z*Z*Z*SM*NAS400-NT

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.

Copyright © 2015 MSC.Software Corporation. All Rights Reserved. This notice shall be marked on any reproduction of this documentation, in whole or in part. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of MSC.Software Corporation is prohibited.

The MSC.Software corporate logo, Adams, Dytran, Easy5, Fatigue, Laminate Modeler, Marc, Mentat, MD Nastran, Patran, MSC, MSC Nastran, Mvision, Patran, SimDesigner, SimEnterprise, SimManager, SimXpert and Sofy are trademarks or registered trademarks of the MSC.Software Corporation in the United States and/or other countries. NASTRAN is a registered trademark of NASA. All other trademarks belong to their respective owners.

CONTENTS

CONTENTS

CONTENTS

CONTENTS

Appendices **Page**

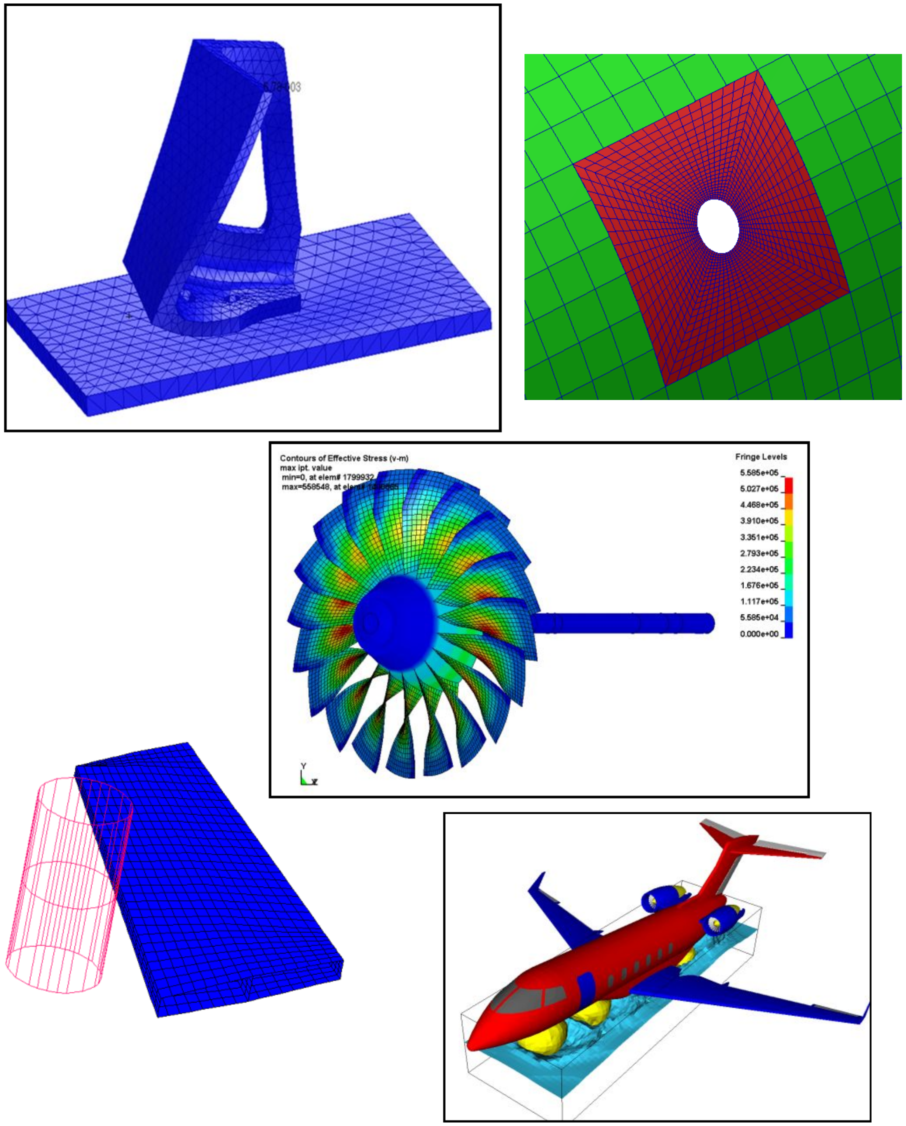
SECTION 1

INTRODUCTION TO MSC NASTRAN

SOL 400

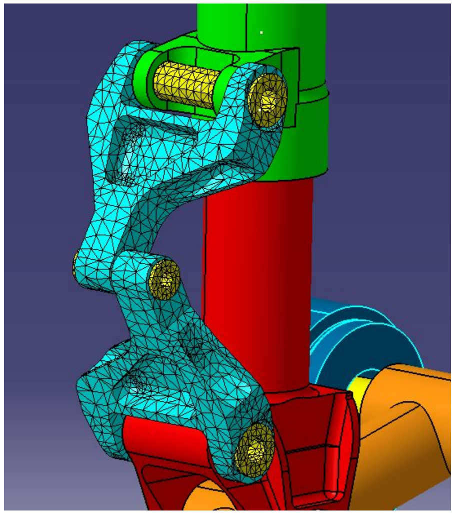
WHAT IS MSC NASTRAN?

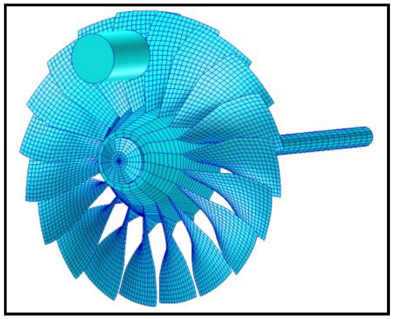
- **Evolution of engineering challenges:**
  - Complex systems vs. "just parts"
  - Interacting environments
  - Disparate tools and databases
- **Constant engineering challenge:**
  - More work in less time
  - Higher fidelity solutions
- **MSC Nastran answers these challenges**
  - Multidiscipline capabilities
  - Common analysis model format
  - Increased efficiency



MSC NASTRAN SUMMARY

- **MSC Nastran brings powerful multidiscipline capabilities**
  - Advanced Linear Solution Library
    - Full 3D Contact in Linear Static
    - Glued 3D Contact in All Linear Solutions
  - Advanced Nonlinear Solution Library
    - Full Nonlinear Contact Algorithms
    - Advanced Nonlinear Elements
    - Analysis Chaining
  - Integrated Explicit Capabilities
    - LS-Dyna and Dytran Components
    - Crash Analysis, Drop Test, Impact
    - Supported from MSC Nastran
    - Input File
    - Fluid-Structure Interaction
  - Advanced Composite Capabilities
  - Connector Elements
  - Advanced Acoustic Options





MSC NASTRAN – OVERVIEW

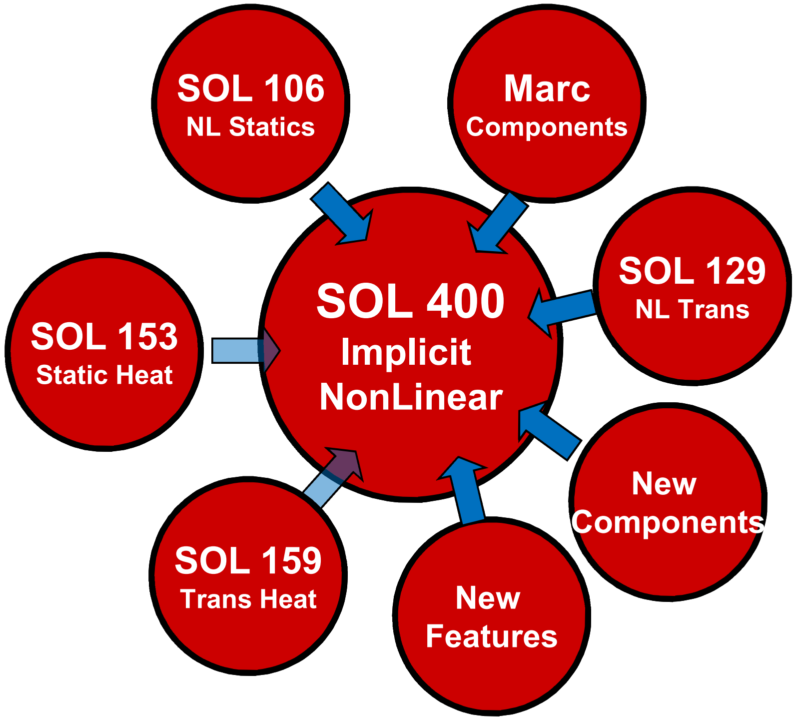
Advanced Integrated Nonlinear (SOL400)	
Adaptive Stepping Enhancements for static and dynamic analyses	☑
Linear Perturbation Analysis	☑
Nonlinear connectors elements	☑
Nonlinear materials and elements (elastomers, soils, plasticity, shape memory)	☑
Progressive failure analysis	☑
Micromechanical failure analysis and composites modeling using technology from Alphastar	☑
Hemi-cube method for radiation	☑
Thermal-structural chaining	☑

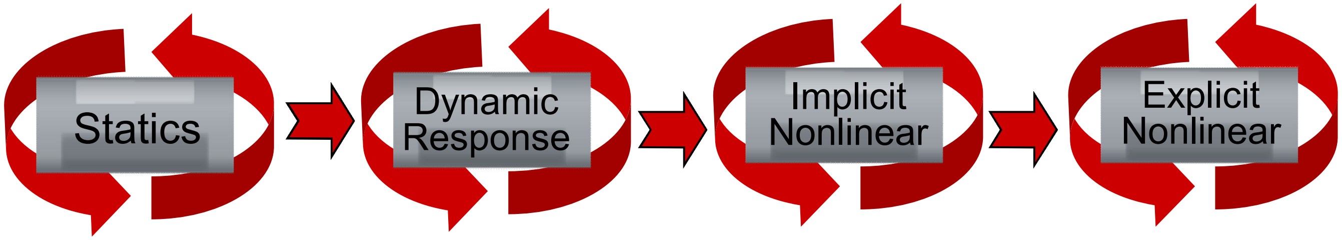
MSC NASTRAN – OVERVIEW

Advanced Integrated Nonlinear (SOL400) - Contact	
ALLBODY contact	☑
Adaptive time stepping analysis	☑
Improved time step cutback for dynamic problems with contact to avoid penetration	☑
Moment carrying glue	☑
Breakable and deactivating glue	☑
Beam to beam contact	☑
Shell edge to edge contact	☑
Improved contact performance	☑
Linear/glued contact	☑

WHAT IS SOL 400 ?

- **Advanced nonlinear solution process**
  - Combines capabilities of multiple solution sequences and software components into a common solution
  - Allows for analysis chaining
    - Automatically chaining together sequences of analyses with output state of one used as input state for another
    - Model complete processes in a single simulation through analysis chaining





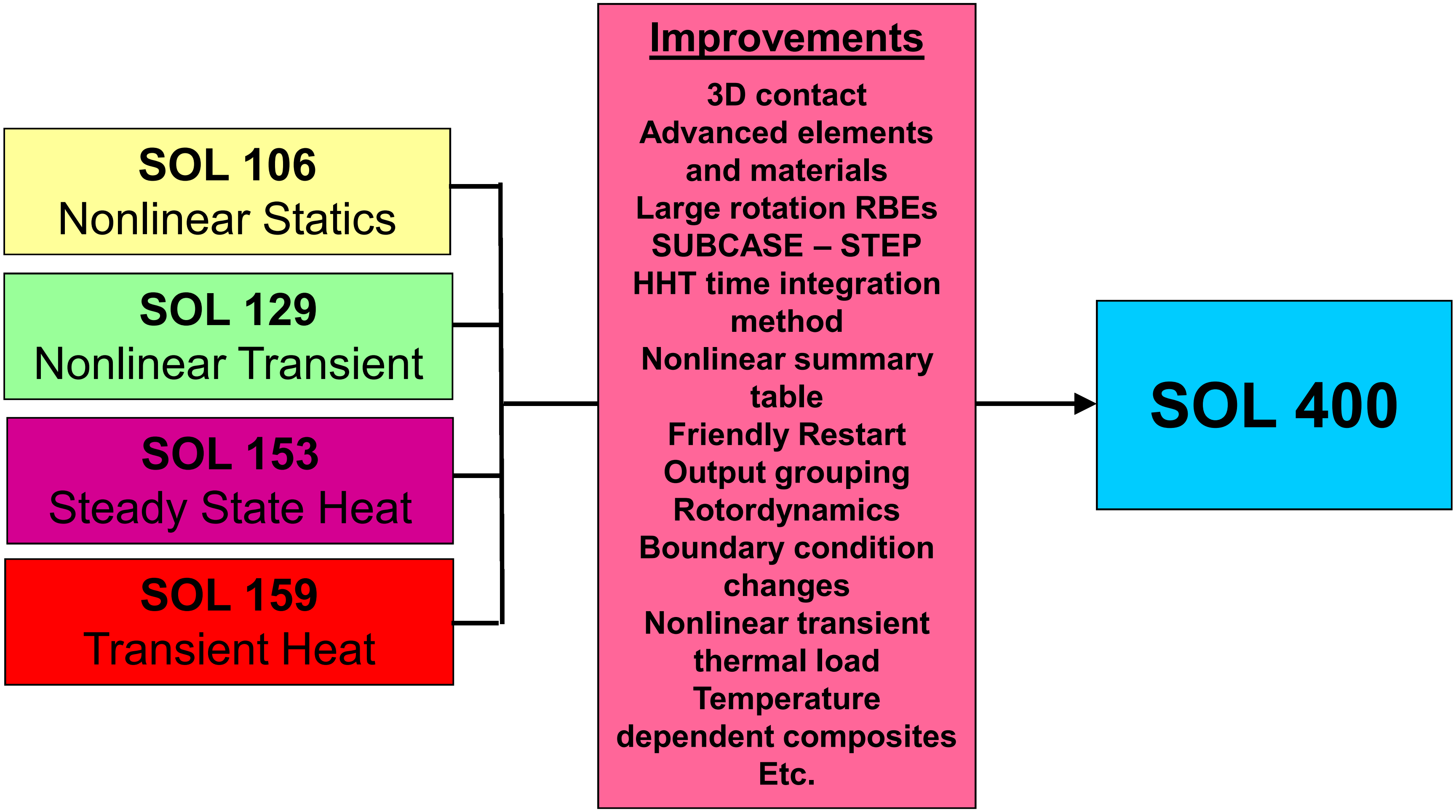
ADVANCED NONLINEAR

- **Advanced Integrated Nonlinear**
  - Utilize "native" MSC Nastran elements
    - No translation required
  - Combine static & transient in one analysis
    - Pre-stress, transient, steady-state analysis chaining
    - Thermal-Structural analysis chaining
    - Multiple, independent loadcases in 1 run
    - Linear Perturbation
  - Use general contact capability
    - Solid-to-solid, surface-to-surface, edge-to-edge, beam-to-beam, etc.
  - Go beyond small-strain element limitations
    - Large strain elements / materials (shells, solids)

ADVANCED NONLINEAR

- **Advanced Integrated Nonlinear**
  - Model large displacement / rotation rigid elements
    - Kinematic RBEi elements
  - Nonlinear Connectors
    - Large displacement / rotation CFAST, CBUSH, and CWELD elements
    - “Fuse” capability of CBUSH element
  - Simulate Composite Progressive Failure
    - Virtual Crack Closure Technique (VCCT)
    - Cohesive Zone Modeling
    - Progressive Failure Analysis (PFA)
    - Genoa Micromechanical modeling
    - Composite Beam

HISTORY OF SOL 400



SOL 106
Nonlinear Statics

SOL 129
Nonlinear Transient

SOL 153
Steady State Heat

SOL 159
Transient Heat

Improvements

3D contact
Advanced elements and materials
Large rotation RBEs
SUBCASE – STEP
HHT time integration method
Nonlinear summary table
Friendly Restart
Output grouping
Rotordynamics
Boundary condition changes
Nonlinear transient thermal load
Temperature dependent composites
Etc.

SOL 400

INTEGRATION OF MARC

- **As an enhancement to SOL 400, Marc has partly been integrated by new DMAP modules**

- **DMAP stands for Direct Matrix Abstraction Programming**
  - All Solution sequences are written in DMAP
  - The DMAP modules IO to the MSC Nastran Database

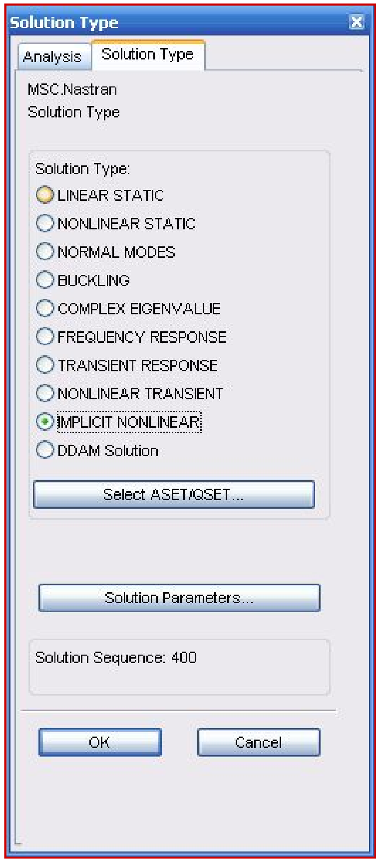
- **Existing Solution Sequences can be modified using DMAP ALTERs and users can even build their own Solution Sequences**

- **In contrast SOL 600 is not really part of MSC Nastran. It is a MSC Nastran input translator for Marc.**

WHAT IS PATRAN?

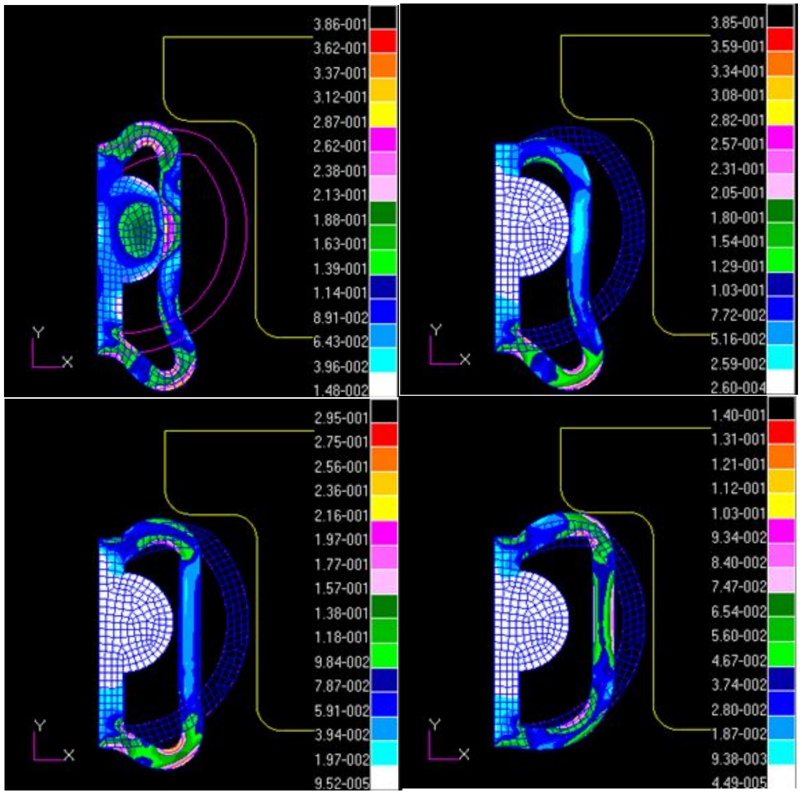
- PATRAN is a finite element pre- and post-processor, which has been integrated with several nonlinear analysis solvers including MSC NASTRAN, MARC, ANSYS and ABAQUS/Standard for implicit solutions; and DYTRAN and LS-DYNA3D for explicit solutions.
- All model definition, analysis submittal and results evaluation can be done through PATRAN and driven via the graphical user interface.
- PATRAN online help facility includes documentation for all GUI forms and topics as well as help on MSC Nastran.


Action: Analyze
Object: Entire Model
Method: Full Run
Code: MSC.Nastran
Type: Thermal
Available Jobs: test_1
Job Name: test_1

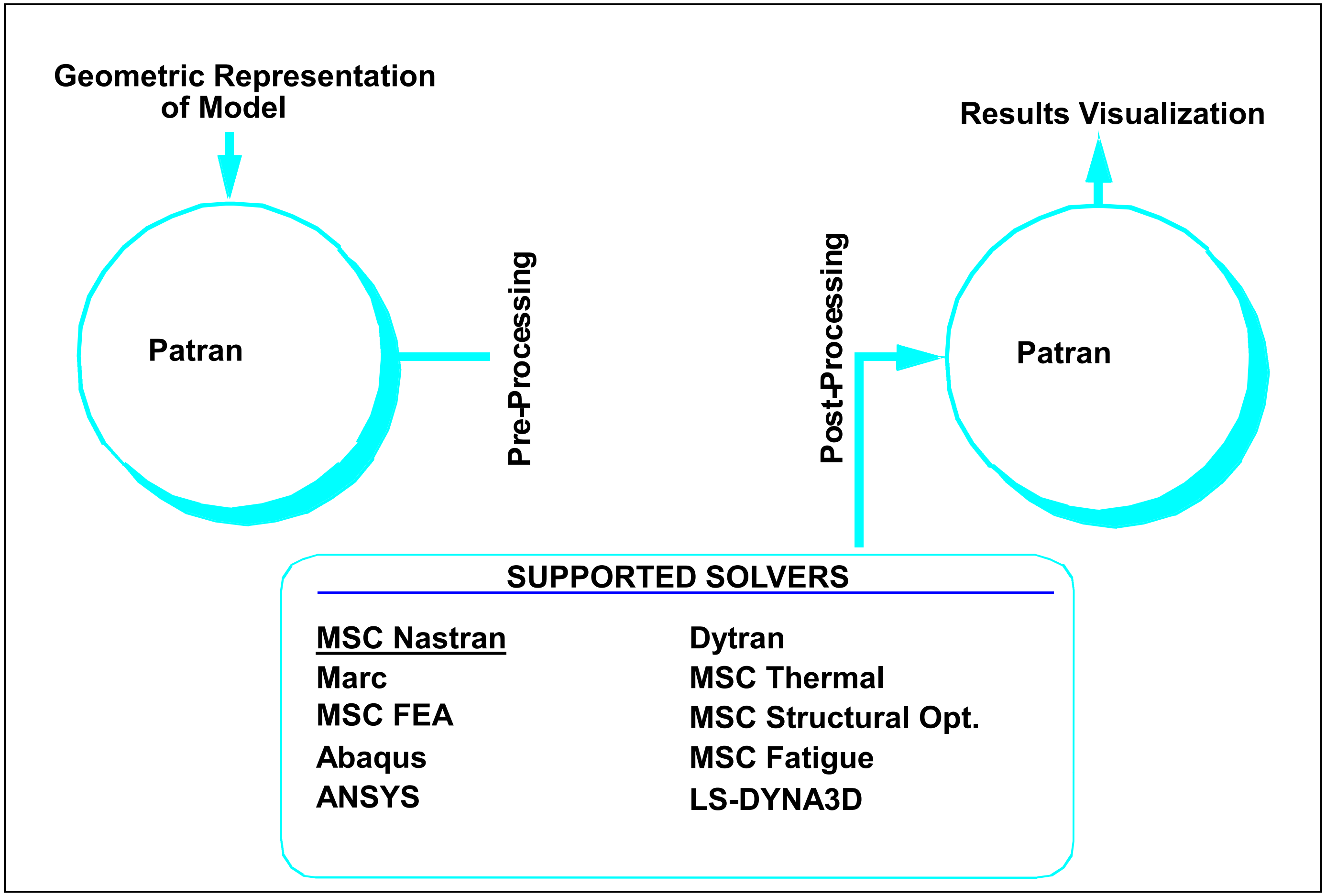

MSC.Nastran Solution Type
Solution Type:
○ LINEAR STATIC
○ NONLINEAR STATIC
○ NORMAL MODES
○ BUCKLING
○ COMPLEX EIGENVALUE
○ FREQUENCY RESPONSE
○ TRANSIENT RESPONSE
○ NONLINEAR TRANSIENT
● IMPLICIT NONLINEAR
○ DDAM Solution
Solution Sequence: 400

WHAT IS PATRAN?

- **Patran helps describe the complex behaviors typical of highly nonlinear problems**
- **Patran provides:**
  - unparalleled geometry integration capabilities
  - robust automated meshing algorithms (the new parasolid geometry editing features expand your meshing options)
  - feature-rich, mature pre- and post-processing capabilities
  - Ability to write out complex entities like geometric rigid contact bodies (nurbs)
  - Fields for spatial distribution of LBC or complex element or material properties



PATRAN IS A PRE- & POST-PROCESSOR



Geometric Representation of Model

Patran

Pre-Processing

Post-Processing

Patran

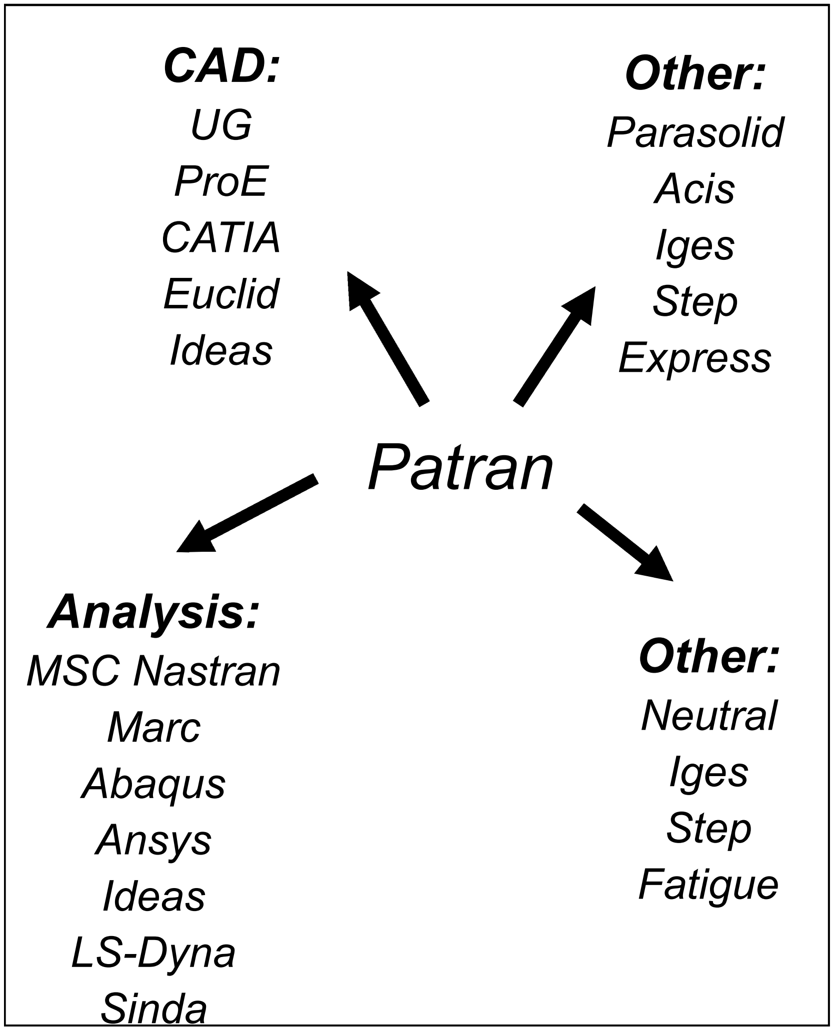
Results Visualization

SUPPORTED SOLVERS

MSC Nastran	**Dytran**
Marc	**MSC Thermal**
MSC FEA	**MSC Structural Opt.**
Abaqus	**MSC Fatigue**
ANSYS	**LS-DYNA3D**

PATRAN IS OPEN ARCHITECTURE

- **Strengths of Patran…**
- **Open Architecture – Interfaces to Any CAD or Analysis Program**
- **Patran has interfaces to all major CAD and Analysis Codes – includes input deck readers for most analysis codes. Provides "customizable" hooks for importing and exporting model information.**
  - Allows you to bring model data to anywhere/ from anywhere…



CAD:
UG
ProE
CATIA
Euclid
Ideas

Other:
Parasolid
Acis
Iges
Step
Express

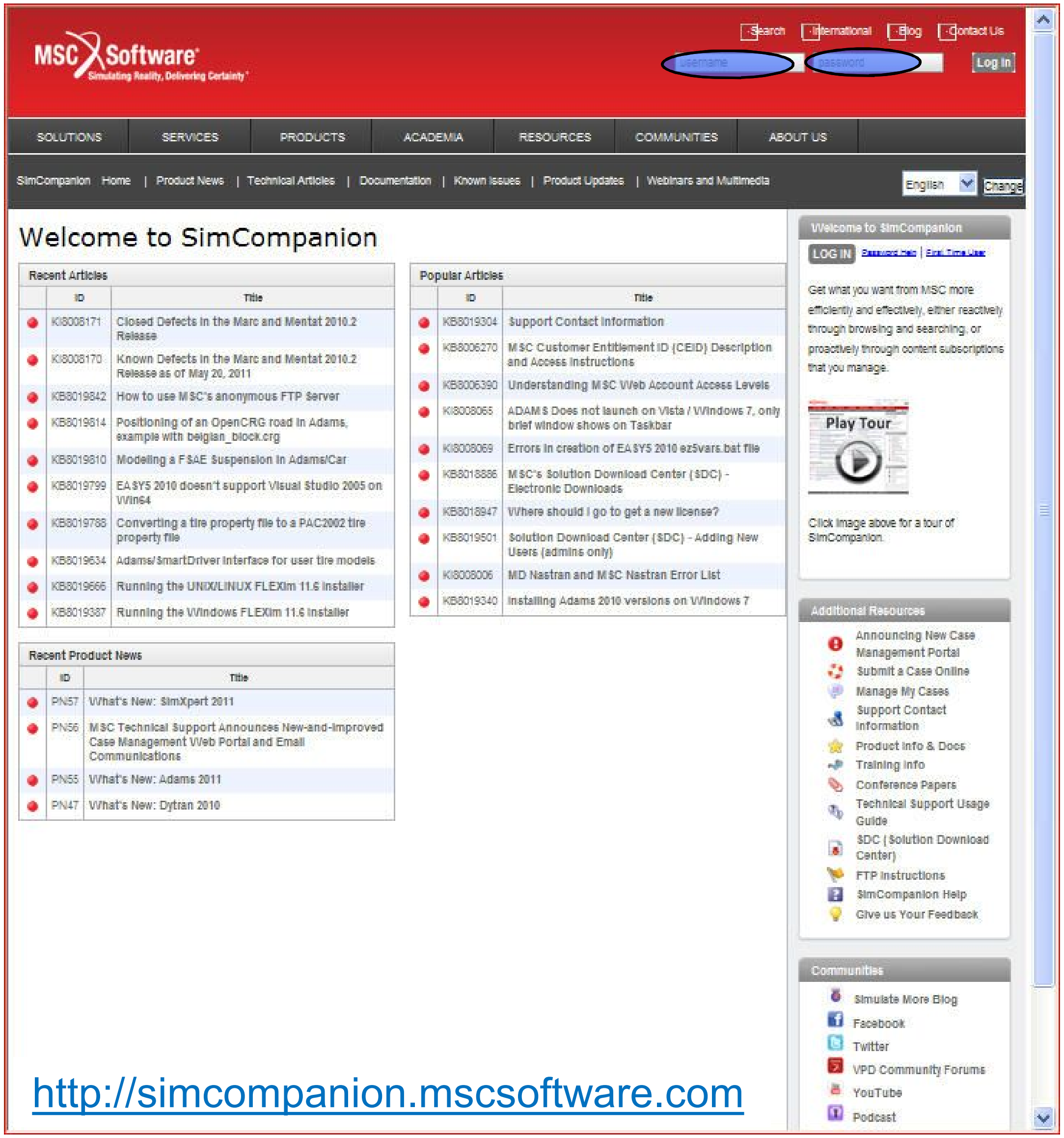
Patran

Analysis:
MSC Nastran
Marc
Abaqus
Ansys
Ideas
LS-Dyna
Sinda

Other:
Neutral
Iges
Step
Fatigue

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



Welcome to SimCompanion

Recent Articles

	ID	Title
●	KI8008171	Closed Defects in the Marc and Mentat 2010.2 Release
●	KI8008170	Known Defects in the Marc and Mentat 2010.2 Release as of May 20, 2011
●	KB8019842	How to use MSC's anonymous FTP Server
●	KB8019814	Positioning of an OpenCRG road in Adams, example with belgian_block.crg
●	KB8019810	Modeling a FSAE Suspension in Adams/Car
●	KB8019799	EASY5 2010 doesn't support Visual Studio 2005 on Win64
●	KB8019788	Converting a tire property file to a PAC2002 tire property file
●	KB8019634	Adams/SmartDriver Interface for user tire models
●	KB8019666	Running the UNIX/LINUX FLEXlm 11.6 Installer
●	KB8019387	Running the Windows FLEXlm 11.6 Installer

Recent Product News

	ID	Title
●	PN57	What's New: SimXpert 2011
●	PN56	MSC Technical Support Announces New-and-Improved Case Management Web Portal and Email Communications
●	PN55	What's New: Adams 2011
●	PN47	What's New: Dytran 2010

Popular Articles

	ID	Title
●	KB8019304	Support Contact Information
●	KB8006270	MSC Customer Entitlement ID (CEID) Description and Access Instructions
●	KB8006390	Understanding MSC Web Account Access Levels
●	KI8008065	ADAMS Does not launch on Vista / Windows 7, only brief window shows on Taskbar
●	KI8008069	Errors in creation of EASY5 2010 ez5vars.bat file
●	KB8018886	MSC's Solution Download Center (SDC) - Electronic Downloads
●	KB8018947	Where should I go to get a new license?
●	KB8019501	Solution Download Center (SDC) - Adding New Users (admins only)
●	KI8008006	MD Nastran and MSC Nastran Error List
●	KB8019340	Installing Adams 2010 versions on Windows 7

Welcome to SimCompanion

Get what you want from MSC more efficiently and effectively, either reactively through browsing and searching, or proactively through content subscriptions that you manage.

Click image above for a tour of SimCompanion.

Additional Resources

- Announcing New Case Management Portal
- Submit a Case Online
- Manage My Cases
- Support Contact Information
- Product Info & Docs
- Training Info
- Conference Papers
- Technical Support Usage Guide
- SDC (Solution Download Center)
- FTP Instructions
- SimCompanion Help
- Give us Your Feedback

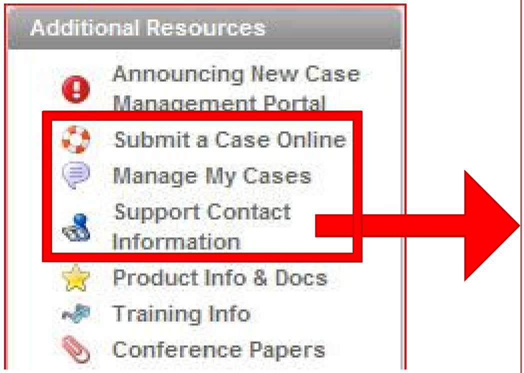
Communities

- Simulate More Blog
- Facebook
- Twitter
- VPD Community Forums
- YouTube
- Podcast

http://simcompanion.mscsoftware.com

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:

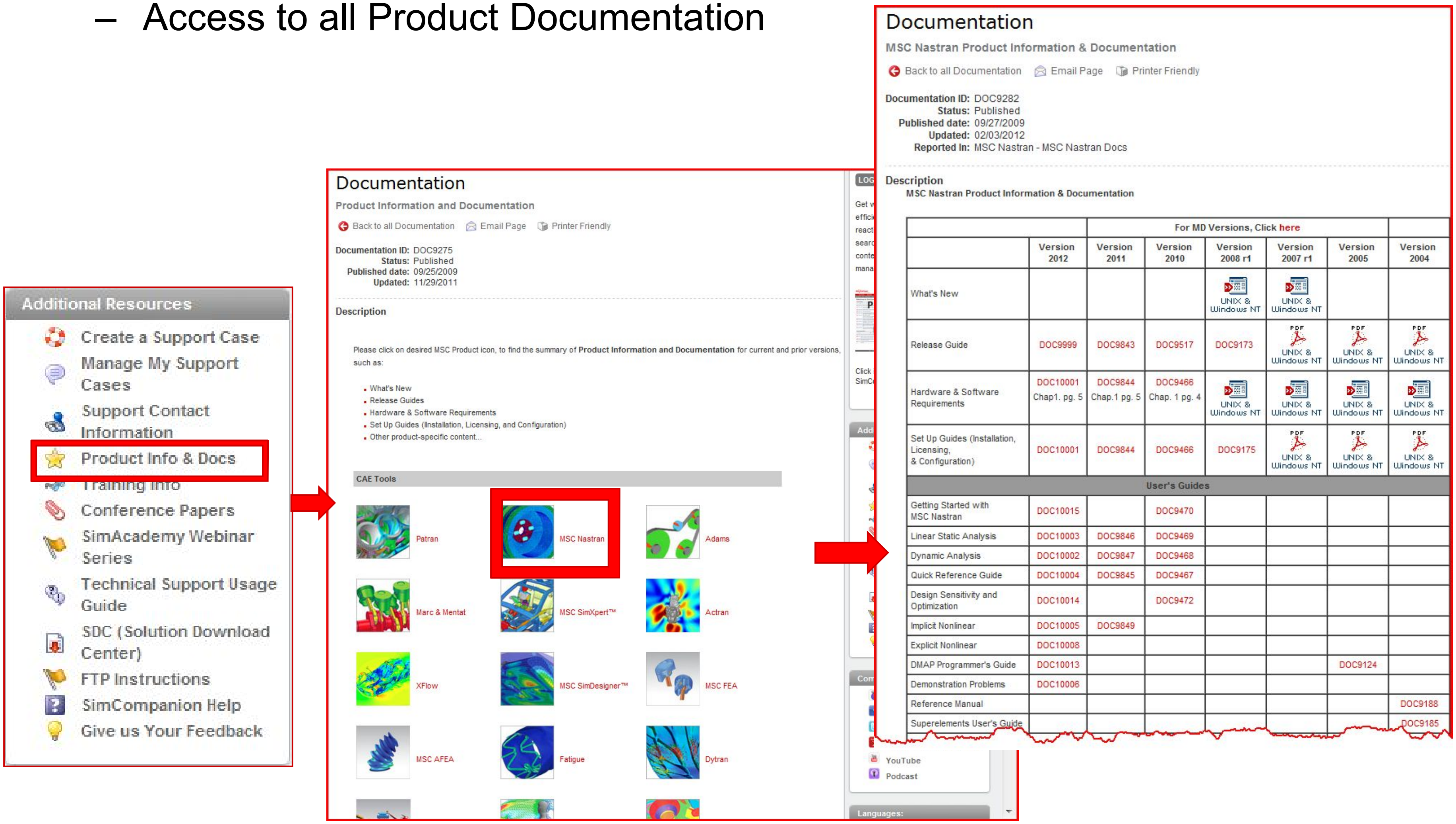
English	
- MD Nastran or MSC Nastran	mscnastran.support@mscsoftware.com
- MD Adams or Adams	mscadams.support@mscsoftware.com
- Patran	mscpatran.support@mscsoftware.com
- Dytran	mscdytran.support@mscsoftware.com
- Easy5	easy5.support@mscsoftware.com
- Fatigue	mscfatigue.support@mscsoftware.com
- Marc	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

SIMCOMPANION

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



Documentation

Product Information and Documentation

Documentation ID: DOC9275
Status: Published
Published date: 09/25/2009
Updated: 11/29/2011

Description

Please click on desired MSC Product icon, to find the summary of **Product Information and Documentation** for current and prior versions, such as:

- What's New
- Release Guides
- Hardware & Software Requirements
- Set Up Guides (Installation, Licensing, and Configuration)
- Other product-specific content...

Documentation

MSC Nastran Product Information & Documentation

Documentation ID: DOC9282
Status: Published
Published date: 09/27/2009
Updated: 02/03/2012
Reported In: MSC Nastran - MSC Nastran Docs

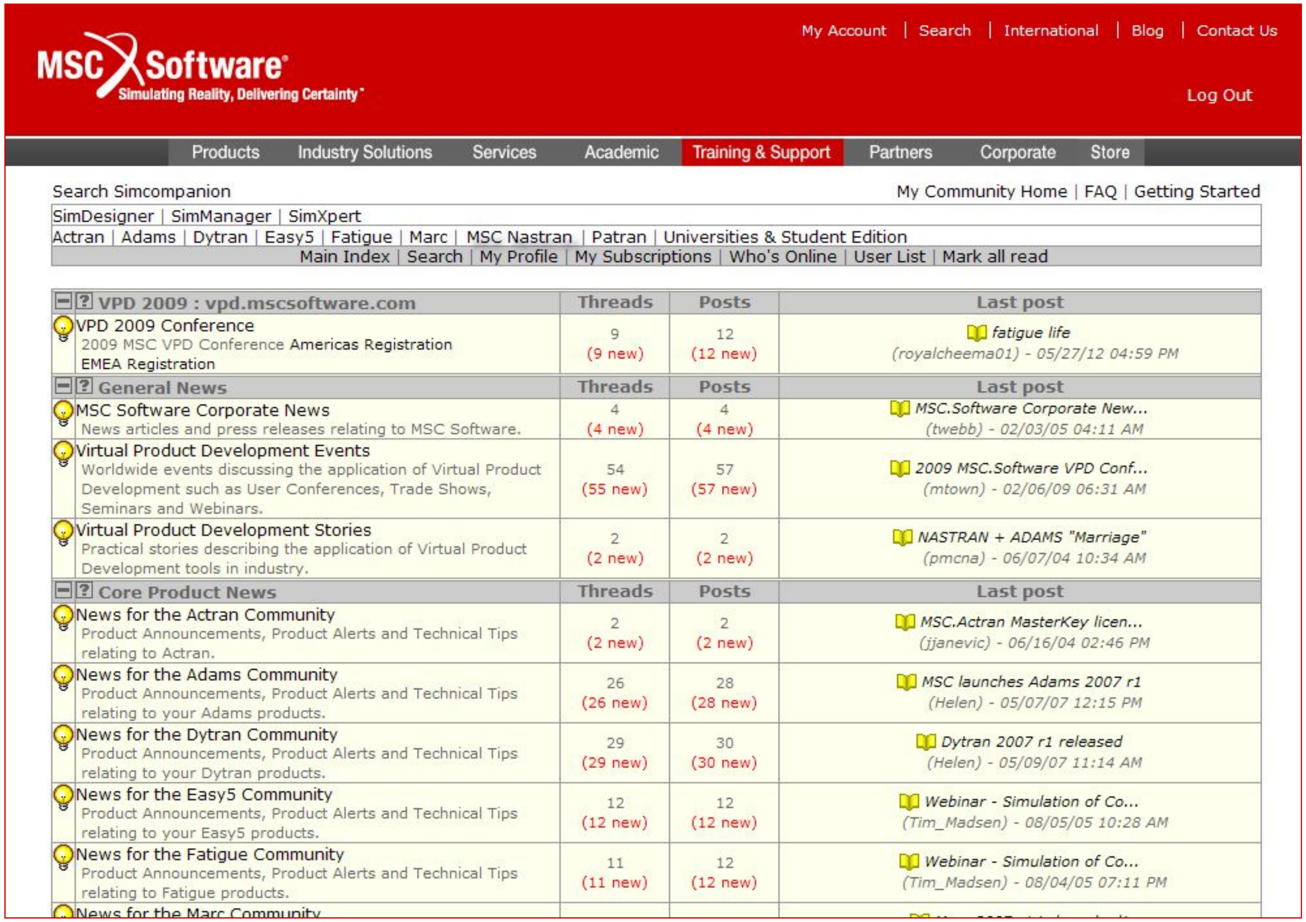
Description

MSC Nastran Product Information & Documentation

		For MD Versions, Click here					
	Version 2012	Version 2011	Version 2010	Version 2008 r1	Version 2007 r1	Version 2005	Version 2004
What's New				UNIX & Windows NT	UNIX & Windows NT		
Release Guide	DOC9999	DOC9843	DOC9517	DOC9173	PDF UNIX & Windows NT	PDF UNIX & Windows NT	PDF UNIX & Windows NT
Hardware & Software Requirements	DOC10001 Chap1. pg. 5	DOC9844 Chap.1 pg. 5	DOC9466 Chap. 1 pg. 4	UNIX & Windows NT	UNIX & Windows NT	UNIX & Windows NT	UNIX & Windows NT
Set Up Guides (Installation, Licensing, & Configuration)	DOC10001	DOC9844	DOC9466	DOC9175	PDF UNIX & Windows NT	PDF UNIX & Windows NT	PDF UNIX & Windows NT
User's Guides							
Getting Started with MSC Nastran	DOC10015		DOC9470				
Linear Static Analysis	DOC10003	DOC9846	DOC9469				
Dynamic Analysis	DOC10002	DOC9847	DOC9468				
Quick Reference Guide	DOC10004	DOC9845	DOC9467				
Design Sensitivity and Optimization	DOC10014		DOC9472				
Implicit Nonlinear	DOC10005	DOC9849					
Explicit Nonlinear	DOC10008						
DMAP Programmer's Guide	DOC10013					DOC9124	
Demonstration Problems	DOC10006						
Reference Manual							DOC9188
Superelements User's Guide							DOC9185

SIMCOMPANION

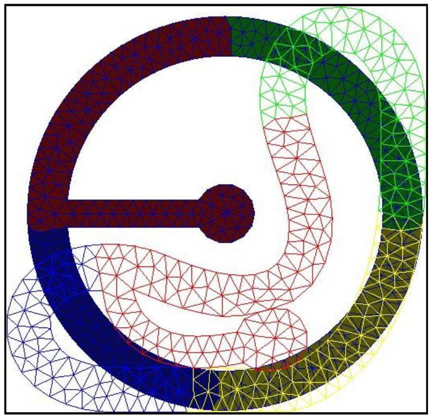
- **Access to Communities**
  - VPD Community Discussion Forums
  - Subscribe to discussion communities of interest

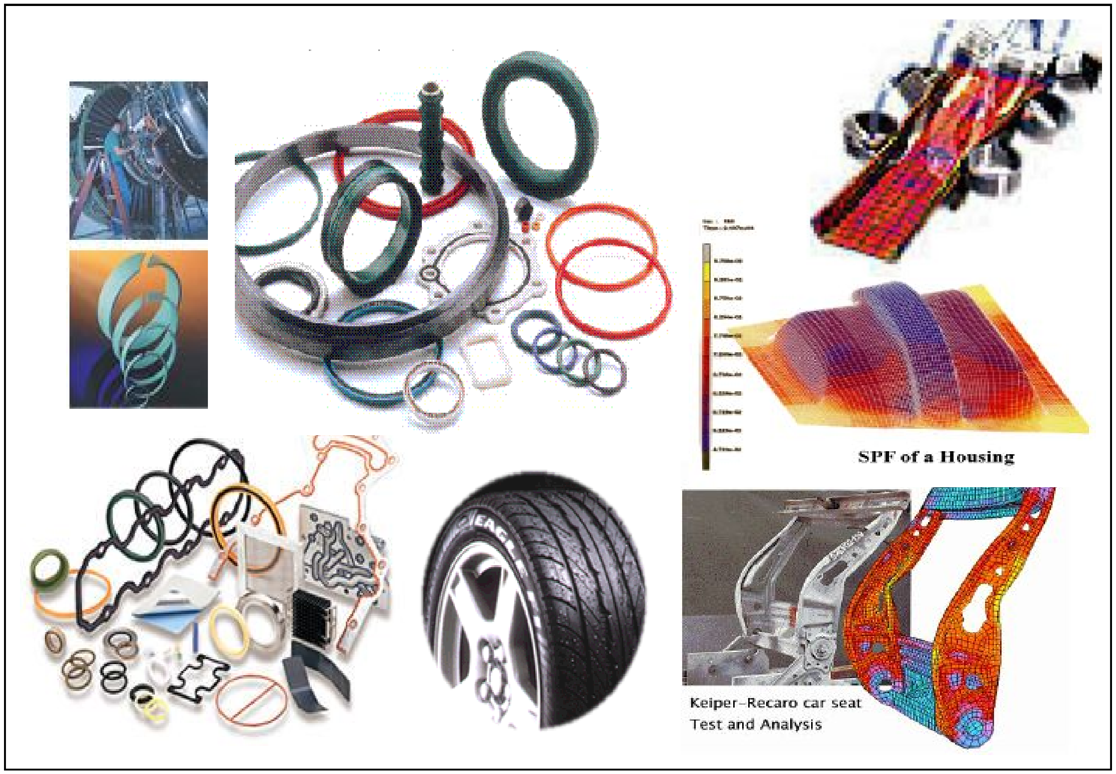


VPD 2009 : vpd.mscsoftware.com	Threads	Posts	Last post
VPD 2009 Conference 2009 MSC VPD Conference Americas Registration EMEA Registration	9 (9 new)	12 (12 new)	fatigue life (royalcheema01) - 05/27/12 04:59 PM
General News	**Threads**	**Posts**	**Last post**
MSC Software Corporate News News articles and press releases relating to MSC Software.	4 (4 new)	4 (4 new)	MSC.Software Corporate New... (twebb) - 02/03/05 04:11 AM
Virtual Product Development Events Worldwide events discussing the application of Virtual Product Development such as User Conferences, Trade Shows, Seminars and Webinars.	54 (55 new)	57 (57 new)	2009 MSC.Software VPD Conf... (mtown) - 02/06/09 06:31 AM
Virtual Product Development Stories Practical stories describing the application of Virtual Product Development tools in industry.	2 (2 new)	2 (2 new)	NASTRAN + ADAMS "Marriage" (pmcna) - 06/07/04 10:34 AM
Core Product News	**Threads**	**Posts**	**Last post**
News for the Actran Community Product Announcements, Product Alerts and Technical Tips relating to Actran.	2 (2 new)	2 (2 new)	MSC.Actran MasterKey licen... (jjanevic) - 06/16/04 02:46 PM
News for the Adams Community Product Announcements, Product Alerts and Technical Tips relating to your Adams products.	26 (26 new)	28 (28 new)	MSC launches Adams 2007 r1 (Helen) - 05/07/07 12:15 PM
News for the Dytran Community Product Announcements, Product Alerts and Technical Tips relating to your Dytran products.	29 (29 new)	30 (30 new)	Dytran 2007 r1 released (Helen) - 05/09/07 11:14 AM
News for the Easy5 Community Product Announcements, Product Alerts and Technical Tips relating to your Easy5 products.	12 (12 new)	12 (12 new)	Webinar - Simulation of Co... (Tim_Madsen) - 08/05/05 10:28 AM
News for the Fatigue Community Product Announcements, Product Alerts and Technical Tips relating to Fatigue products.	11 (11 new)	12 (12 new)	Webinar - Simulation of Co... (Tim_Madsen) - 08/04/05 07:11 PM
News for the Marc Community			

CONCLUSION

- **MSC Nastran Advanced Integrated Nonlinear (SOL400) is a powerful, easy to use tool for simulating manufacturing processes and component designs**

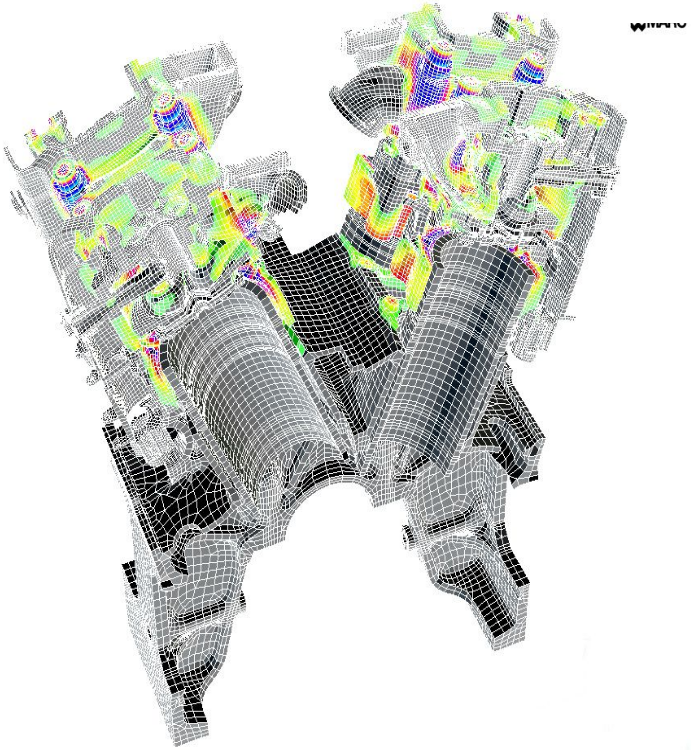




CONCLUSION

- **Combine the World's most advanced Contact and Nonlinear Finite Element Technology with the World's Leading Analysis Code and you get MSC Nastran**

- **This powerful combination will lead to:**
  - Common analysis model format
  - Increased efficiency
  - Reductions in:
    - Need for physical prototypes
    - Model re-creation effort
    - Product development time
  - Increased value of FEA simulation – an already indispensable tool!!



SECTION 2

NONLINEAR VS. LINEAR ANALYSIS

LINEAR ANALYSIS CONSEQUENCES

- **Solving a Linear System of Equations**

$$K \cdot u = P$$

- **In Linear Analysis it follows that:**
  - Loads are independent of deformation
  - Displacements are directly proportional to the loads
  - Results for different loads can be superimposed

FEM QUANTITIES IN LINEAR ANALYSIS

- **Kinematics**

$$u_e = T_{eg} \cdot u_g$$

u_e		T_{eg}		u_g
Element Displacement Vector	$=$	**Displacement Transformation Matrix**	$\cdot$	**Global Displacement Vector**

- **Compatibility**

$$\varepsilon = B \cdot u_e$$

ε		B		u_e
Element Strains	$=$	**Strain Displacement Matrix**	$\cdot$	**Element Displacement Vector**

- **Constitutive Law**

$$\sigma = D \cdot \varepsilon$$

σ		D		ε
Element Stresses	$=$	**Stress-Strain Relationship**	$\cdot$	**Element Strains**

FEM QUANTITIES IN LINEAR ANALYSIS

- Equilibrium

$$P \quad = \quad \Sigma\, T^{T}_{eg} \quad \cdot \quad F_e$$

External Load Vector — **Force Transformation Matrix** — **Element Forces**

- Constraints

$$u_g \quad = \quad \alpha$$

- The transformation matrices do not change
- Force equals displacement transformation
- The constraints (SPC, MPC) do not change

NONLINEAR ANALYSIS

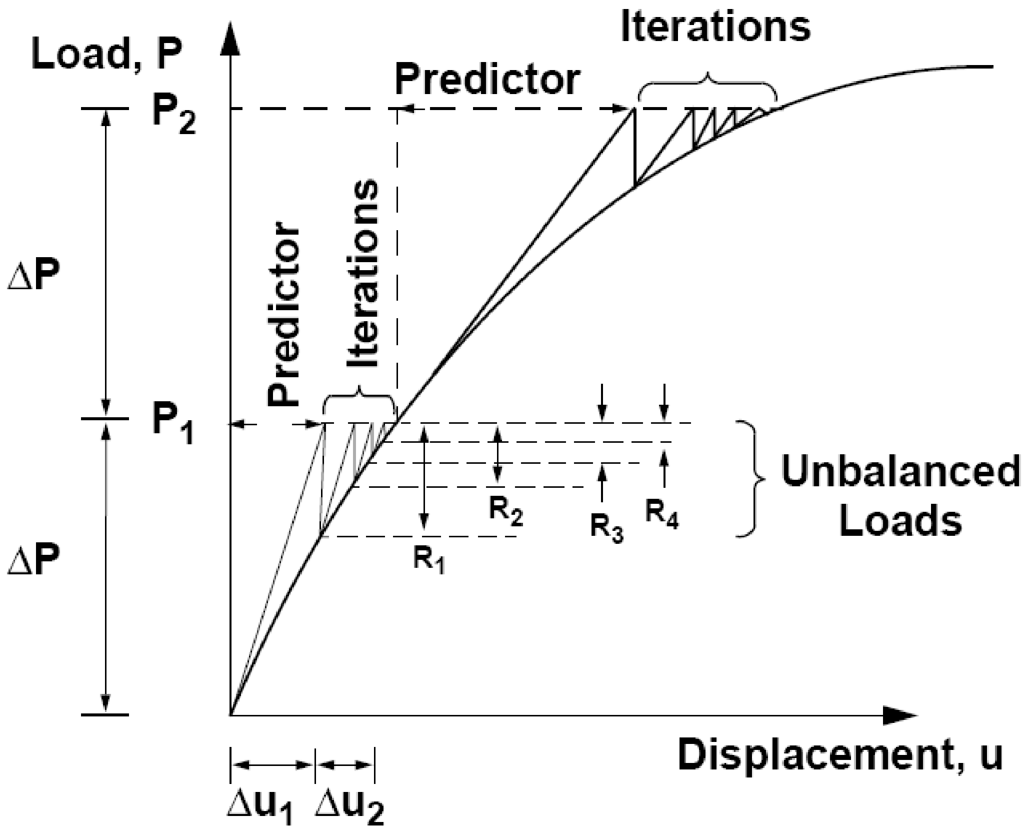
- **In nonlinear analysis, upon deformation the following changes occur:**

– Geometric nonlinearity	$T_{(disp)}, \alpha$
– Follower forces	$T_{(force)}$
– Large strain	B
– Material nonlinearity	D
– Contact	α

- **It follows that the system of equations gets nonlinear, the load sequence is unique and the results must not be superimposed**

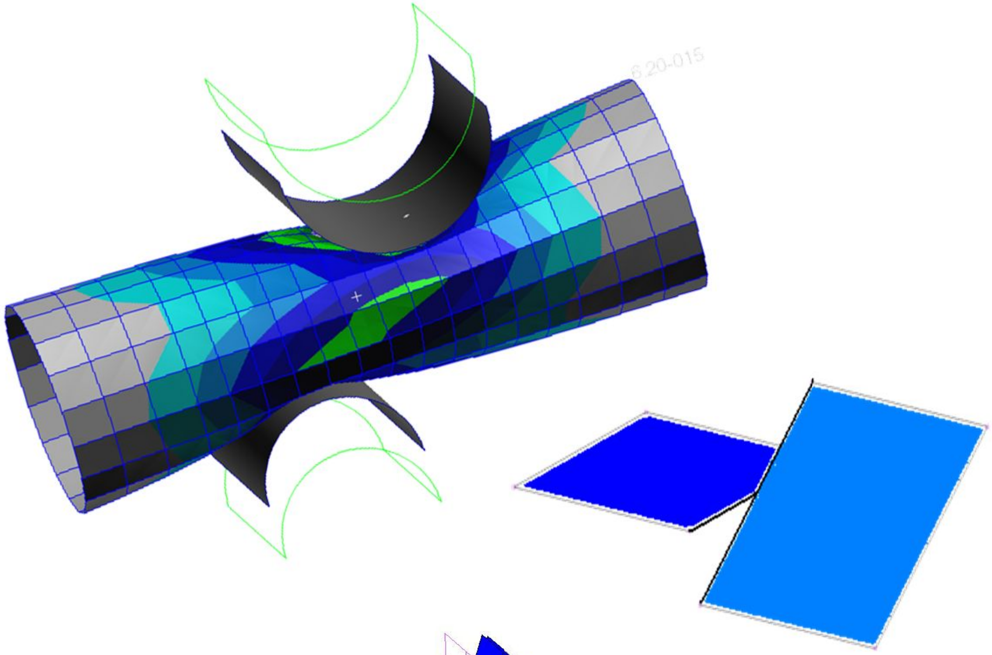
NONLINEAR ANALYSIS

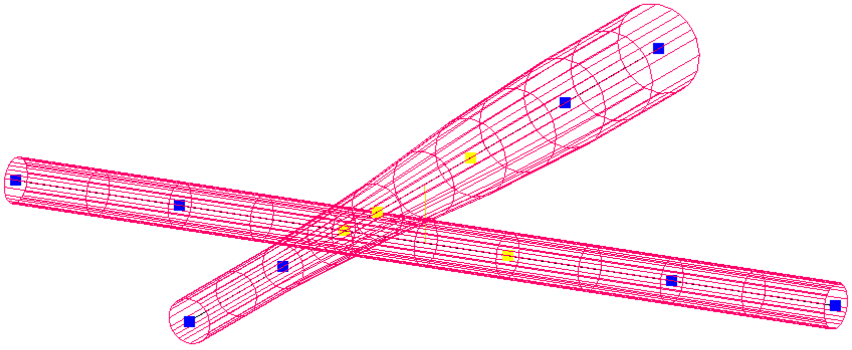
- **Since the system of equations has become nonlinear an iteration strategy is needed**

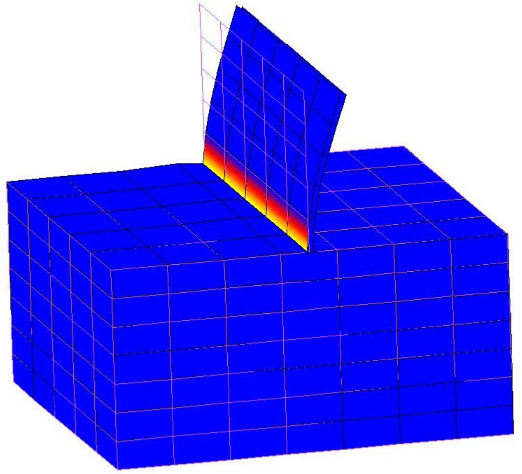


CONTACT AND CONSTRAINT CHANGES

- **Contact**
  - Deformable to Deformable
    - Surface to Surface
    - Edge to Surface
    - Edge to Edge
    - Beam to Beam
  - Rigid to Deformable
- **SPC / MPC**

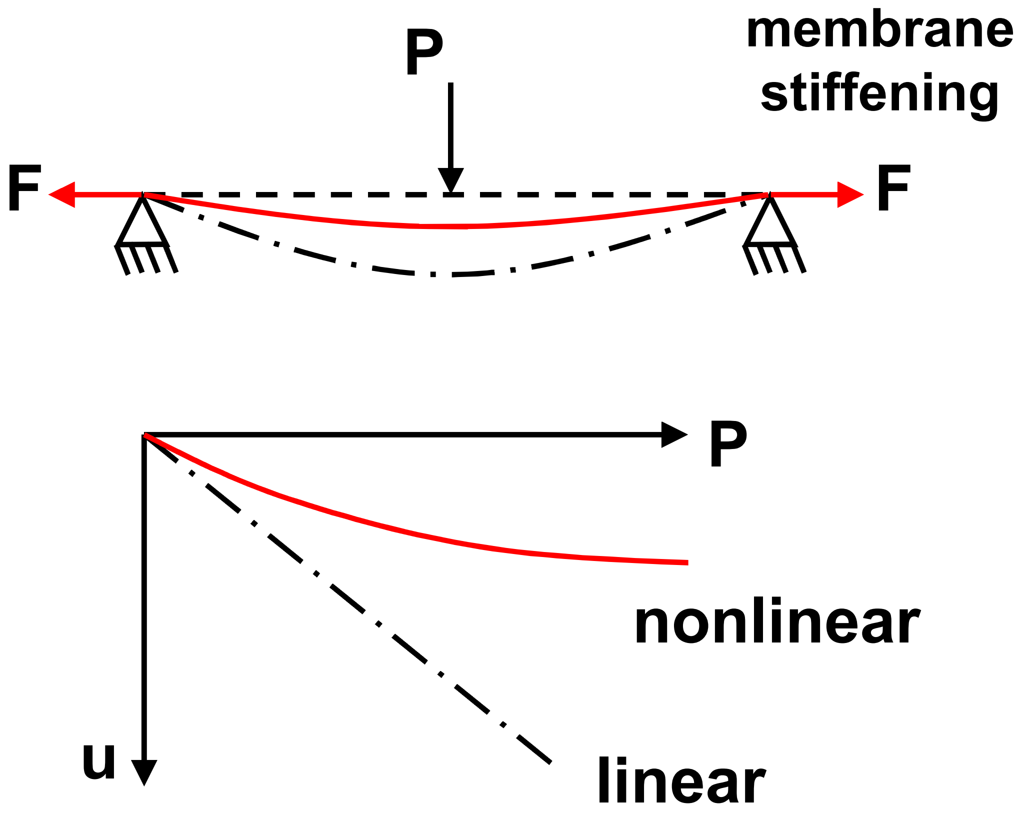


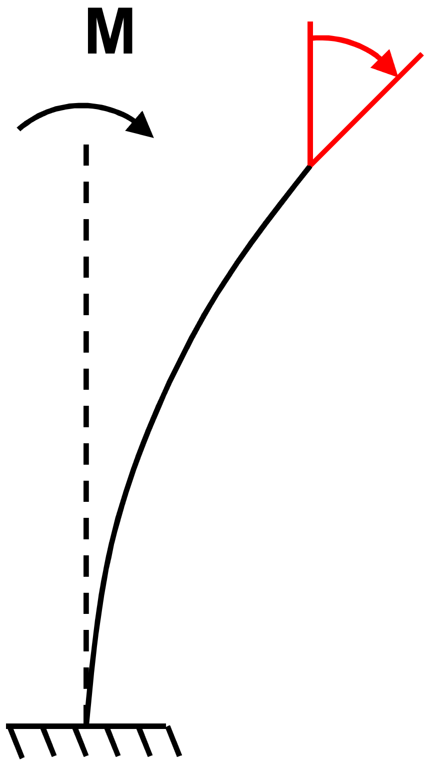




GEOMETRIC NONLINEARITY

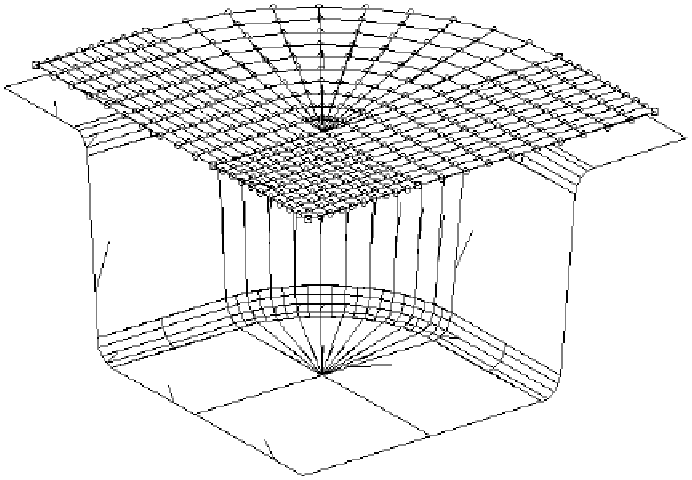
- **Large displacements and rotations, but small strains**



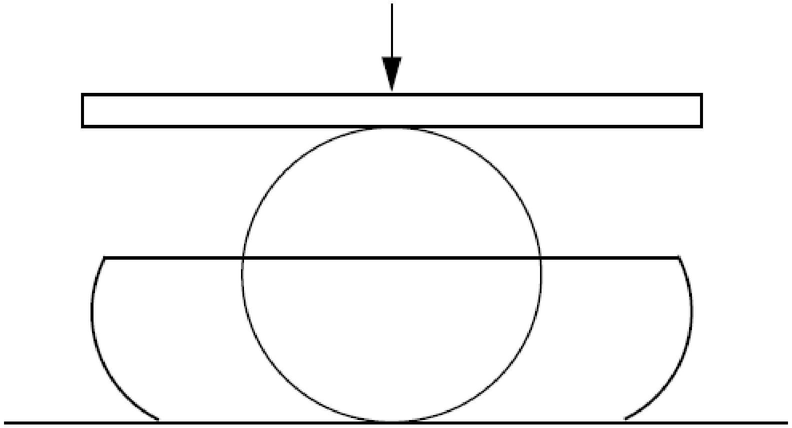


GEOMETRIC NONLINEARITY

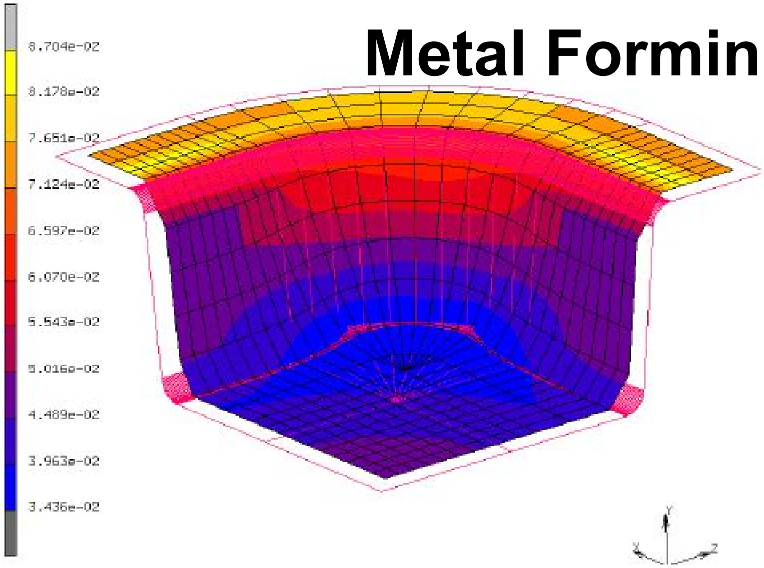
- **Large displacement and rotation and large strains**



Rubber Bearing

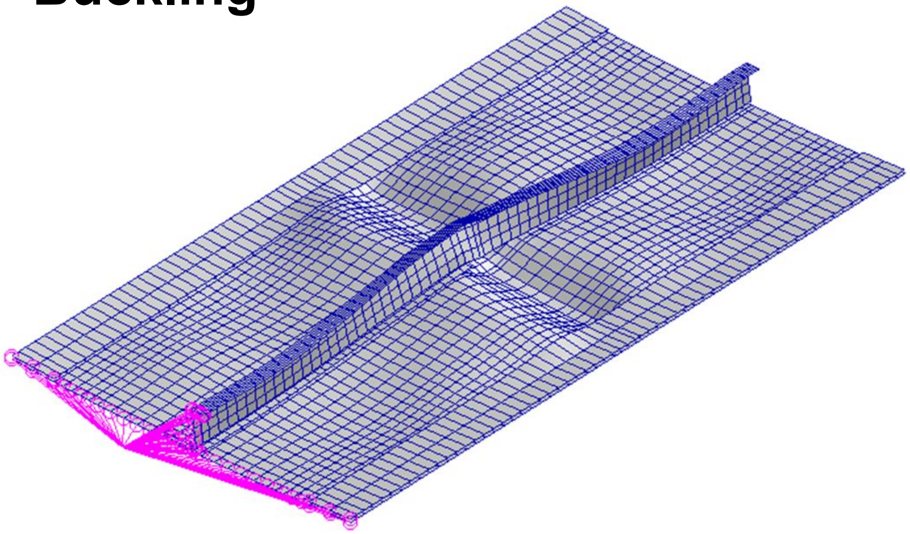


Metal Forming

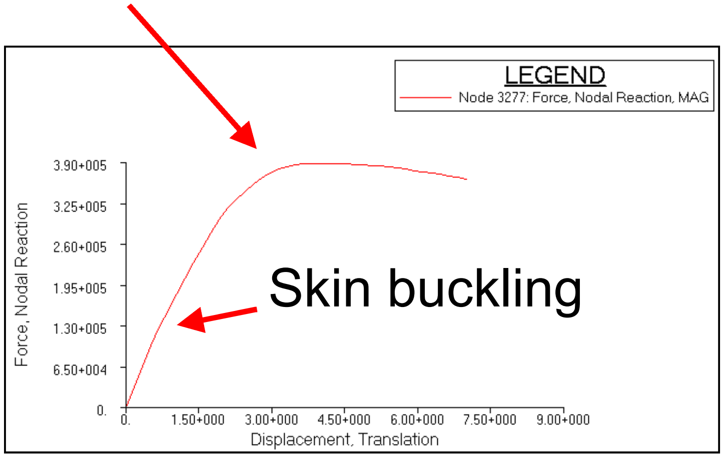


GEOMETRIC NONLINEARITY

- **Buckling**

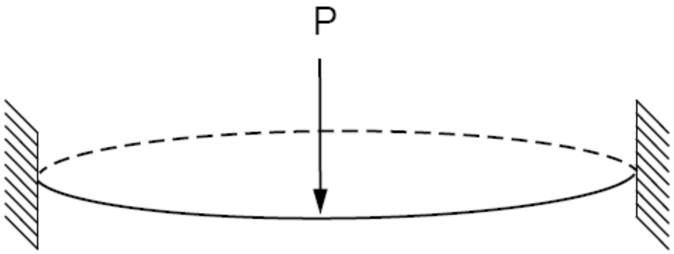


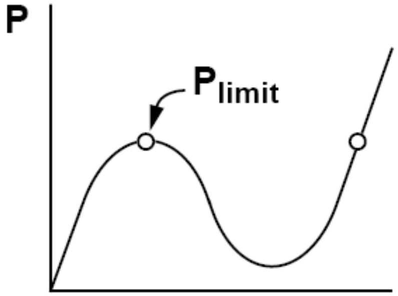
Panel failure



Skin buckling

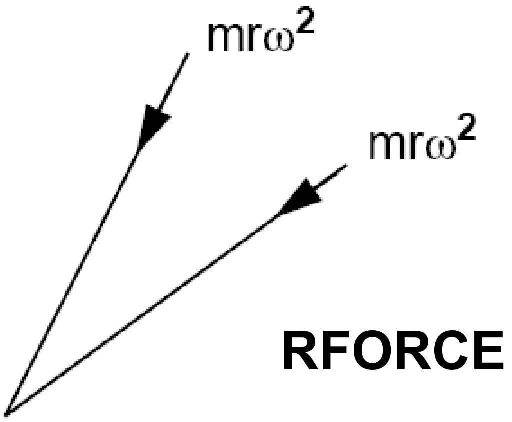
- **Snap-Through**

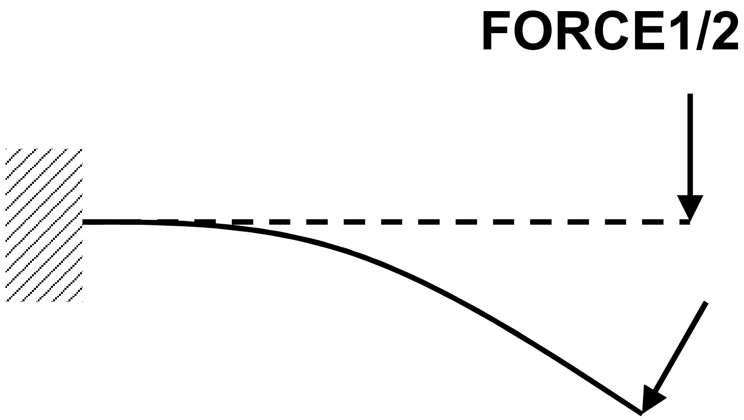


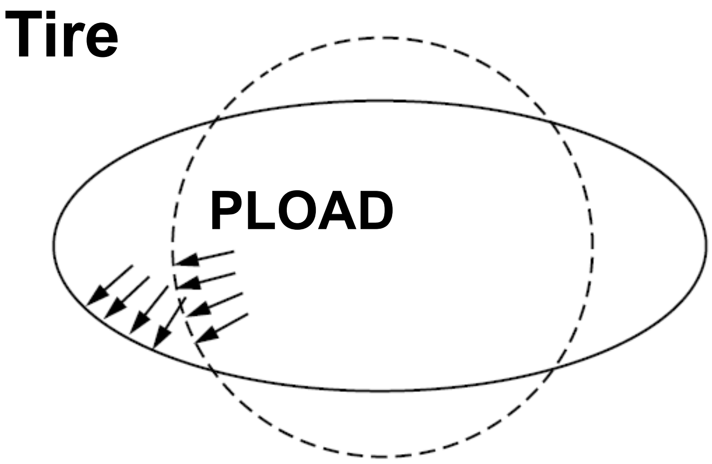


GEOMETRIC NONLINEARITY

- **Follower Forces**



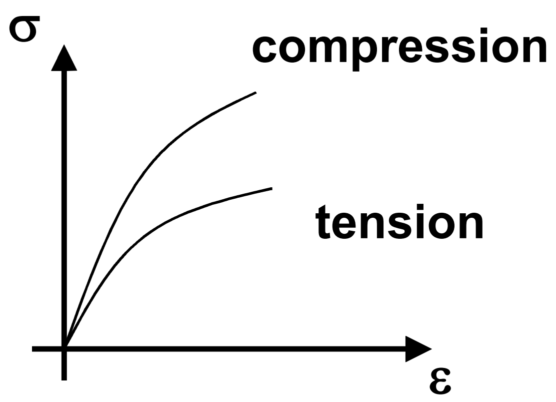




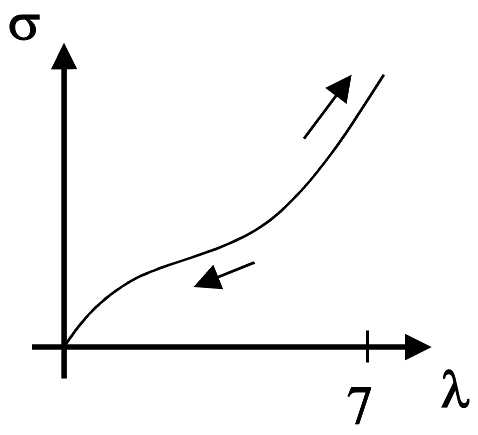
Temperature Loads

MATERIAL NONLINEARITY

- **Nonlinear Elastic**
  - Small strains
  - Different curves for tension and compression
  - After unloading structure is undeformed

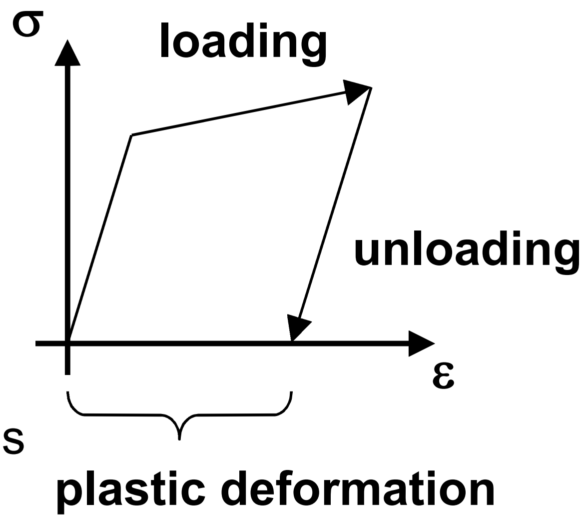


- **Hyperelastic**
  - Normally large strain
  - Poisson’s ratio close to 0.5
  - Mainly rubber

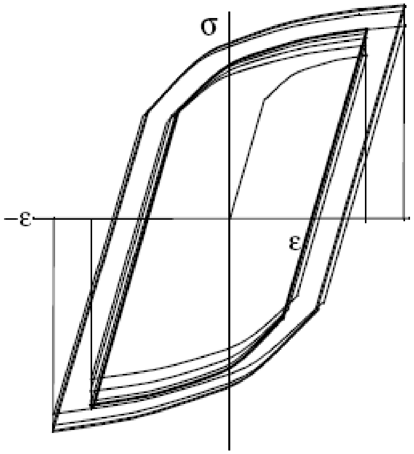


MATERIAL NONLINEARITY

- **Elastic-Plastic**
  - Small and large strain
  - Isotropic, anisotropic, pressure dependent
  - Initial stress and plastic strain

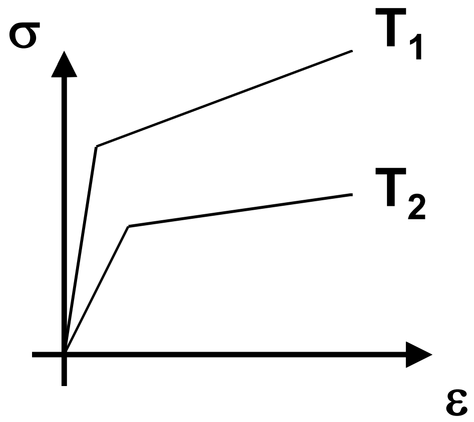


- **Cyclic Plasticity**
  - Yield stress changes with the number of cycles
  - Based on the work of Chaboche

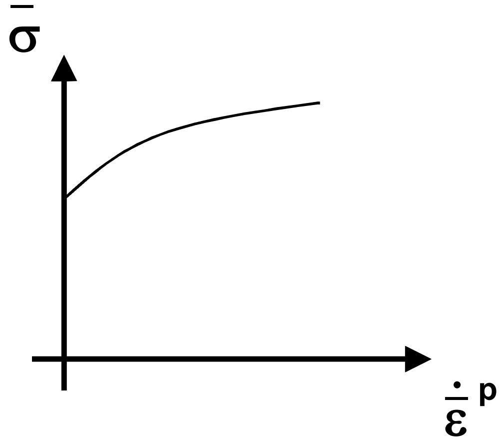


MATERIAL NONLINEARITY

- **Temperature Dependent**
  - Stress–strain curve depends on temperature
  - Applies to each type of material nonlinearity

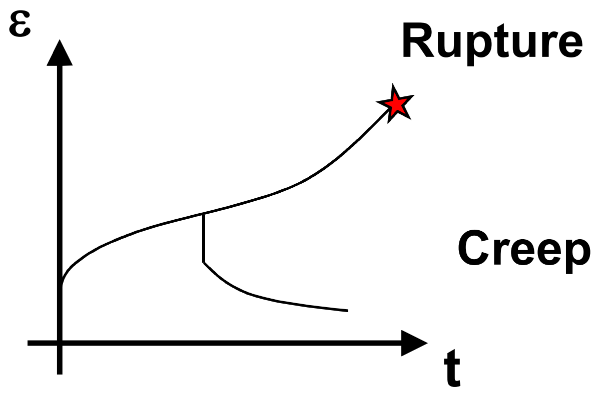


- **Rate Dependent**
  - Yield stress depends on equivalent plastic strain rate
  - Applies to elastic-plastic material

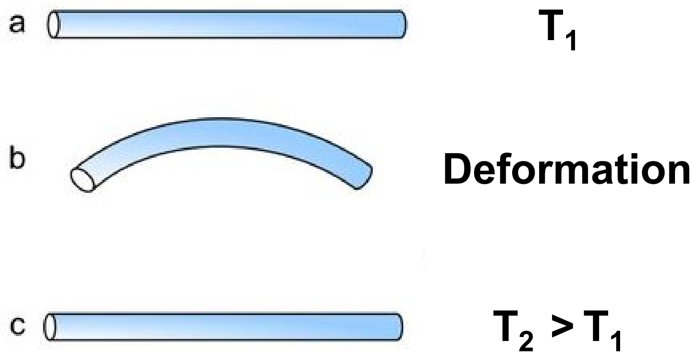


MATERIAL NONLINEARITY

- **Time Dependent**
  - Material properties change with time
  - Creep and relaxation
  - Visco-Elasticity and Visco-Plasticity



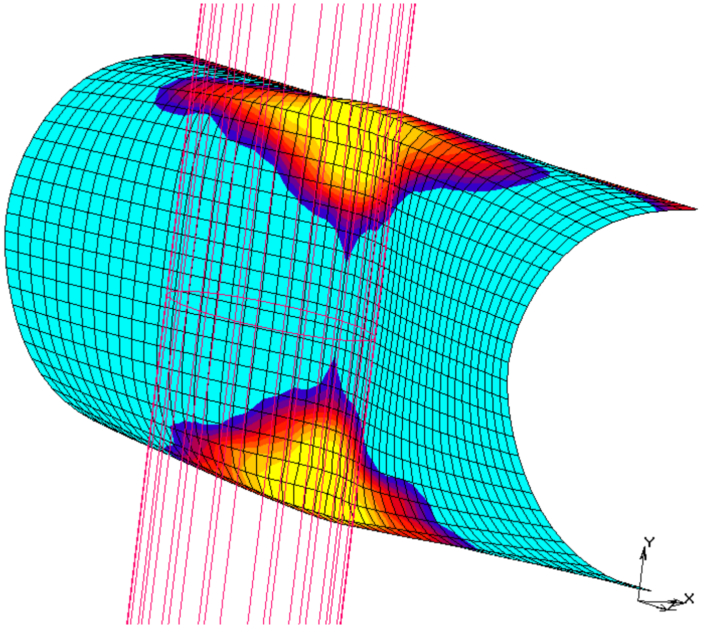
- **Shape Memory**
  - Material properties depend on crystal structure (Martensite & Austenite)
  - Phase changes due to temperature and stresses



MATERIAL NONLINEARITY

- **Progressive failure of composites is a nonlinear phenomenon**
- **Composite Failure Criteria**
  - Maximum Stress
  - Maximum Strain
  - Hill
  - Hoffman
  - Tsai-Wu
  - Hashin
  - Puck
  - Hashin-Tape
  - Hashin-Fabric
  - User defined (UFAIL)

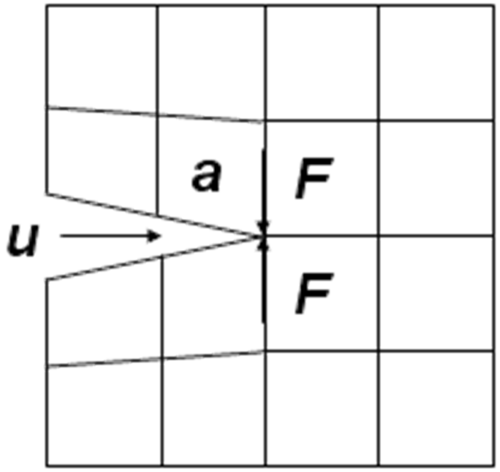
Yellow: means outer ply, fully damaged

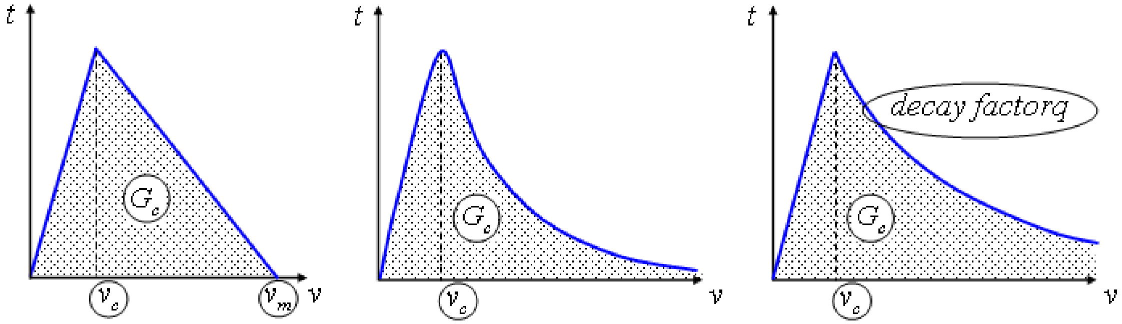


Rigid elliptical cylinder hitting composite shell

MATERIAL NONLINEARITY

- **Fracture**
  - Application: Delamination in Composite Analysis
  - Methods:
    - Virtual Crack Closure Technique (VCCT)
    - Cohesive Zone Modeling





SOL 400 INPUT FILE

- **Nonlinear Iteration Strategy**
  - NLPARM - Nonlinear Parameters for Statics
  - TSTEPNL - Nonlinear Parameters for Transient
  - NLSTEP - New Nonlinear Parameters
- **Geometric Nonlinear Analysis**
  - param, lgdisp, 1
- **Material Nonlinear Analysis**
  - MATS1, for elastic-plastic Material

SOL 400 INPUT FILE EXAMPLE

```
SOL 400
DIAG 8
CEND
TITLE = THIS IS A DEMO INPUT EXAMPLE
SUBCASE 10
   STEP 1
          LOAD = 1
          NLPARM = 110
   STEP 2
          ANALYSIS = NLTRAN
          DLOAD = 3
          TSTEPNL = 130
BEGIN BULK
PARAM,  LGDISP, 1
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
NLPARM  110     25              ITER    1       15      P       NO
+               0.05            -3
MAT1    1       210000.         0.3     7.85-9  1.2-6
MATS1   1               PLASTIC 1000.   1       1       240.
.
```

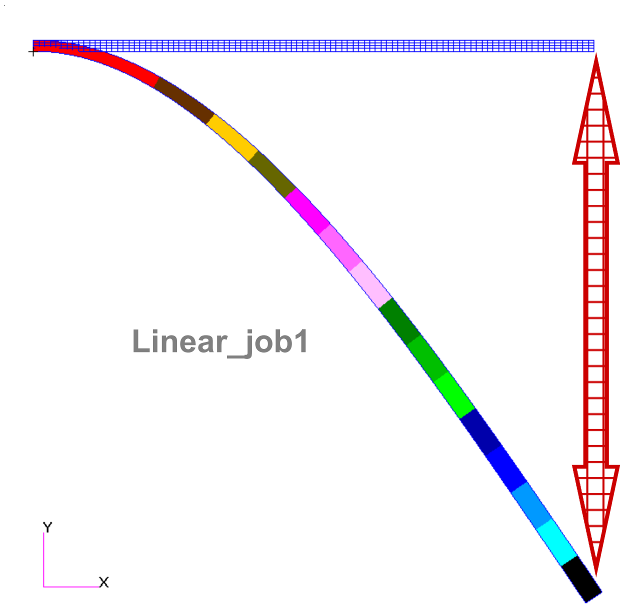
DOCUMENTATION

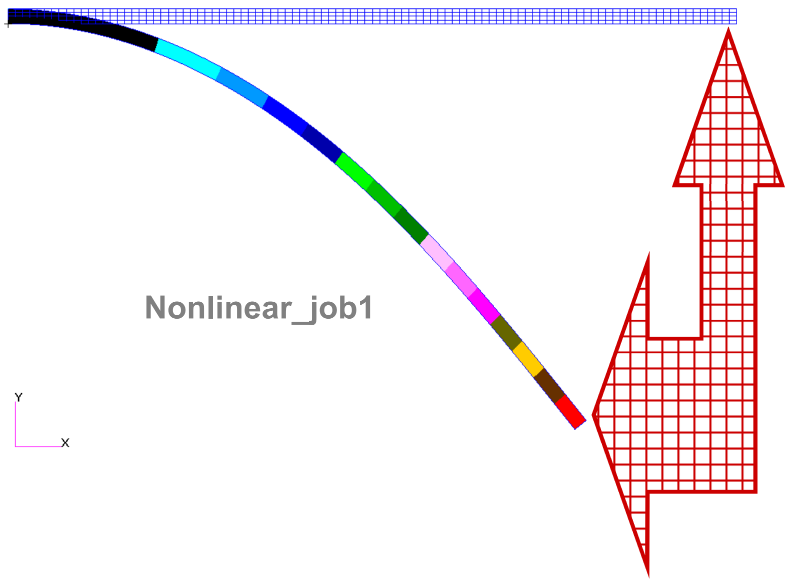
- **MSC Nastran**
  - – MSC Nastran Demonstration Problems Manual
    - contains examples for the efficient use of SOL 400
  - – MSC Nastran Quick Reference Guide
  - – MSC/MD Nastran Release Guides
  - – Implicit Nonlinear User's Guide (SOL 600)

- **Marc User's Manuals**
  - – Volume A – Theory and User Information
  - – Volume B – Element Library
  - – Volume E – Demonstration Problems

EXERCISE

- **Perform Workshop 1: Linear and Nonlinear Analysis of a Cantilever Beam.**





SECTION 3

NONLINEAR SOLUTION STRATEGIES

OVERVIEW

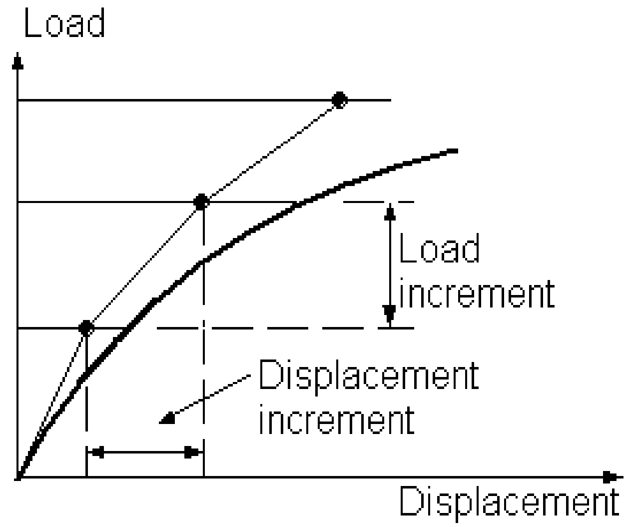
- **Nonlinear Iteration Techniques**
- **Analysis Convergence**
- **Advancing Schemes**
- **Load Incrementation Control**
- **Iteration Parameters**
- **Nonlinear Output Control**
- **Restart**
- **General Guidelines and Limitations**

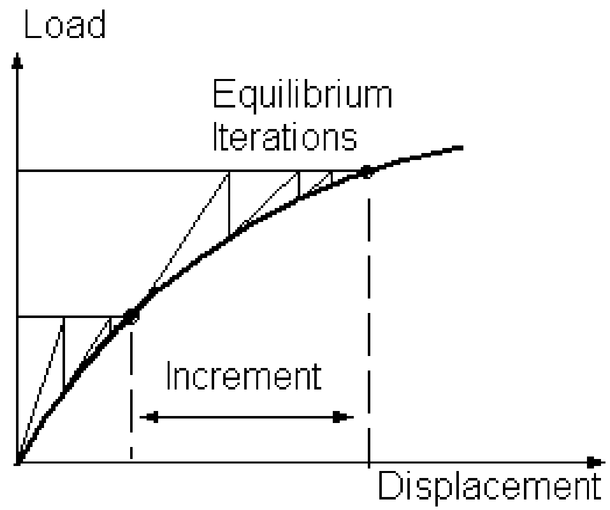
BASIC CONCEPT OF NL ITERATION

- **As discussed in the last section, nonlinear FEA requires the use of iterative solution techniques**
- **MSC Nastran implements several potential method options:**
  - (Full) Newton-Raphson
  - Pure Full Newton Raphson (same as Full but with ‘errors’ calculated based on incremental changes, i.e. not total)
  - Modified Newton Raphson (differing from Full Newton-Raphson by how often the stiffness matrix is reformed)
  - Quasi-Newton (sometimes called a ‘generalization of the secant method’)
  - Secant Method (Mar101 notes state ‘the secant method is based on quasi-Newton update’)
  - Arc length method

ITERATIVE SOLUTION METHODS

- **Pure Incrementation Schemes:**
  - Load applied incrementally
  - No iterative correction
  - Residual eliminated on an incremental basis
  - Path history
  - Stiffness updated incrementally
  - “Drift” from true equilibrium
- **Incremental-Iterative Schemes:**
  - Load applied incrementally
  - Iterative correction to restore incremental equilibrium
  - Path history
  - Stiffness updated incrementally or iteratively
  - Newton-Raphson methods





ITERATIVE SOLUTION METHODS

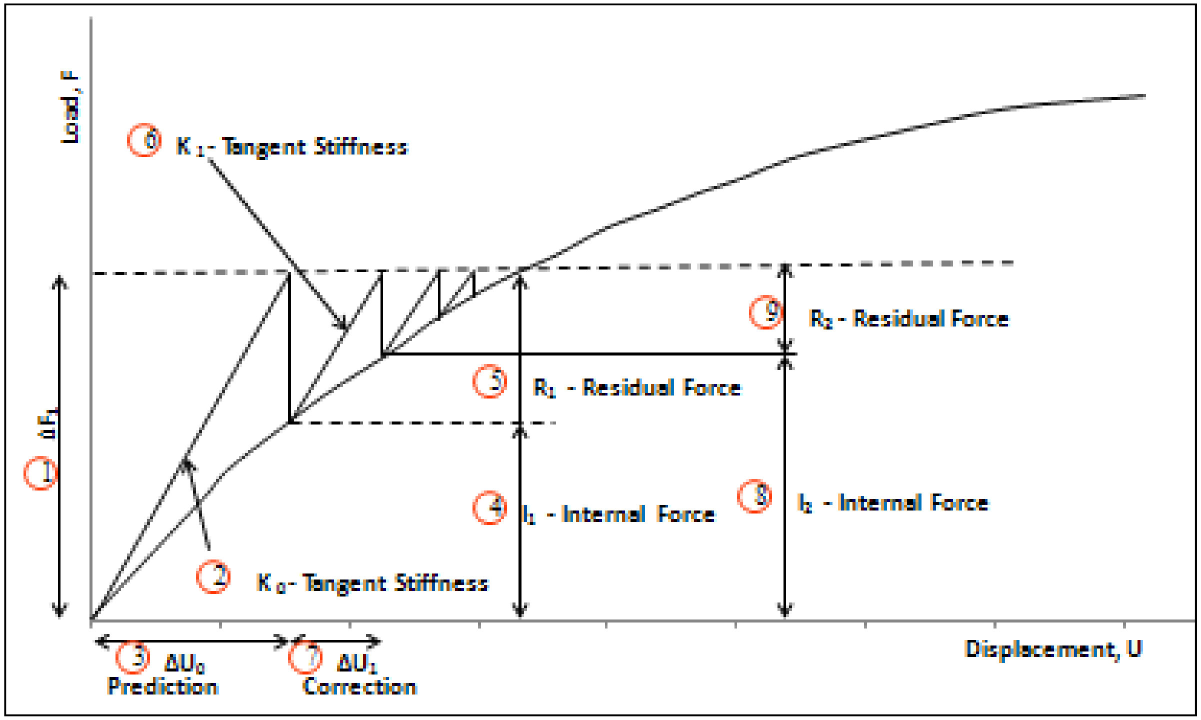
- **For a linear analysis, the solution of equilibrium equations (e.g. Gaussian elimination) can be applied directly in one step**
- **For nonlinear solutions both, the stiffness and external forces may be functions of the nodal displacements.**
- **The aim is to attain equilibrium between the internal forces {I} and the external forces {F}.**
- **For the solution step, we must solve the equations:**
    - {I} – {F} = Out of balance force vector = 0

    or

    - [K] {u} - {F} = Out of balance force vector = 0
- **It is not practical to have a zero out of balance force, therefore it is limited to a small user specified value called the residual force, {R}.**
    - [K] {u} - {F} = Residual force vector = {R}
- **To solve such a nonlinear set of equations, Marc uses a Newton Raphson procedure (default). This is an incremental-iterative method.**

THE NEWTON RAPHSON METHOD

- **I(u) – F(u) = R**
  - Where R is a small user specified out of balance load called the Residual Load
- **Steps 1- 5 are the predictive stages.**
- **Steps 6 - 9 are the corrective (iterative) stages.**



THE NEWTON RAPHSON METHOD

- The implementation of the Newton Raphson method typically uses a Taylor series expansion about the current position to estimate the displacement correction direction and magnitude.

$$\{F\} = \{0\} + \frac{\partial\{F\}}{\partial\{u\}}\{\Delta u\} + \frac{\partial^2\{F\}}{\partial\{u\}^2}\{\Delta u\}^2$$

- Drop quadratic term and compute displacement correction {Δu} .

$$\frac{\partial\{F\}}{\partial\{u\}} = [K_T] = Tangent\ \ Stiffness$$

- Define tangent stiffness matrix:

$$\{F\} = [K_T]\{\Delta u\}$$

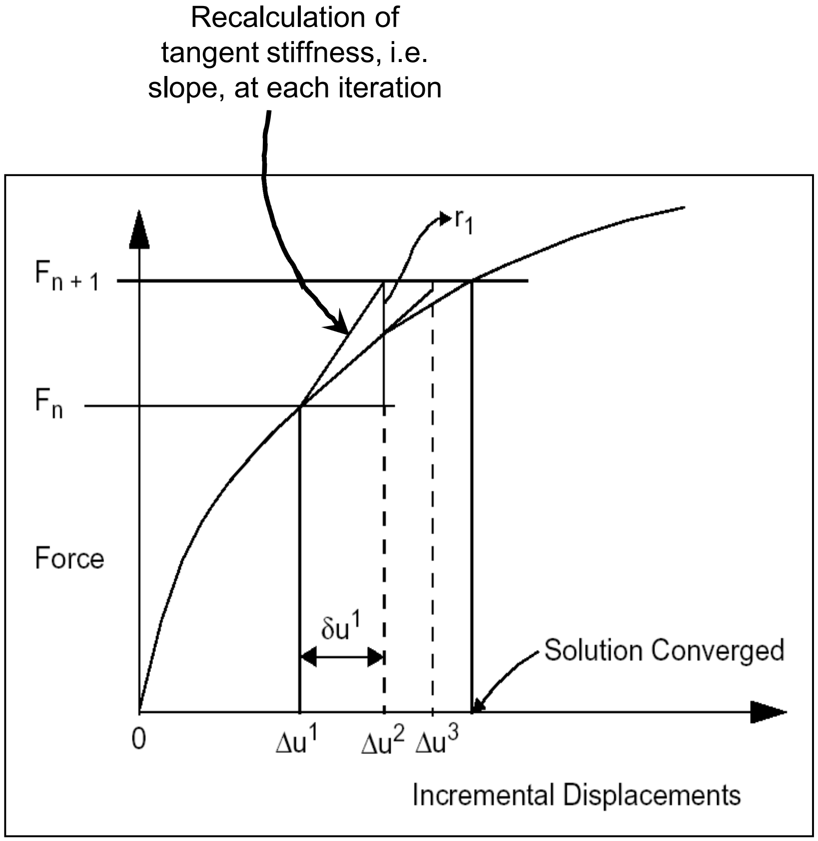
- It relates small changes in load to changes in displacement.

$$\{\Delta u\} = [K_T]^{-1}\{F\}$$

- The procedure involves the computation of {Δu} for a given {F}
- Stop iterations when the residual ({I} – {F}) is equal to the user specified tolerance.

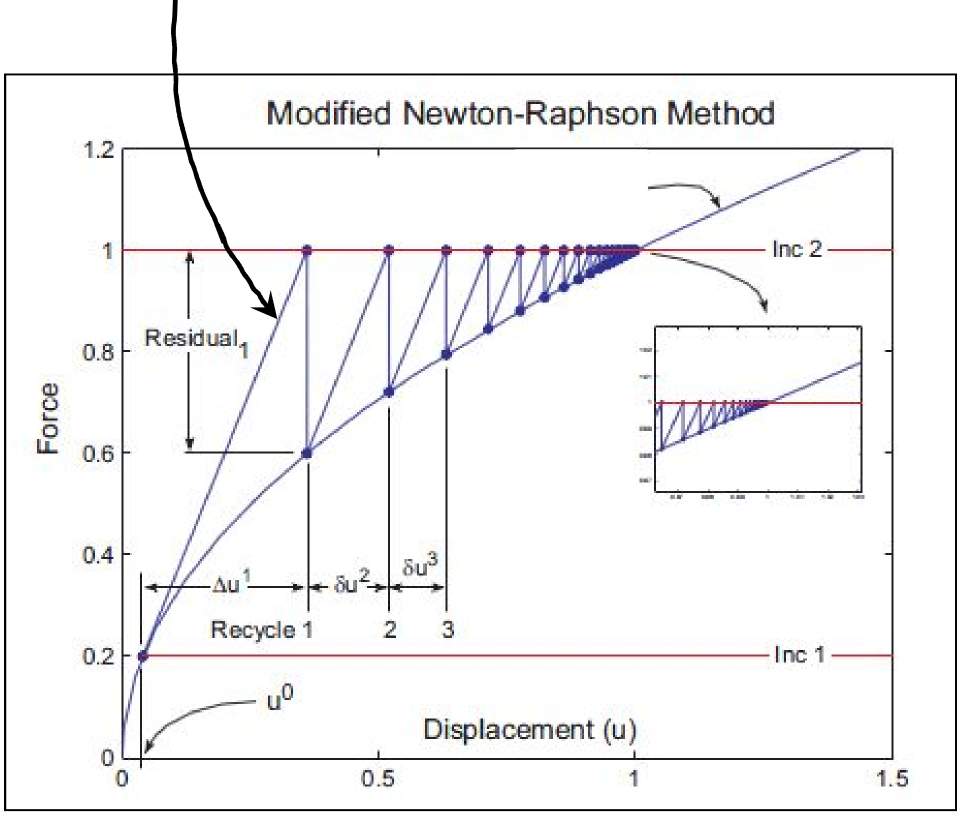
THE NEWTON RAPHSON METHOD

- Quadratic rate of convergence
- Needs few iterations to converge
- Evaluation and (maybe) inversion of tangent stiffness matrix at each iteration
- Expensive for large systems
- Less likely to converge to an unstable solution
- Recommended (default) method
- Recommended for GNL (geometric non linear) solutions
- May fail under extreme material nonlinearity (e.g. brittle cracking)


Recalculation of tangent stiffness, i.e. slope, at each iteration

THE MODIFIED NEWTON RAPHSON METHOD

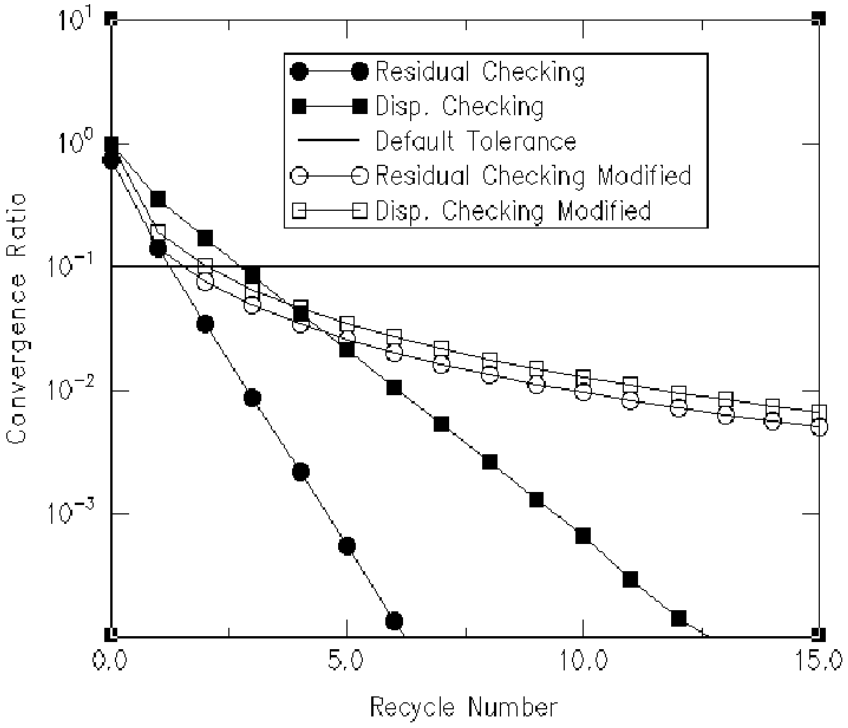
- Evaluation and inversion of tangent stiffness matrix only at start of each increment
- Slow convergence behavior
- More iterations to converge
- Computationally inexpensive per iteration
- May be essential for extreme material nonlinearity
- May be assisted using additional “iterative acceleration” techniques
- Suitable for mildly nonlinear problems



Note for Modified Newton Raphson same slope for all iterations – i.e. **no** recalculation of tangent stiffness

MODIFIED VS. FULL NEWTON CONVERGENCE RATES

- The graph shows the residual and displacement convergence norms during a nonlinear increment for NR and MNR.
- MNR has decreasing convergence rate behaviour.
- For a highly nonlinear application, MNR will be much slower than NR.

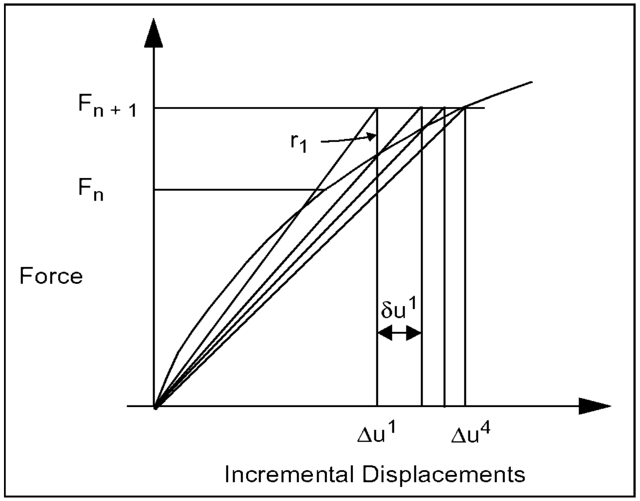


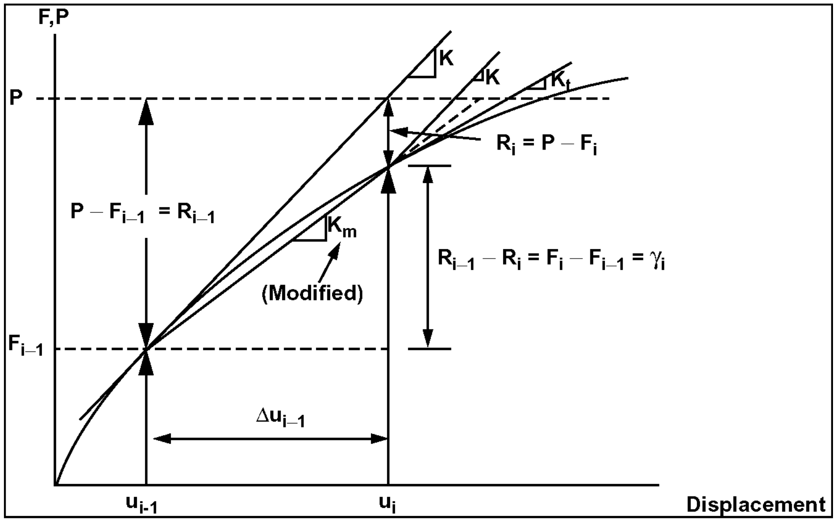
OTHER ITERATIVE SOLUTION METHODS

- **Strain Correction Method**
  - A variant of the full Newton method.
  - Appropriate for shell and beam problems in which rotations are large, but membrane stresses are small.
  - In such cases, rotation increments are usually much larger than the strain increments, which cause nonlinear terms to dominate linear terms.
  - The iterative procedures start with a fully linearized calculation. This means that the nonlinear contributions yield strain increments inconsistent with the calculated displacement increments in the first iteration. These errors give rise to either:
    - Incorrect plasticity calculations (when using small strain plasticity method).
    - Or, in the case of elastic material behaviour, yields erroneous stresses. These stresses have a dominant effect on the stiffness matrix for subsequent iterations or increments, which then causes relatively poor performance.
  - This method uses a linearized strain calculation, with the nonlinear portion of the strain increment applied as an initial strain increment in subsequent iterations.

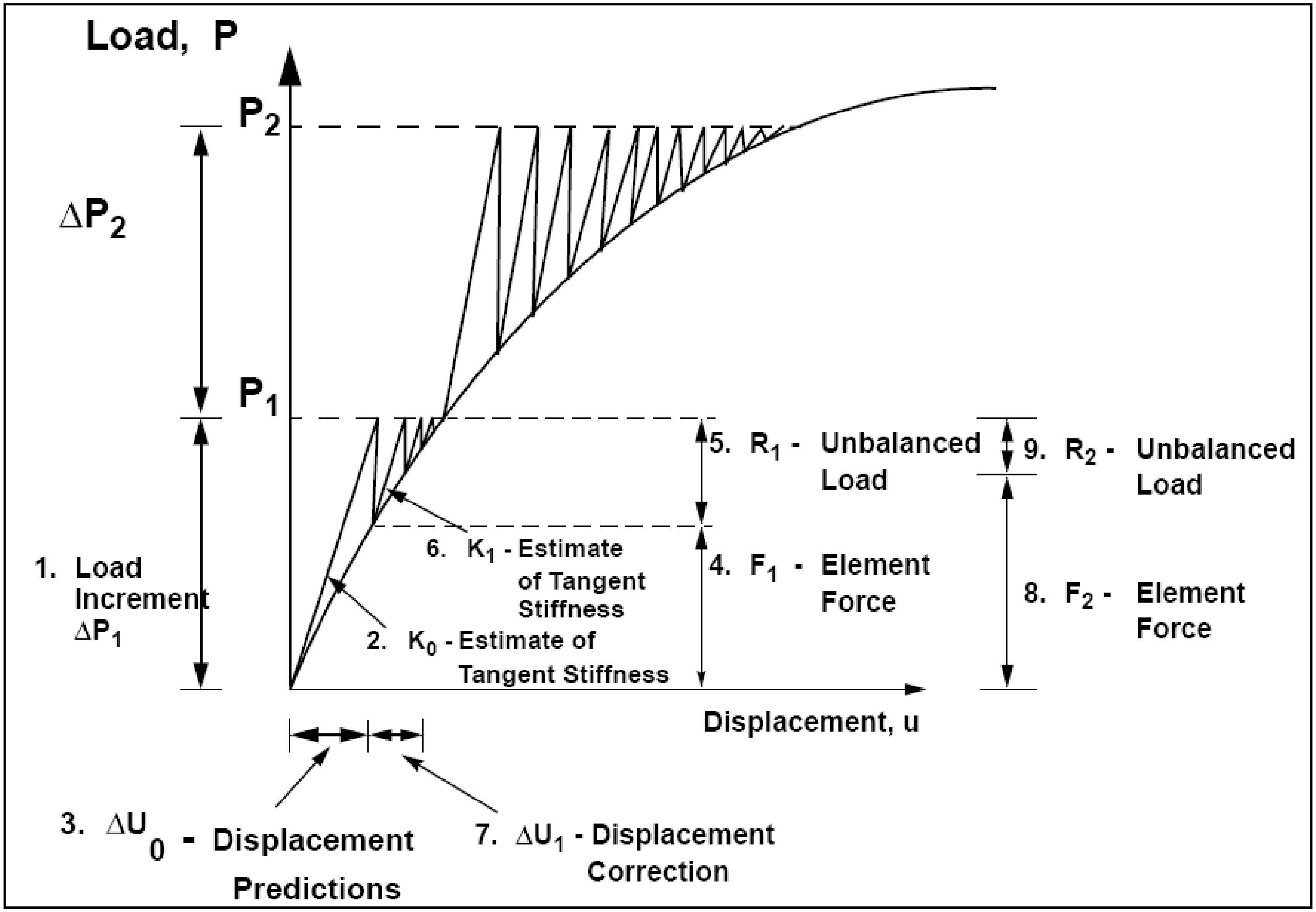
OTHER ITERATIVE SOLUTION METHODS

- **Secant Method**
  - The secant method is similar to the modified Newton-Raphson method.
  - The secant method is based on the Davidon-rank one, quasi-Newton update.
  - The residual is then modified to improve the rate of convergence.
  - When the iterative solver is employed, simple back substitution is not possible, making this process ineffective. Instead, use the full Newton-Raphson method.
  - The secant method is not available through Mentat.





BASIC CONCEPT OF NL ITERATION



BASIC CONCEPT OF ITERATION

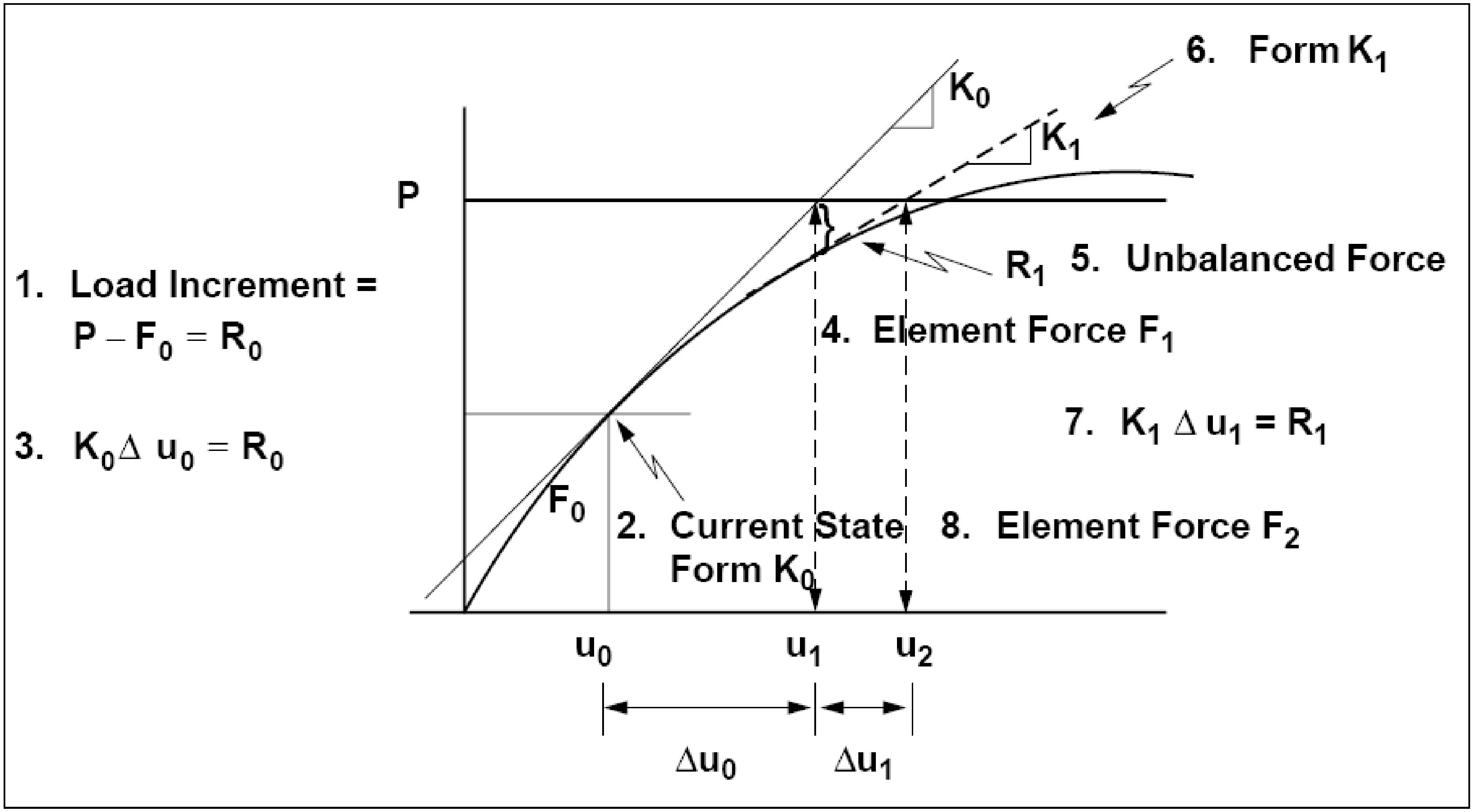
- **Multistep Approach**
  - Determine an increment (e.g. load, displacement or arc length)
  - Determine an estimate of the tangent stiffness matrix (default). The user can request that the stiffness matrix remains the same or will be updated exactly (tangent stiffness matrix).
  - Determine the displacement increment
  - Calculate the element forces
  - Calculate the unbalanced load and check for convergence. If converged, go to Step 1

BASIC CONCEPT OF ITERATION

- **If not converged, continue as follows:**
  - Determine an estimate of tangent stiffness matrix
  - Determine the displacement increment due to the unbalanced load
  - Calculate the element resisting forces
  - Calculate the unbalanced load and check for convergence. If converged, go to Step 1. If not converged, go to Step 6 or start the **divergence** procedure (explained later).

FULL NEWTON RAPHSON METHOD

- **Stiffness Update at each iteration**



1. Load Increment = $P - F_0 = R_0$

2. Current State Form K_0

3. $K_0 \Delta u_0 = R_0$

4. Element Force F_1

5. Unbalanced Force

6. Form K_1

7. $K_1 \Delta u_1 = R_1$

8. Element Force F_2

FULL NEWTON RAPHSON METHOD

- **Mathematical Algorithm**

Solve: $K(u_0)\Delta u_0 = P - F_0 = \Delta R(u_0)$

$u_1 = u_0 + \Delta u_0$

Solve: $K(u_1)\Delta u_1 = R(u_1)$

$u_2 = u_1 + \Delta u_1$

Solve: $K(u_2)\Delta u_2 = R(u_2)$

$u_3 = u_2 + \Delta u_2$

.
.
.

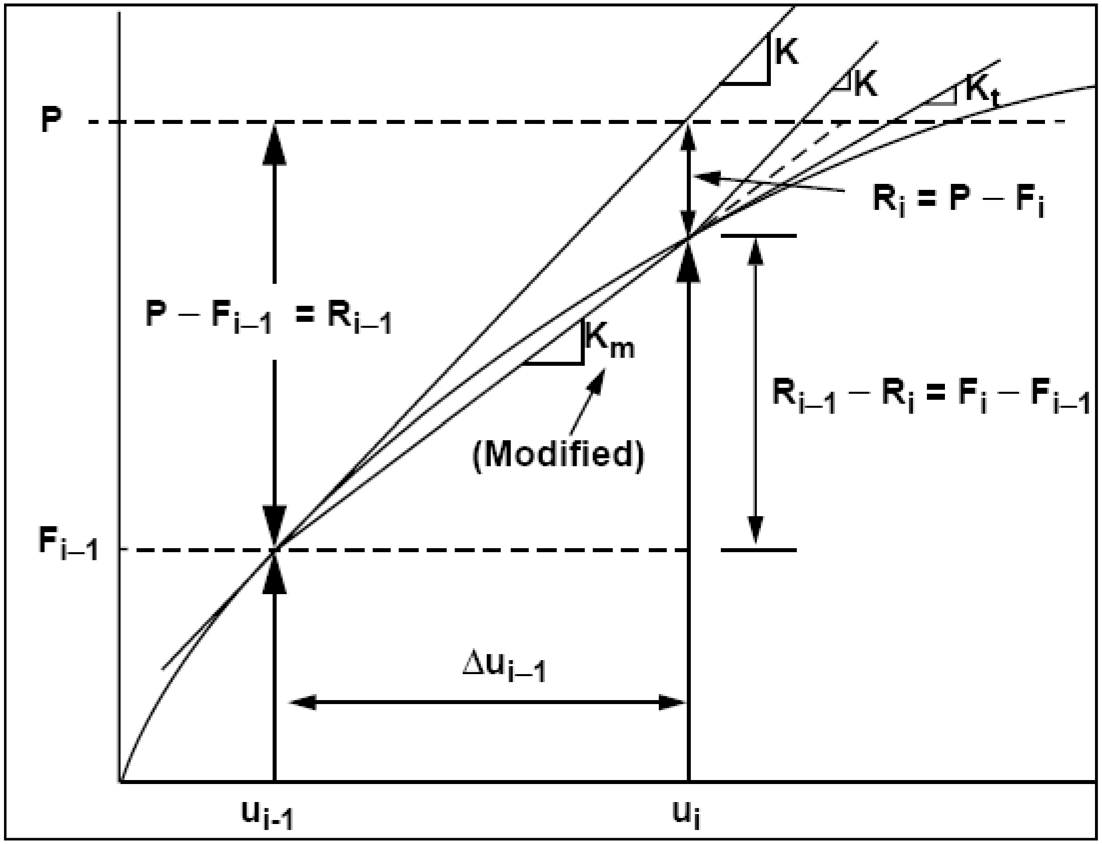
until reaching convergence

QUASI NEWTON METHOD

- **Approximate Stiffness Update at each iteration based on BFGS Method (implies only 8 vector products instead of generation, assembly and decomposition of the exact stiffness matrix)**
- **Limited to the maximum number of Quasi Newton vectors MAXQN**
- **The exact stiffness matrix will only be updated when the convergence is expected to be faster and in case of divergence**

QUASI NEWTON METHOD

- **Approximate Stiffness Concept for 1 DOF**



Secant Stiffness

$$K_m = \frac{R_{i-1} - R_i}{\Delta u_{i-1}}$$

$$= K - \frac{R_i}{\Delta u_{i-1}}$$

$$= K - K_s$$

spring in the direction of unbalanced force

LINE SEARCH

- **Improves the displacements for an iteration**

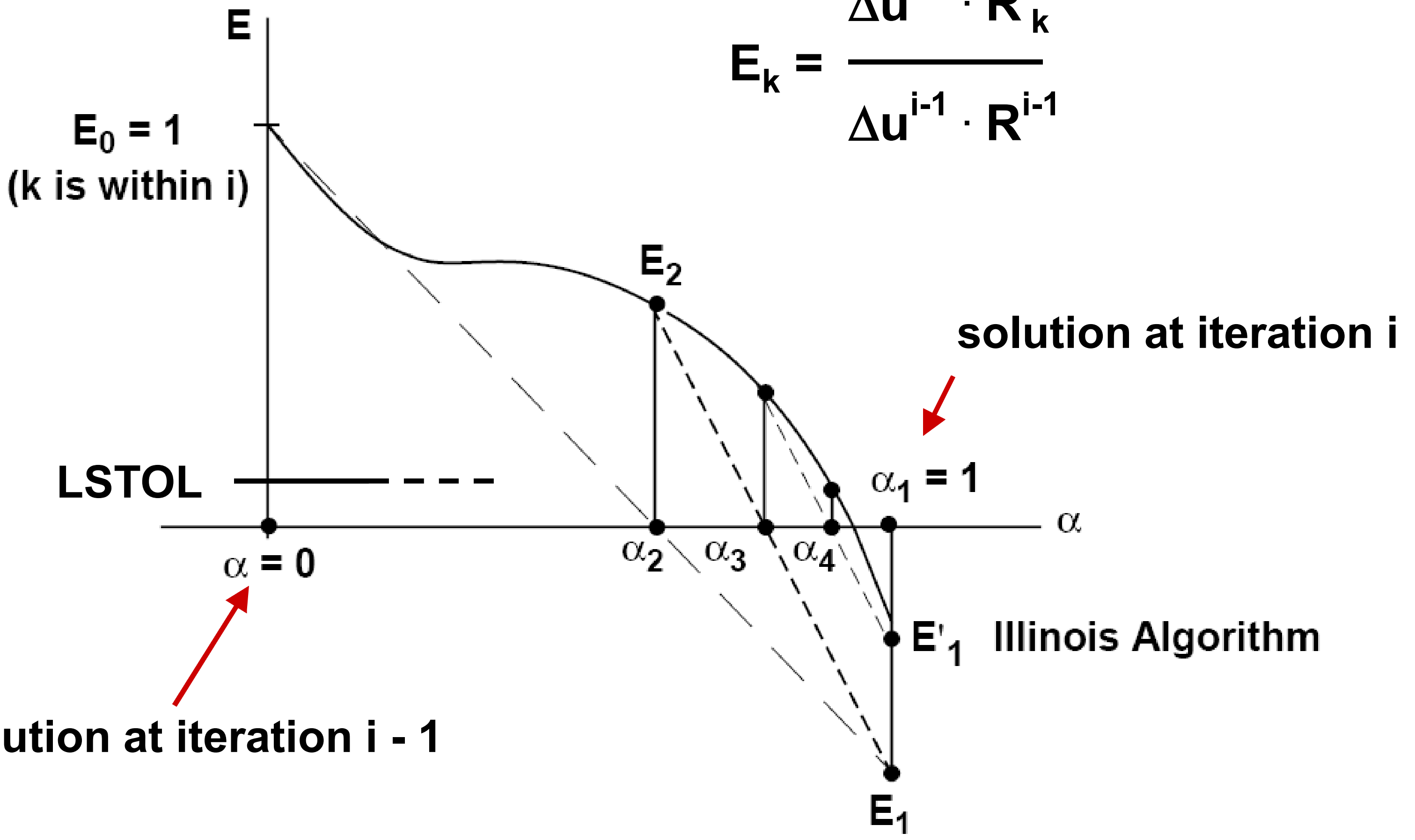
$$\mathbf{u}_i = \mathbf{u}_{i\text{-}1} + \alpha \cdot \Delta\mathbf{u}_i$$

  - Finds the Line Search Factor a at which the error function E(a) falls below the Line Search Tolerance LSTOL (for E see next page)
  - The Line Search Factor a is 0 at the start of iteration i and 1 at its end
  - When E(0) and E(1) have the same sign, extrapolation will be used
  - Otherwise interpolation will be used

LINE SEARCH

- **Interpolation**

$$E_k = \frac{\Delta u^{i-1} \cdot R^{i}_{k}}{\Delta u^{i-1} \cdot R^{i-1}}$$



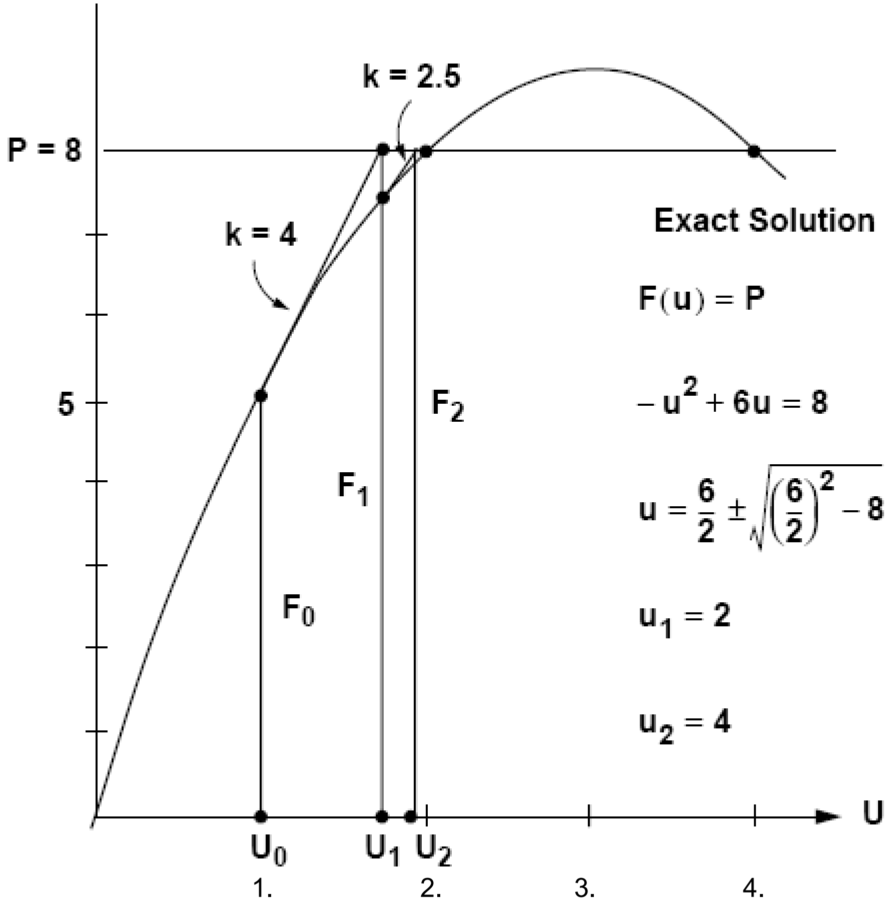
solution at iteration i

Illinois Algorithm

solution at iteration i - 1

PERFORMANCE COMPARISON

- **Example: simple quadratic function**



Exact Solution

$$F(u) = P$$

$$-u^2 + 6u = 8$$

$$u = \frac{6}{2} \pm \sqrt{\left(\frac{6}{2}\right)^2 - 8}$$

$$u_1 = 2$$

$$u_2 = 4$$

R < 0.01

PERFORMANCE COMPARISON

- **Full Newton-Raphson Method (quadratic rate of convergence)**

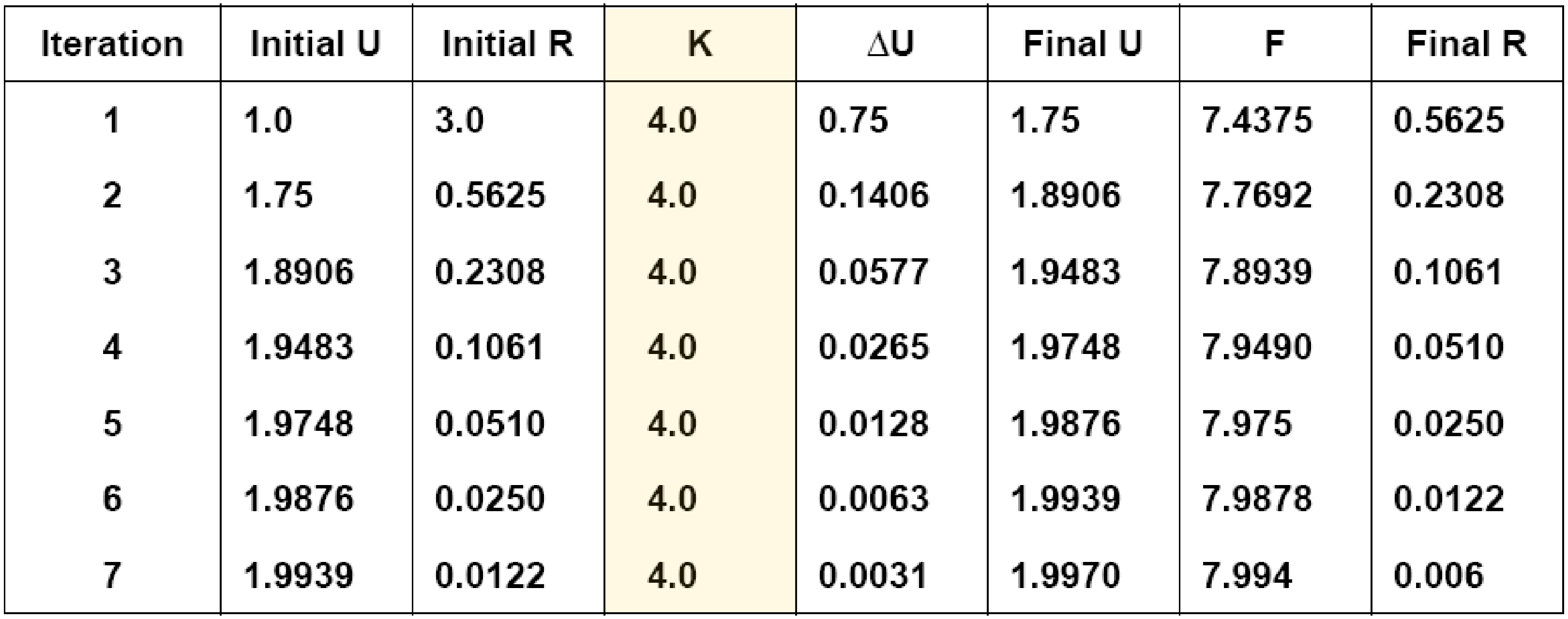
Iteration	Initial U	Initial R	K	ΔU	Final U	F	Final R
1	1.0	3.0	4.0	0.75	1.75	7.4375	0.5625
2	1.75	0.5625	2.5	0.225	1.975	7.9494	0.0506
3	1.975	0.0506	2.05	0.0247	1.997	7.9940	0.006

- **Quasi Newton Method**

Iteration	Initial U	Initial R	K	ΔU	Final U	F	Final R
1	1.0	3.0	4.0	0.75	1.75	7.4375	0.5625
2	1.75	0.5625	3.25	0.1731	1.9231	7.8403	0.1597
3	1.9231	0.1597	2.327	0.0686	1.9917	7.9833	0.0167
4	1.9917	0.0167	2.084	0.008	1.9997	7.9994	0.0006

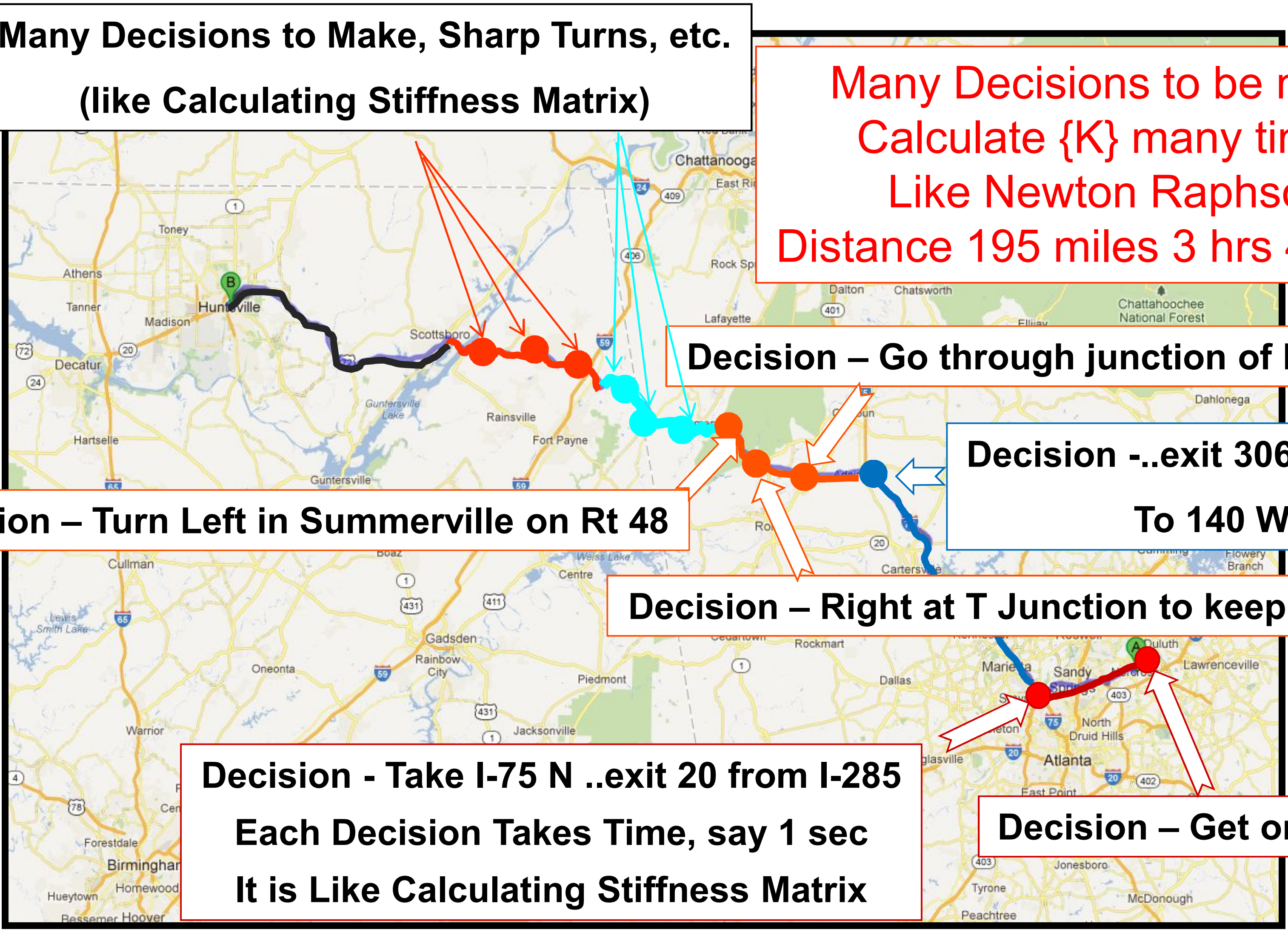
PERFORMANCE COMPARISON

- **In contrast, Modified Newton-Raphson (linear rate of convergence)**
  - No stiffness update during an increment



Iteration	Initial U	Initial R	K	ΔU	Final U	F	Final R
1	1.0	3.0	4.0	0.75	1.75	7.4375	0.5625
2	1.75	0.5625	4.0	0.1406	1.8906	7.7692	0.2308
3	1.8906	0.2308	4.0	0.0577	1.9483	7.8939	0.1061
4	1.9483	0.1061	4.0	0.0265	1.9748	7.9490	0.0510
5	1.9748	0.0510	4.0	0.0128	1.9876	7.975	0.0250
6	1.9876	0.0250	4.0	0.0063	1.9939	7.9878	0.0122
7	1.9939	0.0122	4.0	0.0031	1.9970	7.994	0.006

ATLANTA – HUNTSVILLE – NEWTON-RAPHSON



Many Decisions to Make, Sharp Turns, etc.

(like Calculating Stiffness Matrix)

Many Decisions to be made
Calculate {K} many times
Like Newton Raphson
Distance 195 miles 3 hrs 45 mins

Decision – Go through junction of Hwy Rt 53

Decision -..exit 306 from I-75

To 140 W

Decision – Turn Left in Summerville on Rt 48

Decision – Right at T Junction to keep on Rt 140

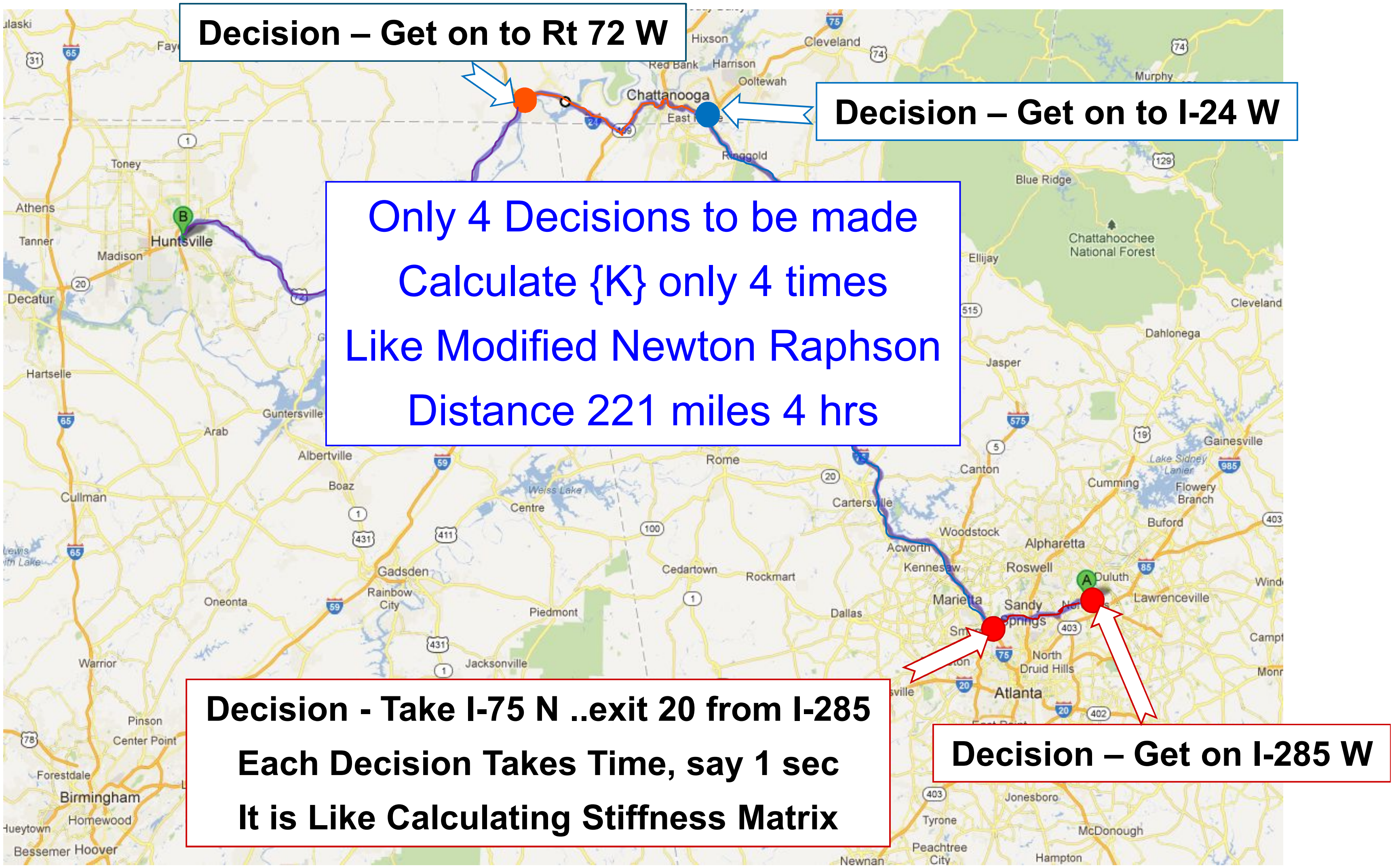
Decision - Take I-75 N ..exit 20 from I-285

Each Decision Takes Time, say 1 sec

It is Like Calculating Stiffness Matrix

Decision – Get on I-285 W

ATLANTA – HUNTSVILLE – MODIFIED NEWTON-RAPHSON



Decision – Get on to Rt 72 W

Decision – Get on to I-24 W

Only 4 Decisions to be made

Calculate {K} only 4 times

Like Modified Newton Raphson

Distance 221 miles 4 hrs

Decision - Take I-75 N ..exit 20 from I-285

Each Decision Takes Time, say 1 sec

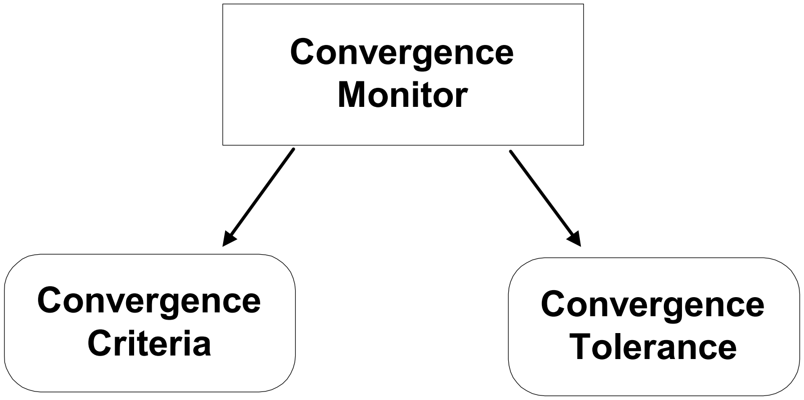
It is Like Calculating Stiffness Matrix

Decision – Get on I-285 W

ANALYSIS CONVERGENCE

ANALYSIS CONVERGENCE

- **Implementation of previous discussion in MSC Nastran**
- **Each load increment requires ‘Convergence’**
- **Two aspects to the convergence control**
  - The convergence criteria
    - Residual Norm
    - Displacement Norm
    - Strain Energy Norm
  - The convergence tolerance
    - Threshold value below which convergence is deemed to have occurred



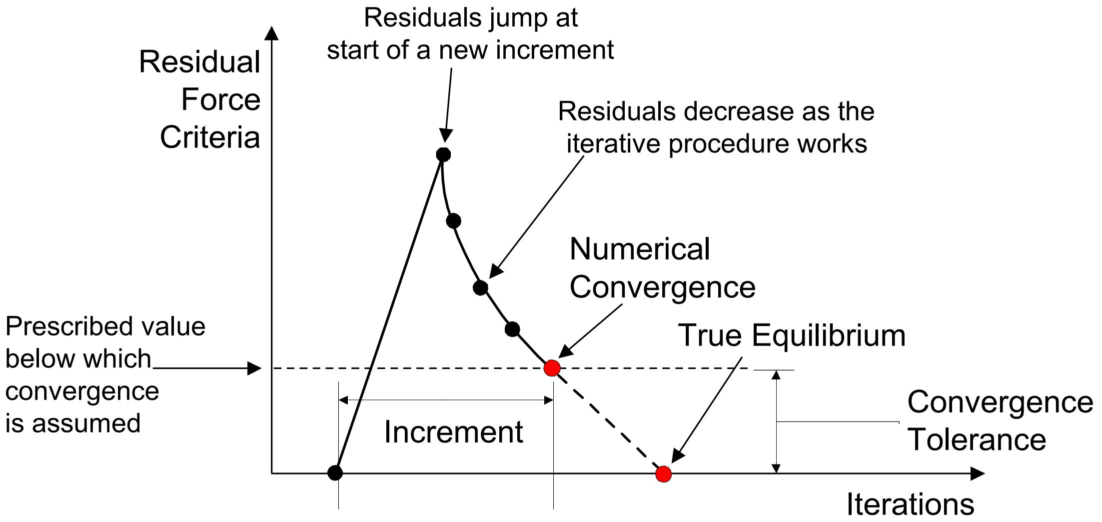
Convergence Monitor

Convergence Criteria

Convergence Tolerance

ANALYSIS CONVERGENCE

- **The convergence criteria monitor the extent to which the iterative procedure has reached equilibrium state.**
- **Termination of the iterative process when the convergence ratio is less than the specified tolerance (default 0.1 for residual and displacement).**
- **Too 'slack' a tolerance gives a false state of equilibrium. In this case, the reference state used during the iterative procedure can "drift" from equilibrium and may cause the material response to differ from the true response.**
- **Too 'tight' a tolerance results in unnecessary iterations.**



CONVERGENCE CRITERIA

- **For SOL400 Several Convergence criteria are available**
  - Relative Residual Criteria
    - Ensure that the out-of-balance force is much smaller than the external forces.
  - Relative Displacement Criteria
    - Ensure that the maximum displacement of the last iteration is small compared to the maximum displacement of the increment.
  - Relative Strain Energy Criteria
    - To ensure that the iterative strain energy is small compared to the strain energy of the increment.

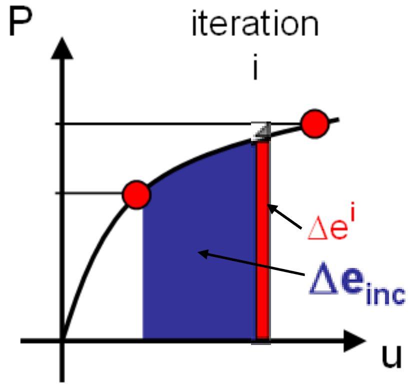
CONVERGENCE CRITERIA

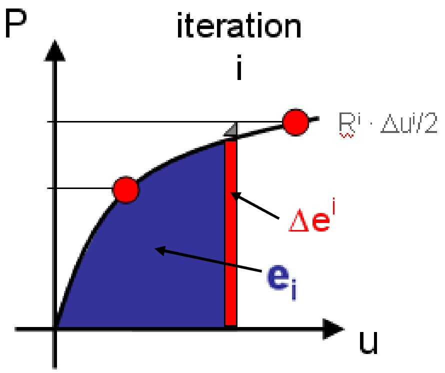
- **Because Displacements and Residual Forces are vector quantities there are three methods to compute the 'ERROR' criteria for them:**
  - Weighted average method
    - the errors are computed using the weighted average of all DOF's of the model. Default.
  - Vector component method
    - convergence checking is performed on the maximum vector component of all DOF's in the model.
  - Length method
    - the length of a vector at a grid point is first computed by the SRSS method. Then the error checking is performed on the maximum length of all grid points in the model.

CONVERGENCE CRITERIA

- **For displacements and energy the calculation of the convergence criteria quantities can be relative w.r.t the increment or absolute w.r.t the actual solution, i.e., for energy:**

$$\frac{\Delta e^{i}}{\Delta e_{inc}} < \mathbf{EPSW} \quad \text{or} \quad \frac{\Delta e^{i}}{e_{i}} < \mathbf{EPSW}$$

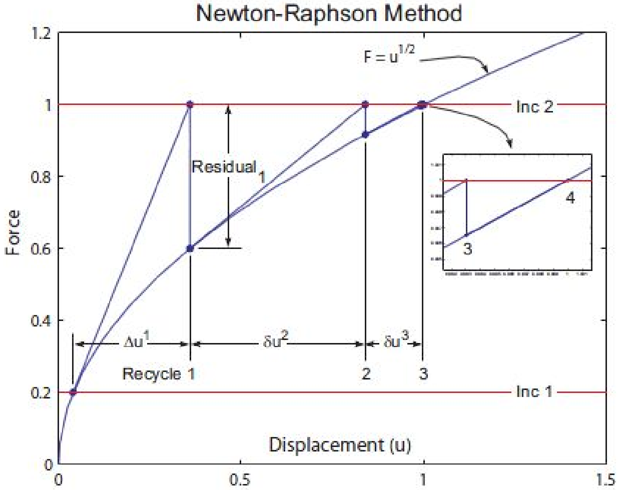




CONVERGENCE CRITERIA

- **Relative Residual Criteria**
  - Aim: Ensure that the out-of-balance force much smaller than the external forces.

$$\frac{\|F_{residual}\|_\infty}{\|F_{reaction}\|_\infty} < TOL_1$$



  - $\|F_{residual}\|$ is the internal (out of balance) force vector and represents the component with the highest absolute value.
  - $\|F_{reaction}\|$ is the external reaction force vector and represents the component with the highest absolute value.
  - TOL1 is the user specified convergence tolerance (default 0.1).
  - Moments can be included in a similar way according to:

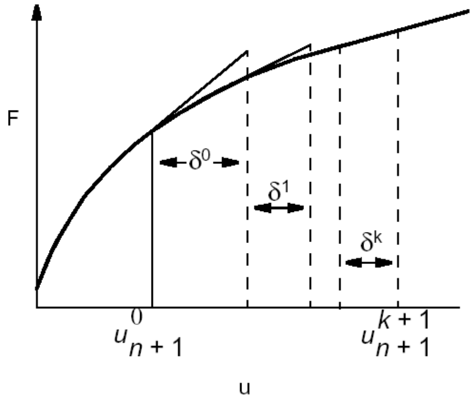
$$\frac{\|F_{residual}\|_\infty}{\|F_{reaction}\|_\infty} < TOL_1 \text{ and } \frac{\|M_{residual}\|_\infty}{\|M_{reaction}\|_\infty} < TOL_2$$

    - where ||M|| represents the moment vectors
  - TOL2 is the user specified convergence tolerance (0.1).

CONVERGENCE CRITERIA

- **Relative Displacement Criteria**
  - Aim: Ensure that the maximum displacement last iteration is small compared to the maximum displacement of the increment/solution

$$\frac{\|\delta u\|_\infty}{\|\Delta u\|_\infty} < TOL_1$$



  - ||∂u|| is the maximum correction to the incremental displacement vector (iterative displacement).
  - ||Δu|| is the maximum incremental/solution displacement.
  - TOL1 is the user specified convergence tolerance (default 0.1).
  - If TOL1 < 0 ERROR is calculated relative to Δu of increment
  - If TOL1 > 0 ERROR is calculated relative to Δu of solution
  - Rotations can be included in a similar way according to:

$$\frac{\|\delta u\|_\infty}{\|\Delta u\|_\infty} < TOL_1 \text{ and } \frac{\|\delta \phi\|_\infty}{\|\Delta \phi\|_\infty} < TOL_2$$

where ||ϕ|| represents the rotations

  - TOL2 is the user specified convergence tolerance (0.1).
  - A disadvantage of this approach is that it results in at least one iteration, regardless of the accuracy of the solution.

CONVERGENCE CRITERIA

- **Relative Strain Energy Criteria**
  - Aim: To ensure that the iterative strain energy is small compared to the strain energy of the increment.
  - With this method, the entire model is checked since the energies are the total energies, integrated over the whole volume.

$$\frac{\delta E}{\Delta E} < TOL_1$$

  - ∂E is the iterative change in strain energy.
  - ∆E is the incremental strain energy.
  - If $TOL_1 < 0$ ERROR is calculated relative to Δu of increment
  - If $TOL_1 > 0$ ERROR is calculated relative to Δu of solution
  - TOL1 is the user specified convergence tolerance (default 0.1).
  - A disadvantage of this approach is that it results in at least one iteration, regardless of the accuracy of the solution.
  - The advantage of this method is that it evaluates the global accuracy as opposed to the local accuracy associated with a single node.

CONVERGENCE CRITERIA - SUMMARY

Convergence Mode	**No V and N used in conjunction with UPW**	**V used in conjunction with UPW**	**N used in conjunction with UPW**
Displacement checking : **U**	EPSU=|$K^{1/2}$ddu| / |$K^{1/2}$du| K – diag. stiffness ddu – iterative disp du – total disp if EPSU > 0 du – incremental disp if EPSU < 0	EPSU=max(ddu) / max(du) ddu – iterative disp at DOF du – incremental disp at DOF	EPSU=max|ddu| / max|du| |ddu| – iterative disp magnitude |du| – incremental disp magnitude
Residual checking : **P**	EPSP=|R*U| / |P*U| R – residuals P – external loads U – total disp	EPSP = max(R) / max (F) R = residual at DOF F = reaction at DOF Separate checks for forces and moments	EPSP = max|R| / max |F| |R| = residual magnitude |F| = reaction magnitude Separate checks for forces and moments
Energy Checking : **W**	EPSW=|F ddu| / E F – internal forces ddu – iterative disp. E – total energy if EPSW > 0 E – incremental energy if EPSW < 0	EPSW=max|F ddu|/max |Fdu| F – internal forces ddu – iterative disp du – incremental disp	Same as V checking

RELATIVE CONVERGENCE CRITERIA ISSUES

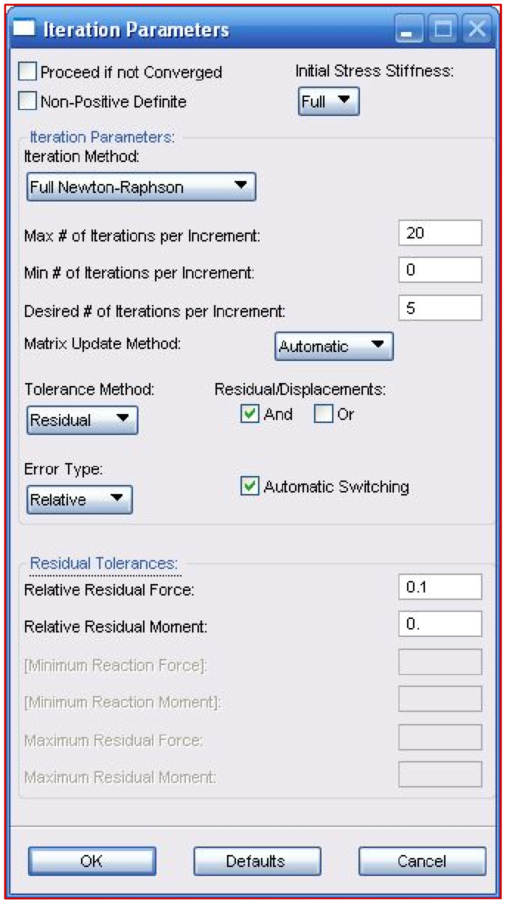
- **Some analyses cause the relative convergence criteria to become meaningless:**
  - Maximum displacement increment becomes too small.
    - maximum displacement change at node 3 degree of freedom 1 is equal to 2.3E-13
    - maximum displacement increment at node 2 degree of freedom 1 is equal to 6.9E-09
    - displacement convergence ratio 3.462
  - Maximum reaction force becomes too small.
    - maximum residual force at node 2311 degree of freedom 1 is equal to 6.058E-08
    - maximum reaction force at node 2294 degree of freedom 2 is equal to 1.460E-08
    - residual convergence ratio 4.148
  - Extremely small strain energy density.
- **Each of these scenarios represent the denominator of the convergence test becoming (numerically) too small to handle. That is, dividing any numerator by, say, 1e^{-10} will cause a large convergence norm.**
- **This gives the wrong impression that the analysis is not converging.**
- **Remedies:**
  - Specify Auto-Switch (see later)
  - Remove offending criteria
  - Tighten remaining criteria

RELATIVE CONVERGENCE CRITERIA ISSUES

Analysis Type	Convergence Variable		
	Displacement/ Rotation	Residual Force/ Torque	Strain Energy
Stress-free motion	Yes	No	No
Springback	No	Yes	No
Free Thermal Expansion	Yes	No	No
Constraint Thermal Expansion	No	Yes	Yes
Yes – Relative tolerance testing works well No - Should use auto-switching also.			

AUTOMATIC CRITERIA SWITCHING

- **Remedy to meaningless convergence criteria.**
- **Automatically detects small values in convergence calculations, and**
  - Switches to residual checking, if displacement increments become very small (Max_Disp_Inc./Smallest_Elem_Size < 1E-06).
  - Switches to displacement checking, if reaction forces become very small (<1E-08).
  - Switches to energy checking if structure is free of stress and deformation (strain energy density <1E-15).
- **Automatic switching is only available for relative testing (only testing type available in SOL400)**



☐ Proceed if not Converged
☐ Non-Positive Definite
Initial Stress Stiffness: Full

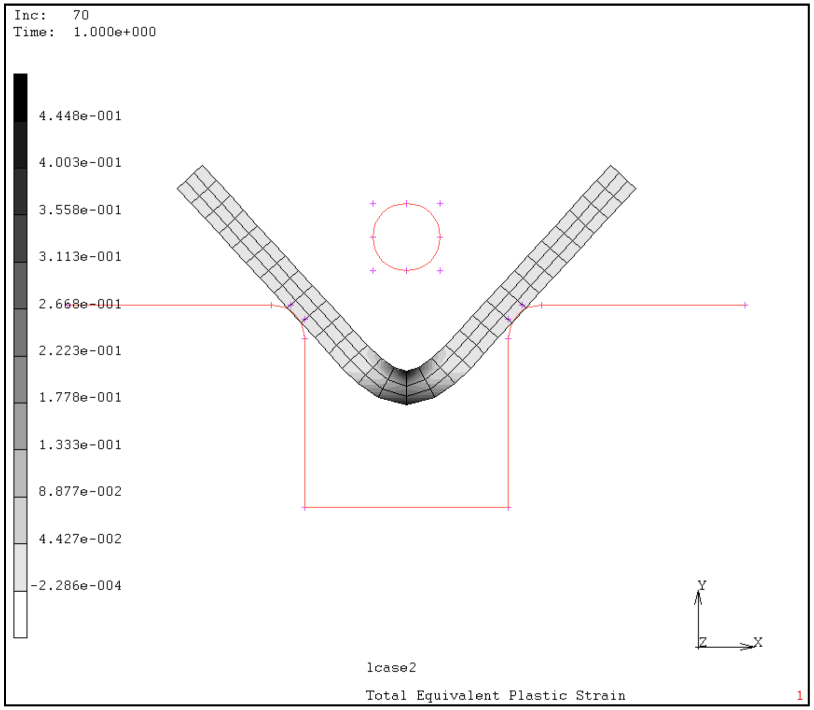
Iteration Parameters:
Iteration Method: Full Newton-Raphson
Max # of Iterations per Increment: 20
Min # of Iterations per Increment: 0
Desired # of Iterations per Increment: 5
Matrix Update Method: Automatic
Tolerance Method: Residual
Residual/Displacements: ☑ And ☐ Or
Error Type: Relative
☑ Automatic Switching

Residual Tolerances:
Relative Residual Force: 0.1
Relative Residual Moment: 0.
[Minimum Reaction Force]:
[Minimum Reaction Moment]:
Maximum Residual Force:
Maximum Residual Moment:

Auto Step + Relative Criteria + Auto Switch = Robust Solution

AUTOMATIC CRITERIA SWITCHING

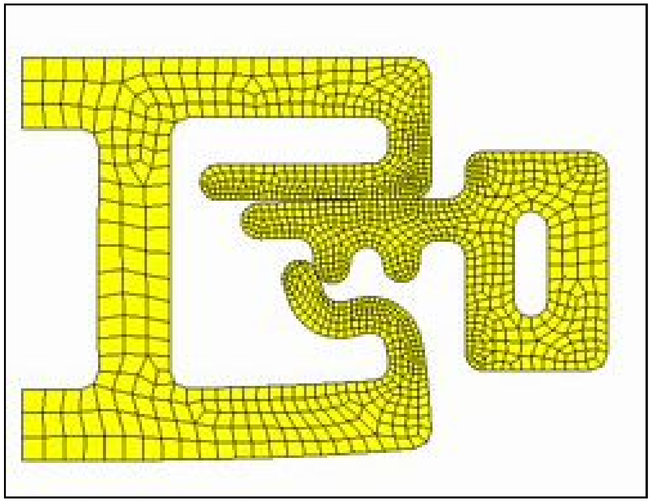
- **This example uses displacement convergence testing throughout the analysis of both draw bending and spring-back.**
- **At the last stages of the spring-back analysis, the convergence testing is automatically switched to strain energy testing due to extremely small displacement increment.**

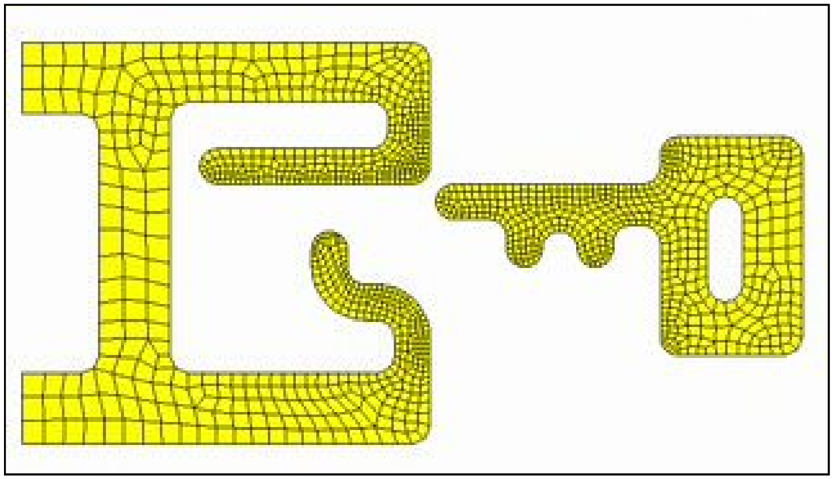


Draw bending and spring-back

AUTOMATIC CRITERIA SWITCHING

- This example uses residual force convergence testing throughout the analysis.
- However, at the beginning of the analysis the key has rigid body motion which results in very small reaction forces.
- Therefore the convergence testing is automatically switched to displacement testing and reset to residual testing as soon as the key comes in contact with the lock.





DIVERGENCE

- **Several conditions exist for which an iteration is said to diverge**
  - Convergence cannot be achieved within the maximum number of iterations MAXITER
  - The number of divergences (NDIV) reaches or exceeds the maximum MAXDIV
    - NDIV=NDIV + 2, when the error ratio E (see line search) exceeds 1 or falls below -10^{12}
    - NDIV=NDIV + 1, for $-10^{12} < E < -1$
  - An element related limit is exceeded during an iteration, i.e. FSTRESS
- **A combination of bisection and stiffness updates will be used to recover the solution**

ANALYSIS CONVERGENCE – GENERAL RECOMMENDATIONS

- **There is no single ‘right’ set of convergence criteria**
- **Program defaults attempt to make ‘intelligent’ selections, these should provide a good starting point. Use CTRLDEF settings discussed in future section**
- **Relative convergence testing is the default, in some cases it is not the best solution:**

Analysis Type	**Convergence Variable**		
	Displacement/ Rotation	**Residual Force/ Torque**	**Strain Energy**
Stress-free motion	Yes	No	No
Springback	No	Yes	No
Free Thermal Expansion	Yes	No	No
Constraint Thermal Expansion	No	Yes	Yes

Yes – relative tolerance testing works.
No – relative tolerance testing doesn’t work.

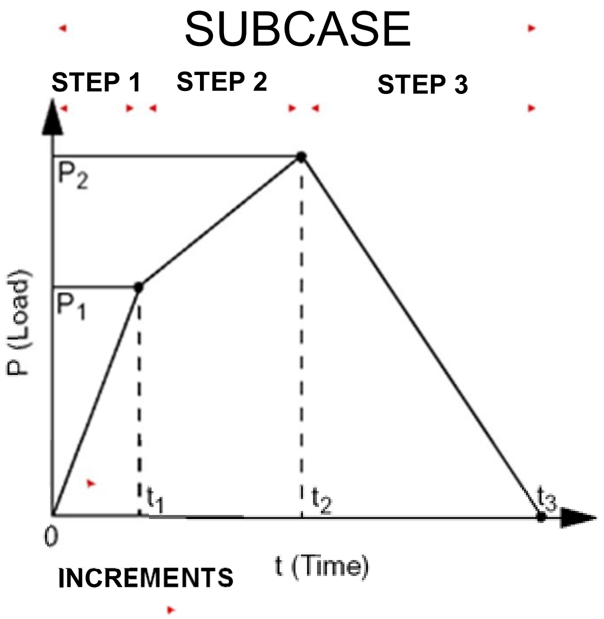
- **Auto-switching is, by default, not activated but can prevent inappropriate convergence requirements**

ADVANCING SCHEMES

ADVANCING SCHEMES

- **The iterations take place within increments, the increments belong to steps and the steps belong to subcases. Example:**

```
SOL 400
CEND
TITLE = ADVANCING SCHEMES
SUBCASE 10
   STEP 1
           LOAD = 1
           NLSTEP = 1
   STEP 2
           LOAD = 3
           NLSTEP = 1
   STEP 3
           LOAD = 3
           NLSTEP = 2
BEGIN BULK
NLSTEP  1
+       fixed   10      1
+       mech    pv              0.001           pfnt
```



ITERATIONS –subdivide INCREMENTS (not shown)

- **SUBCASE (Independent top level)**
  - **STEP (subdivide SUBCASE)**
    - **INCREMENT (subdivide STEP)**
      - **ITERATION (subdivide INCREMENT)**

ADVANCING SCHEMES

- **The SUBCASEs are independent**
  - This is different to SOL 106/129 where a subcase starts at the end of the previous subcase
  - to step back to the old SOLs 106/129 behavior one can use system cell 366 (STPFLG) = 1
- **A STEP starts at the end of the previous step or after the increment defined by NLIC**
- **Each step can be performed with another type of analysis, i.e. statics, transient, normal modes, frequency response, etc.**

LOAD INCREMENTATION CONTROL

- **There are three principal ways to apply loading in SOL400:**
  - The total load is divided equally into a number of specified solution increments.

    Constant load incrementation ("**Fixed**")

    $$\Delta F^{i} = \frac{F}{N_{inc}}$$

  - The next load increment is evaluated from the previous load multiplied by a growth/reduction factor.

    Variable load incrementation ("**adaptive**")

    $$\Delta F^{i} = \alpha \cdot \Delta F^{i-1}$$

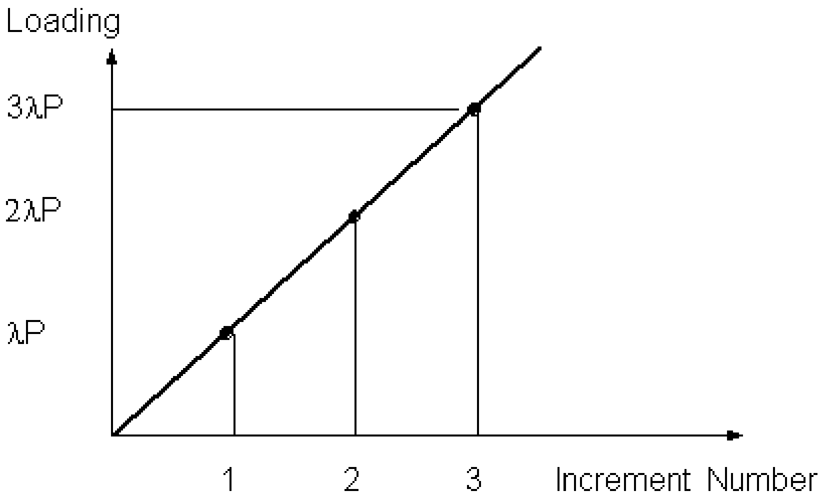
  - The next load increment is evaluated from the previous load increment multiplied by the arc-length magnitude.

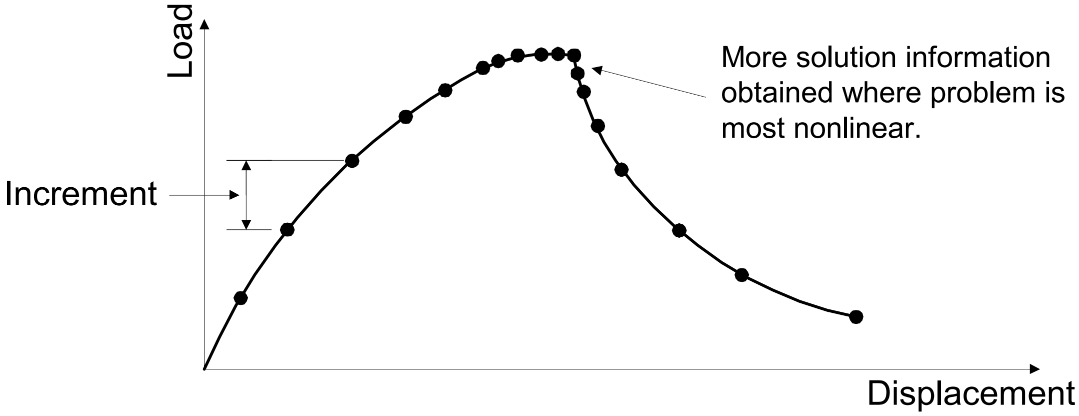
    Variable load incrementation ("adaptive/**arc- length**")

    $$\Delta F^{i} = \Delta\lambda \cdot F$$

LOAD INCREMENTATION CONTROL

- **Fixed**
  - Gives more user control.
  - Good where nonlinearity is mild or where the response is smooth.
- **Adaptive**
  - Recommended method
  - Automatically locates and handles sudden nonlinearity.

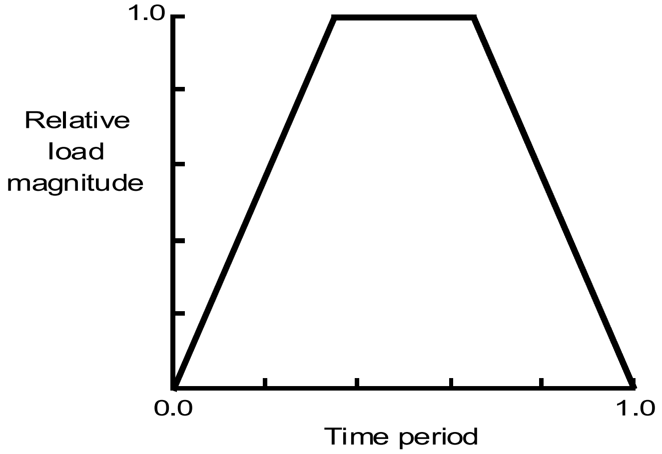


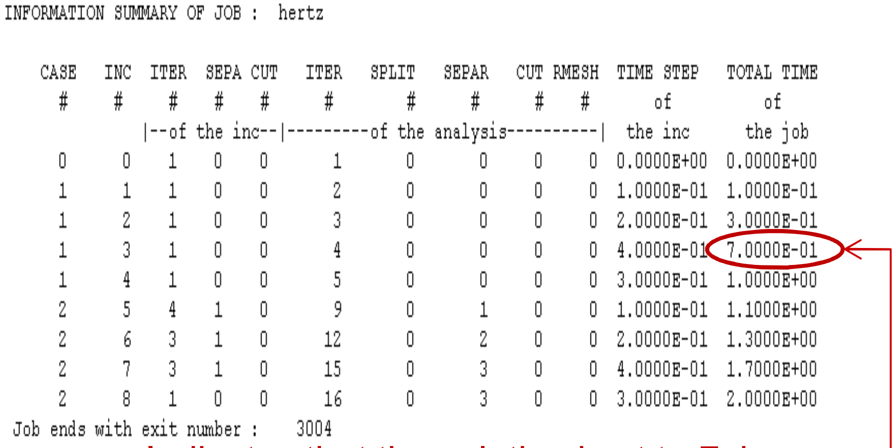


More solution information obtained where problem is most nonlinear.

THE TIME SCALE

- **SOL400 uses the concept of time to monitor load incrementation – even in static analyses (typically, a time period of unity for each STEP).**
- **It is invaluable in providing a scale over which a history of events is described when using load tables.**
- **For fixed increment load incrementation both the number of increments required and the total time of the solution are specified.**
- **For adaptive load incrementation, only time is needed.**
- **Easy to determine how far the analysis has progressed from the time output. The increment number does not give this information in an adaptive solution (50th/100 increments is not necessarily half way through).**



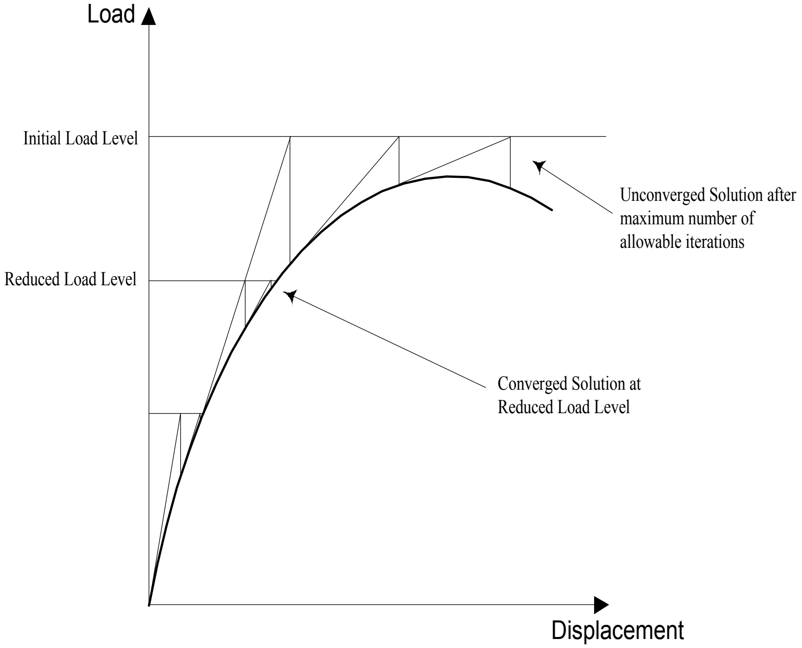


```
 INFORMATION SUMMARY OF JOB :  hertz


    CASE   INC  ITER  SEPA CUT   ITER    SPLIT    SEPAR    CUT RMESH TIME STEP   TOTAL TIME
      #     #     #     #    #     #         #        #      #     #      of          of
                 |--of the inc--|---------of the analysis----------|  the inc      the job
      0      0    1     0    0      1        0        0      0     0 0.0000E+00  0.0000E+00
      1      1    1     0    0      2        0        0      0     0 1.0000E-01  1.0000E-01
      1      2    1     0    0      3        0        0      0     0 2.0000E-01  3.0000E-01
      1      3    1     0    0      4        0        0      0     0 4.0000E-01  7.0000E-01
      1      4    1     0    0      5        0        0      0     0 3.0000E-01  1.0000E+00
      2      5    4     1    0      9        0        1      0     0 1.0000E-01  1.1000E+00
      2      6    3     1    0     12        0        2      0     0 2.0000E-01  1.3000E+00
      2      7    3     1    0     15        0        3      0     0 4.0000E-01  1.7000E+00
      2      8    1     0    0     16        0        3      0     0 3.0000E-01  2.0000E+00
 Job ends with exit number :    3004
```

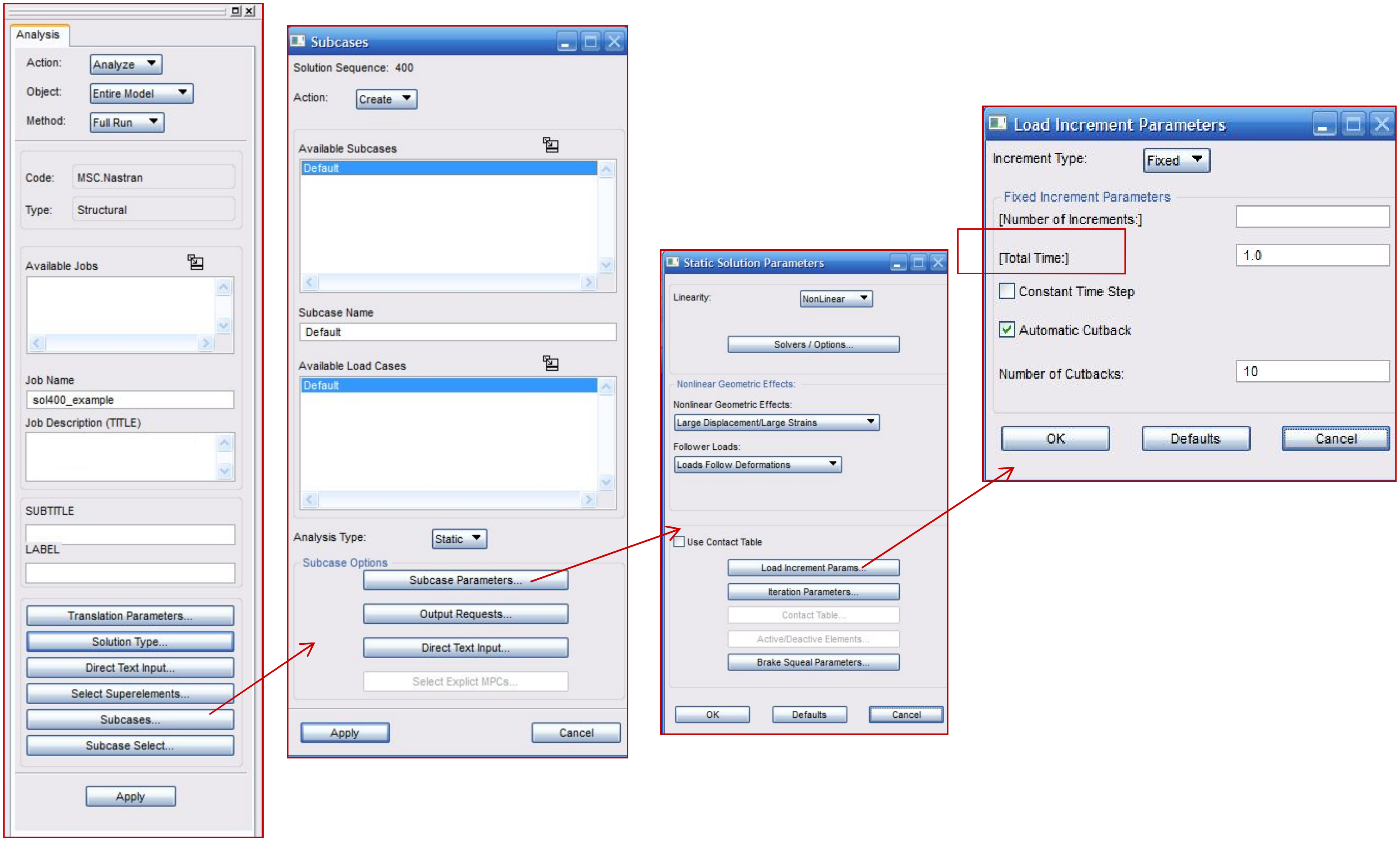
Indicates that the solution is at t=.7, in this example the first load case is 1.0 units long so it is .7/1.0 completed

LOAD INCREMENTATION CUT BACK (STEP REDUCTION)

- If convergence is not achievable at a specified load level, a step “cut back” is carried out.
- It saves having to restart the analysis at a specified load level.
- A cut back is where an increment is restarted – but at a reduced load level.
- The increment number is reset for the adaptive incrementation.
- Cut backs will continue until convergence is obtained or until the maximum number of reductions have reached.
- The maximum number of reductions may be specified by the user.
- Following convergence at a reduced level, the reduced step size may remain constant (fixed incrementation) or increase again (adaptive incrementation).



PATRAN LOAD INCREMENT PARAMETERS



Analysis

Action: Analyze

Object: Entire Model

Method: Full Run

Code: MSC.Nastran

Type: Structural

Available Jobs

Job Name: sol400_example

Job Description (TITLE)

SUBTITLE

LABEL

Subcases

Solution Sequence: 400

Action: Create

Available Subcases: Default

Subcase Name: Default

Available Load Cases: Default

Analysis Type: Static

Static Solution Parameters

Linearity: NonLinear

Nonlinear Geometric Effects: Large Displacement/Large Strains

Follower Loads: Loads Follow Deformations

☐ Use Contact Table

Load Increment Parameters

Increment Type: Fixed

Fixed Increment Parameters

[Number of Increments:]

[Total Time:] 1.0

☐ Constant Time Step

☑ Automatic Cutback

Number of Cutbacks: 10

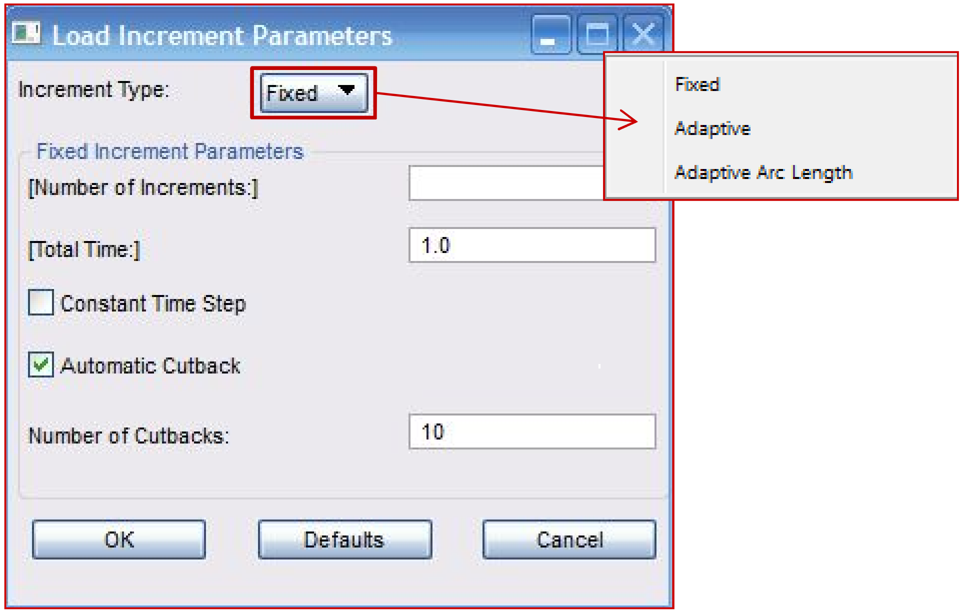
LOAD INCREMENT PARAMETERS

- **The form to control the load incrementation**
  - The data required depends upon the type of load stepping procedure specified:
    - Fixed

      or
    - Adaptive

      or
    - Adaptive Arc length

      or
    - Adaptive Temperature
  - It is possible to mix-and-match different incrementation procedures in an analysis, each subcase allows its own settings.



Increment Type: Fixed

Fixed Increment Parameters

[Number of Increments:]

[Total Time:] 1.0

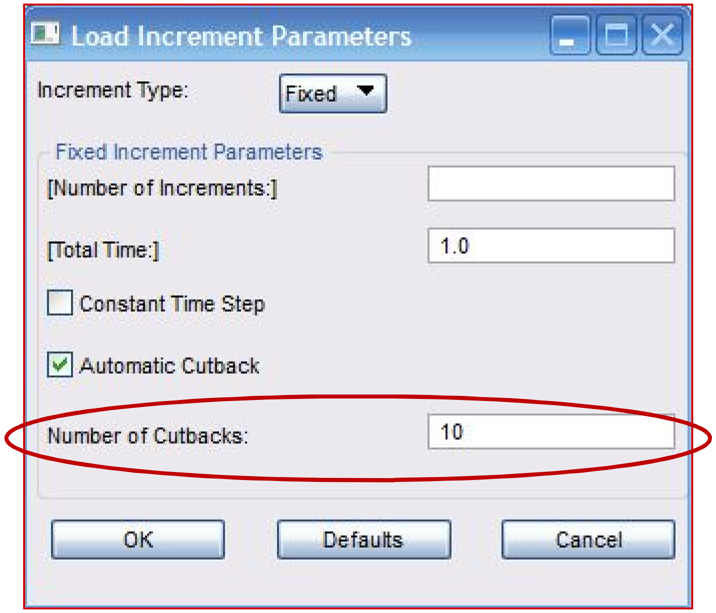
☐ Constant Time Step

☑ Automatic Cutback

Number of Cutbacks: 10

LOAD INCREMENT PARAMETERS: FIXED

- **Number of Cutbacks:**
  - Optionally invoked
  - The increment will stop if the maximum number of cutbacks is reached.
  - Upon detection of exit numbers 3002, 1009 or 2400, the increment is cut back.



Increment Type: Fixed

Fixed Increment Parameters

[Number of Increments:]

[Total Time:] 1.0

☐ Constant Time Step

☑ Automatic Cutback

Number of Cutbacks: 10

NLSTEP parameter:

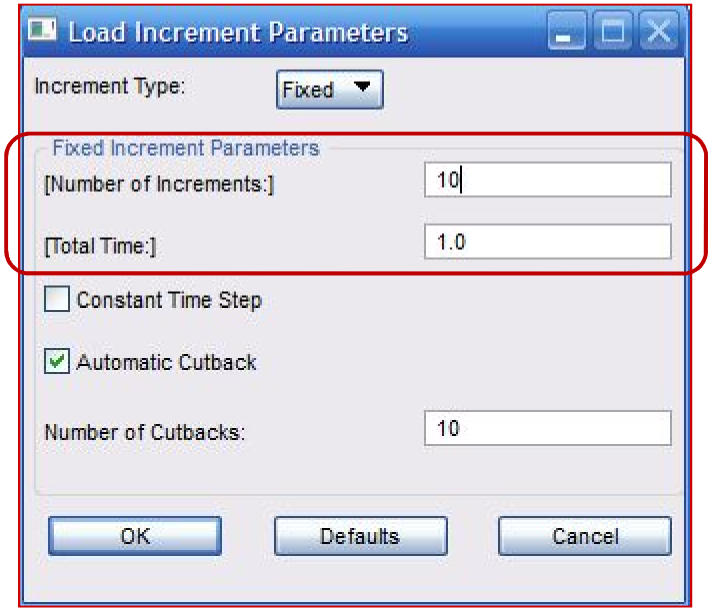
[NINC]

[TOTTIME]

[MAXBIS]

LOAD INCREMENT PARAMETERS: FIXED

- **Number of Increments**
  - User defined number of load increments, within which the total load will be applied in equal steps.
- **Total Time**
  - The user defined total time duration of this step.



Load Increment Parameters

Increment Type: Fixed

Fixed Increment Parameters

[Number of Increments:] 10

[Total Time:] 1.0

☐ Constant Time Step

☑ Automatic Cutback

Number of Cutbacks: 10

NLSTEP parameter:

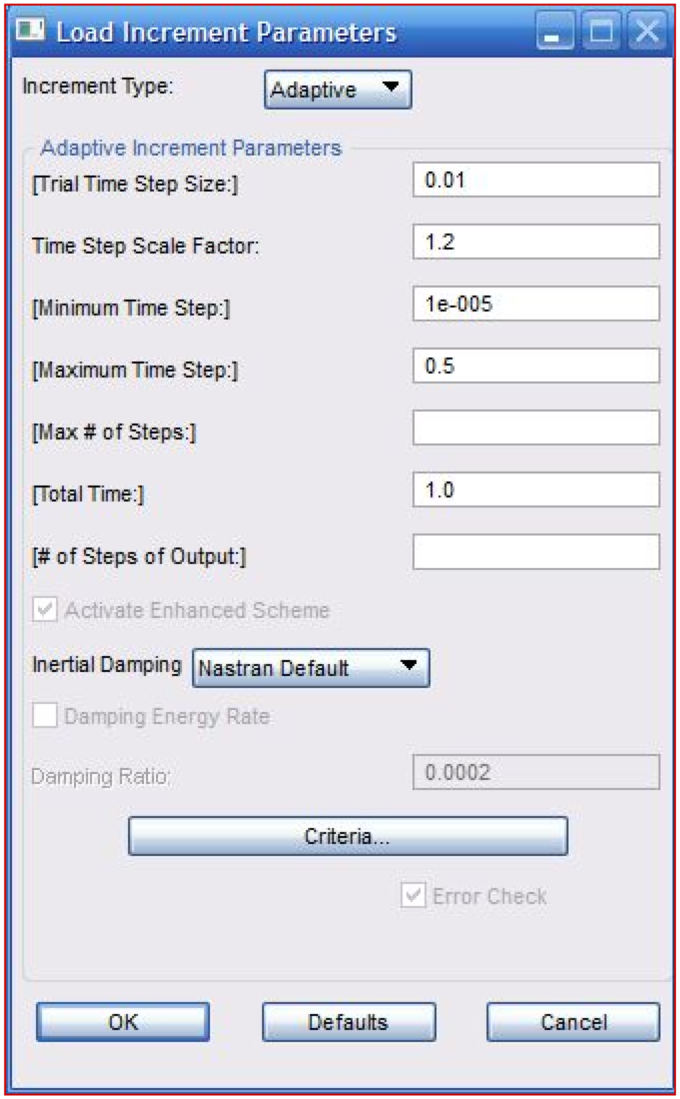
[NINC]

[TOTTIME]

[MAXBIS]

LOAD INCREMENT PARAMETERS: ADAPTIVE

- **Trial Time Step Size**
  - Patran : Initial fraction of loadcase time
  - User defined time increment
  - The starting value for the load stepping procedure.
  - For example, given a total solution time of unity, a starting step size of 0.01 would represent 1% of the total load to be applied.
  - Use default value
  - Discussed later



Increment Type: Adaptive

[Trial Time Step Size:] 0.01

Time Step Scale Factor: 1.2

[Minimum Time Step:] 1e-005

[Maximum Time Step:] 0.5

[Max # of Steps:]

[Total Time:] 1.0

[# of Steps of Output:]

☑ Activate Enhanced Scheme

Inertial Damping: Nastran Default

☐ Damping Energy Rate

Damping Ratio: 0.0002

☑ Error Check

NLSTEP parameter:

[DTINITF]

[SFACT]

[DTMINF]

[DTMAXF]

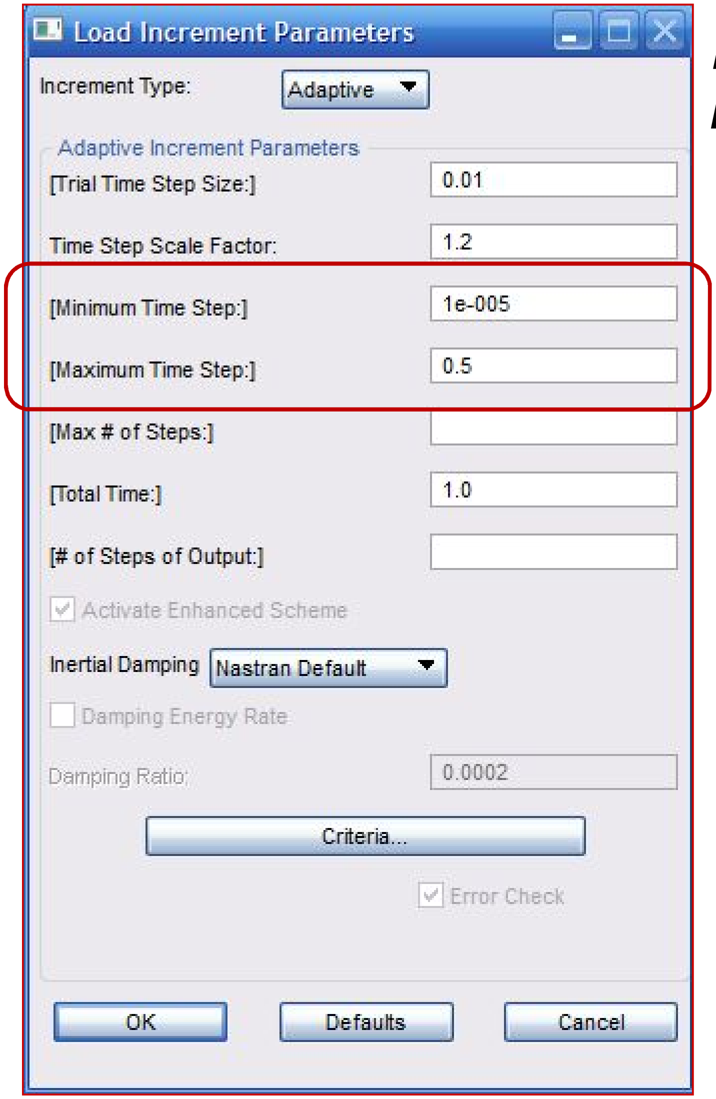
[NSMAX]

[TOTTIME]

[INTOUT]

LOAD INCREMENT PARAMETERS: ADAPTIVE

- **Minimum Time Step**
  - Patran: “[Minimum Time Step:]”
  - The solution algorithm is not allowed to use a time step smaller than this.
  - The analysis will not stop if it is reached.
  - The default value is the smaller of the current time increment or 1E-05 times the total time.
- **Maximum Time Step (optional)**
  - Patran: “[Maximum Time Step:]”
  - The algorithm is not allowed to use a value larger than this.
  - The analysis will not stop if it is reached.



Increment Type: Adaptive

[Trial Time Step Size:] 0.01

Time Step Scale Factor: 1.2

[Minimum Time Step:] 1e-005

[Maximum Time Step:] 0.5

[Max # of Steps:]

[Total Time:] 1.0

[# of Steps of Output:]

☑ Activate Enhanced Scheme

Inertial Damping: Nastran Default

☐ Damping Energy Rate

Damping Ratio: 0.0002

☑ Error Check

NLSTEP parameter:

[DTINITF]

[SFACT]

[DTMINF]

[DTMAXF]

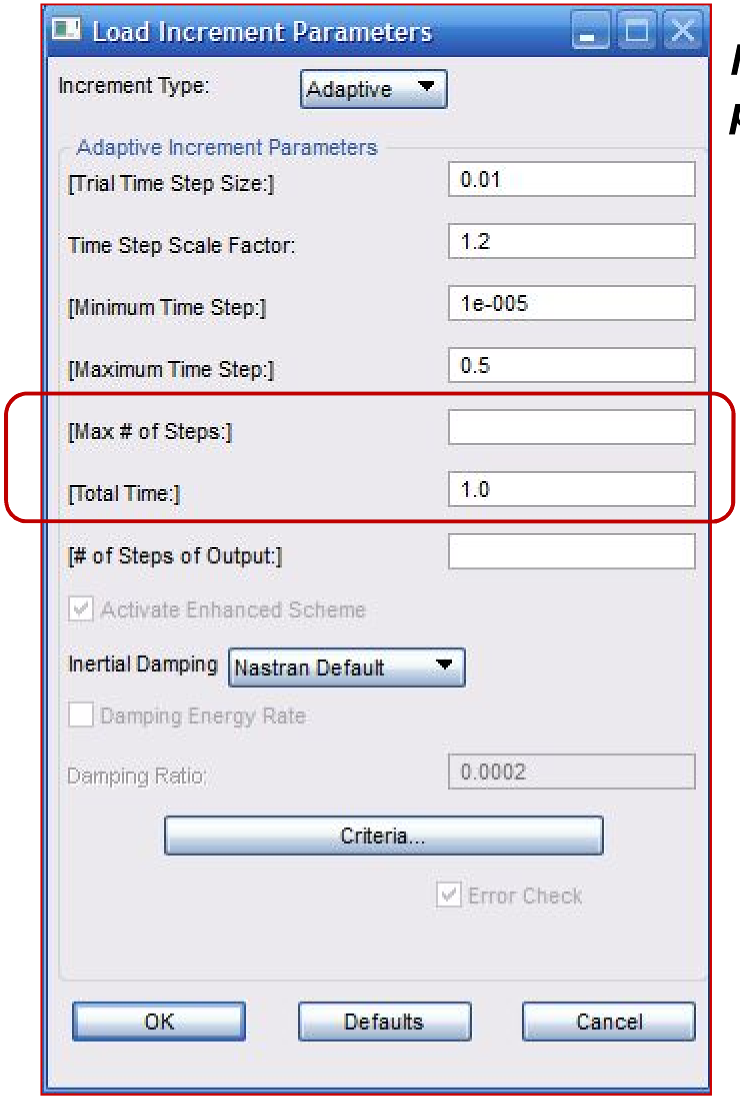
[NSMAX]

[TOTTIME]

[INTOUT]

LOAD INCREMENT PARAMETERS: ADAPTIVE

- **Total Time**
  - Patran: “Total Time”
  - The user defined total time duration of this loadstep.
- **Max # of Steps**
  - Patran: “Maximum # Steps”
  - The maximum number of increments in the current loadstep.
  - The analysis will stop if it is reached.



Increment Type: Adaptive

[Trial Time Step Size:] 0.01

Time Step Scale Factor: 1.2

[Minimum Time Step:] 1e-005

[Maximum Time Step:] 0.5

[Max # of Steps:]

[Total Time:] 1.0

[# of Steps of Output:]

☑ Activate Enhanced Scheme

Inertial Damping: Nastran Default

☐ Damping Energy Rate

Damping Ratio: 0.0002

☑ Error Check

NLSTEP parameter:

[DTINITF]

[SFACT]

[DTMINF]

[DTMAXF]

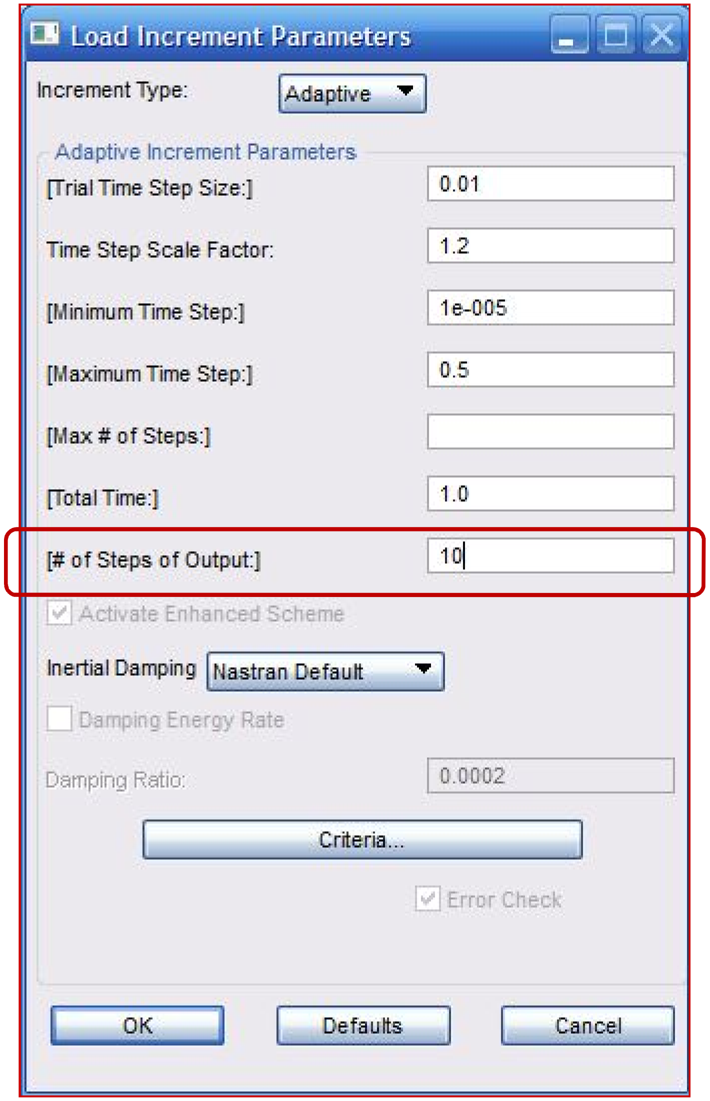
[NSMAX]

[TOTTIME]

[INTOUT]

LOAD INCREMENT PARAMETERS: ADAPTIVE

- **# of Steps of Output**
  - Patran: “# of Steps of Output”
  - By default MSC Nastran will output for all increments to results file
  - For highly nonlinear analysis, when using adaptive increment time stepping very large numbers of increments can be required resulting in very large results files.
  - In those cases limiting the number of output steps is important to prevent very large result file.
  - In the example shown MSC Nastran will output 10 evenly spaced sets of results during the increment duration of 1.0, thus results at, or near, solutions at .1,.2,.3,…



Increment Type: Adaptive

Adaptive Increment Parameters

[Trial Time Step Size:] 0.01

Time Step Scale Factor: 1.2

[Minimum Time Step:] 1e-005

[Maximum Time Step:] 0.5

[Max # of Steps:]

[Total Time:] 1.0

[# of Steps of Output:] 10

☑ Activate Enhanced Scheme

Inertial Damping Nastran Default

☐ Damping Energy Rate

Damping Ratio: 0.0002

☑ Error Check

NLSTEP parameter:

[DTINITF]

[SFACT]

[DTMINF]

[DTMAXF]

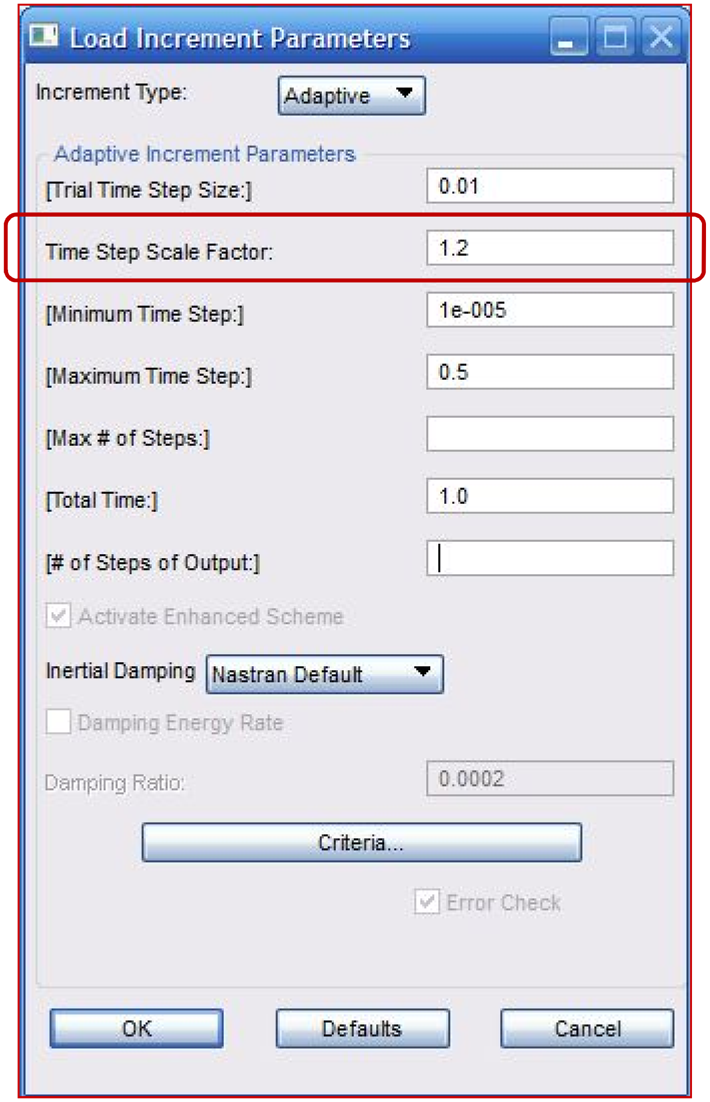
[NSMAX]

[TOTTIME]

[INTOUT]

LOAD INCREMENT PARAMETERS: ADAPTIVE

- **Time Step Scale Factor**
  - Factor used to increase/decrease the time step is user-defined (default is 1.2).
  - No increase of the time step during the current increment.
  - Scale factor used is bounded by the user defined minimum and maximum factors.



Increment Type: Adaptive

[Trial Time Step Size:] 0.01

Time Step Scale Factor: 1.2

[Minimum Time Step:] 1e-005

[Maximum Time Step:] 0.5

[Max # of Steps:]

[Total Time:] 1.0

[# of Steps of Output:]

☑ Activate Enhanced Scheme

Inertial Damping Nastran Default

☐ Damping Energy Rate

Damping Ratio: 0.0002

☑ Error Check

NLSTEP parameter:

[DTINITF]

[SFACT]

[DTMINF]

[DTMAXF]

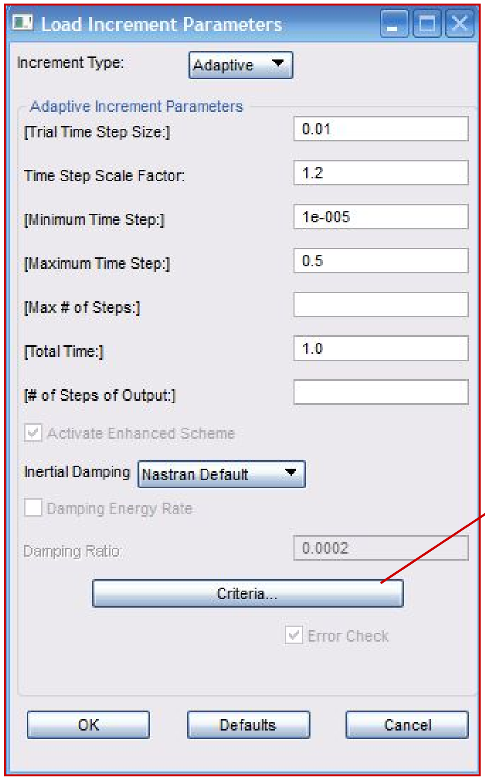
[NSMAX]

[TOTTIME]

[INTOUT]

“PHYSICAL” CRITERIA FOR LOAD INCREMENTATION

- **User defined range of values for one or more of the available criteria.**
- **Specify a maximum value permissible within each range.**
- **Can be used in conjunction with “standard” adaptive incrementation control.**



Load Increment Parameters

Increment Type: Adaptive

Adaptive Increment Parameters

[Trial Time Step Size:] 0.01

Time Step Scale Factor: 1.2

[Minimum Time Step:] 1e-005

[Maximum Time Step:] 0.5

[Max # of Steps:]

[Total Time:] 1.0

[# of Steps of Output:]

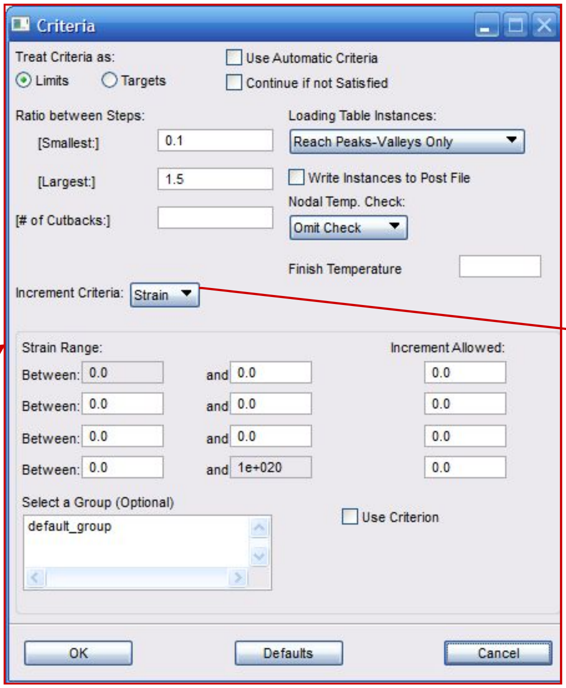
☑ Activate Enhanced Scheme

Inertial Damping Nastran Default

☐ Damping Energy Rate

Damping Ratio 0.0002

☑ Error Check



Criteria

Treat Criteria as: ● Limits ○ Targets

☐ Use Automatic Criteria

☐ Continue if not Satisfied

Ratio between Steps:

[Smallest:] 0.1

[Largest:] 1.5

[# of Cutbacks:]

Loading Table Instances: Reach Peaks-Valleys Only

☐ Write Instances to Post File

Nodal Temp. Check: Omit Check

Finish Temperature

Increment Criteria: Strain

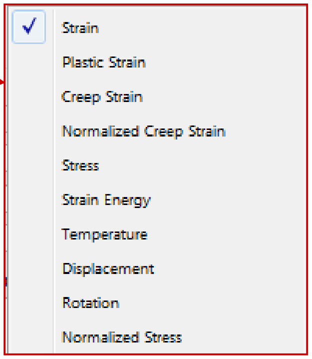
Strain Range:

				Increment Allowed:
Between:	0.0	and	0.0	0.0
Between:	0.0	and	0.0	0.0
Between:	0.0	and	0.0	0.0
Between:	0.0	and	1e+020	0.0

Select a Group (Optional)

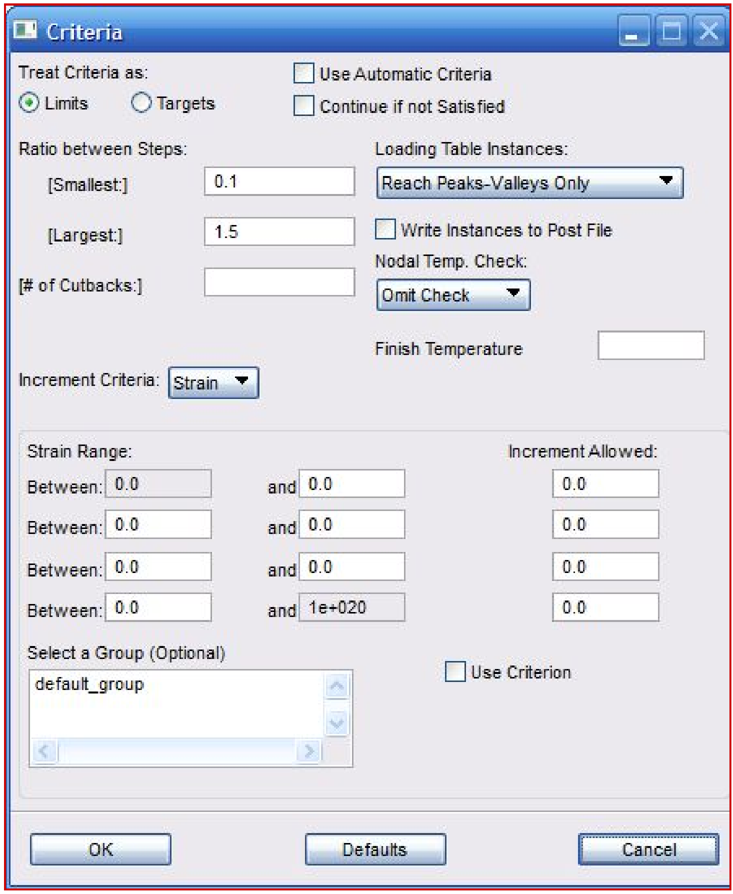
default_group

☐ Use Criterion



“PHYSICAL” CRITERIA FOR LOAD INCREMENTATION

- **Limits (default)**
  - Decrease current time step if a physical criterion is violated.
  - “Desired-Actual” control is still used to determine the next increment step size.
- **Targets**
  - Decrease current time if a physical criterion is violated.
  - “Physical” criterion is used to determine the next increment step size (not the “desired-actual” control).
  - If the calculated values of the criteria are higher (lower) than the user-defined values in any iteration, the time step is scaled down (up) and the current increment is repeated.
  - The scale factor used for reduction (increase) is the ratio between the actual value and the target value.
  - This factor is limited by user-specified minimum and maximum factors (defaults to 0.1 and 10 respectively).



Treat Criteria as: ● Limits ○ Targets

☐ Use Automatic Criteria

☐ Continue if not Satisfied

Ratio between Steps:

[Smallest:] 0.1

[Largest:] 1.5

[# of Cutbacks:]

Loading Table Instances: Reach Peaks-Valleys Only

☐ Write Instances to Post File

Nodal Temp. Check: Omit Check

Finish Temperature

Increment Criteria: Strain

Strain Range:				Increment Allowed:
Between:	0.0	and	0.0	0.0
Between:	0.0	and	0.0	0.0
Between:	0.0	and	0.0	0.0
Between:	0.0	and	1e+020	0.0

Select a Group (Optional)

default_group

☐ Use Criterion

NLSTEP parameter:

[CRITID/LIMTAR /IPHYS]

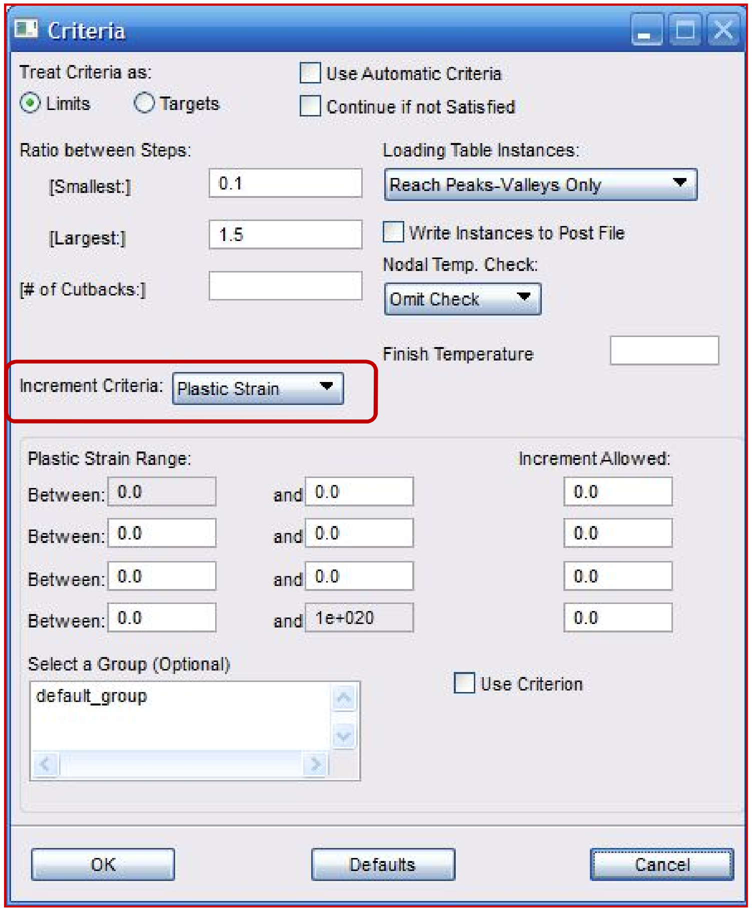
[RSMALL]

[RBIG]

TABSCTL defined

“PHYSICAL” CRITERIA FOR LOAD INCREMENTATION

- **Criteria Range**
  - A number of physical criteria can be selected (more than one if required).
  - Consider the Plastic Strain criterion.
  - The range of plastic strains together with the maximum permitted strain increment are required.
  - If a plastic strain increment is detected to be beyond that permitted, a step reduction will occur to ensure that the strain remains within the specified bounds.



Treat Criteria as: ● Limits ○ Targets

☐ Use Automatic Criteria

☐ Continue if not Satisfied

Ratio between Steps:

[Smallest:] 0.1

[Largest:] 1.5

[# of Cutbacks:]

Loading Table Instances: Reach Peaks-Valleys Only

☐ Write Instances to Post File

Nodal Temp. Check: Omit Check

Finish Temperature

Increment Criteria: Plastic Strain

Plastic Strain Range:				Increment Allowed:
Between:	0.0	and	0.0	0.0
Between:	0.0	and	0.0	0.0
Between:	0.0	and	0.0	0.0
Between:	0.0	and	1e+020	0.0

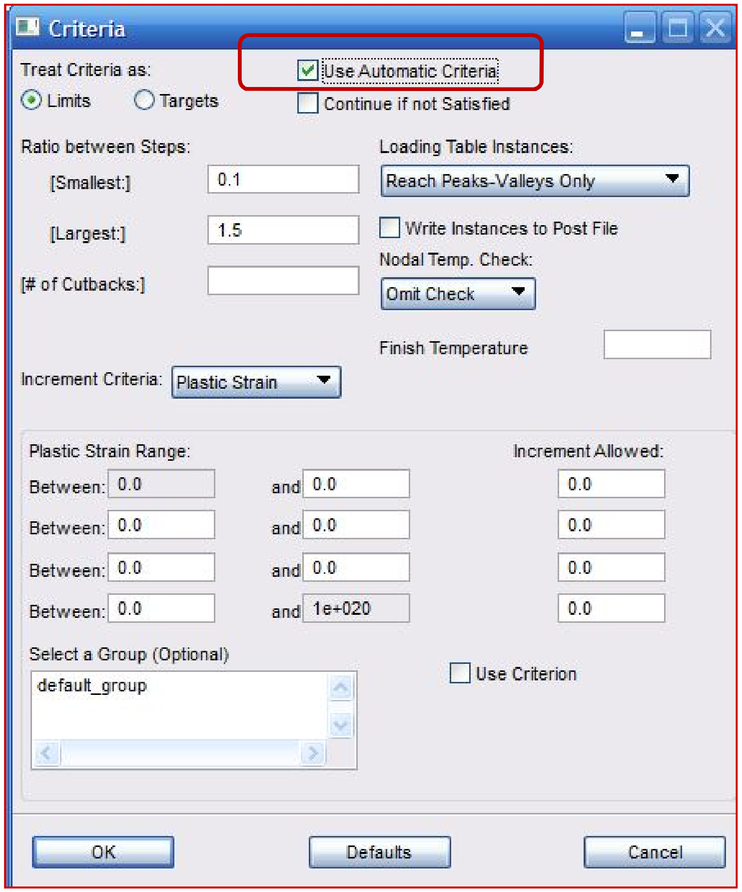
Select a Group (Optional)

default_group

☐ Use Criterion

“AUTOMATIC” CRITERIA FOR LOAD INCREMENTATION

- Additional option allows for automatic physical criteria to be used.
- These automatic criteria serve as upper-bound limits to prevent run-away Newton-Raphson iterations.
- Criteria are only added in the analysis if there are no competing explicitly defined user-criteria found.
- Criteria are only used as limits; they are used to control the time step for the current increment but not for the next increment.



Treat Criteria as: ● Limits ○ Targets

☑ Use Automatic Criteria

☐ Continue if not Satisfied

Ratio between Steps:

[Smallest:] 0.1

[Largest:] 1.5

[# of Cutbacks:]

Loading Table Instances: Reach Peaks-Valleys Only

☐ Write Instances to Post File

Nodal Temp. Check: Omit Check

Finish Temperature

Increment Criteria: Plastic Strain

Plastic Strain Range:			Increment Allowed:
Between: 0.0	and	0.0	0.0
Between: 0.0	and	0.0	0.0
Between: 0.0	and	0.0	0.0
Between: 0.0	and	1e+020	0.0

Select a Group (Optional)

default_group

☐ Use Criterion

NLSTEP parameter:

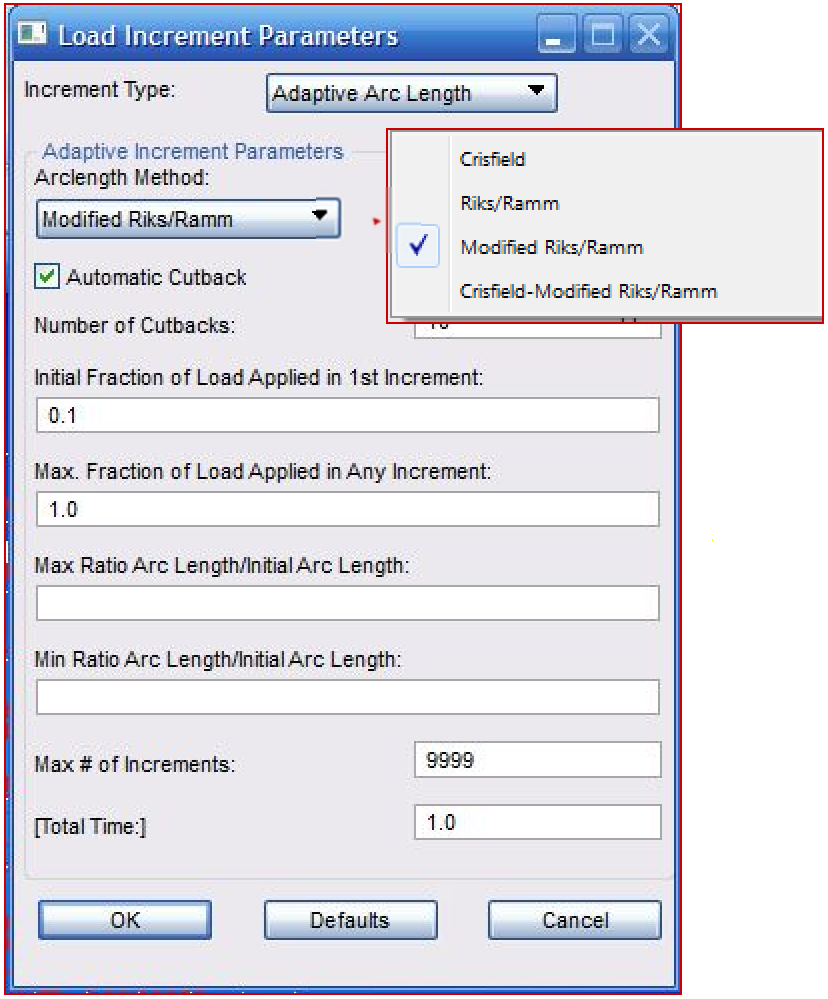
[IPHYS]

ADDITIONAL AUTOMATIC CRITERIA FOR LOAD INCREMENTATION

- **Five additional criteria**
  1. **Strain** criterion for large displacement analyses – maximum equivalent total strain increment set to 50%.
  2. **Plastic Strain** criterion for large displacement, finite strain analyses - maximum equivalent plastic strain increment set to 10%.
  3. **Creep Strain** criterion for explicit creep analyses - maximum creep strain change/elastic strain set to 0.5.
  4. **Stress change** criterion for explicit creep analyses – maximum equivalent stress change/equivalent stress set to 0.5.
  5. **State variable** criterion for large displacement analyses - maximum temperature increment is such that the equivalent stress increment associated with the change in thermal properties of the materials does not exceed 50% of the total equivalent stress.

LOAD INCREMENT PARAMETERS: ADAPTIVE/ARC LENGTH

- **Max Ratio Arc Length/Initial Arc Length:**
  - Keeps the arc length step size from growing too large.
- **Min Ratio Arc Length/Initial Arc Length:**
  - Keeps the arc length step size from reducing too much.



Increment Type: Adaptive Arc Length

Adaptive Increment Parameters

Arclength Method: Modified Riks/Ramm

☑ Automatic Cutback

Number of Cutbacks:

Initial Fraction of Load Applied in 1st Increment: 0.1

Max. Fraction of Load Applied in Any Increment: 1.0

Max Ratio Arc Length/Initial Arc Length:

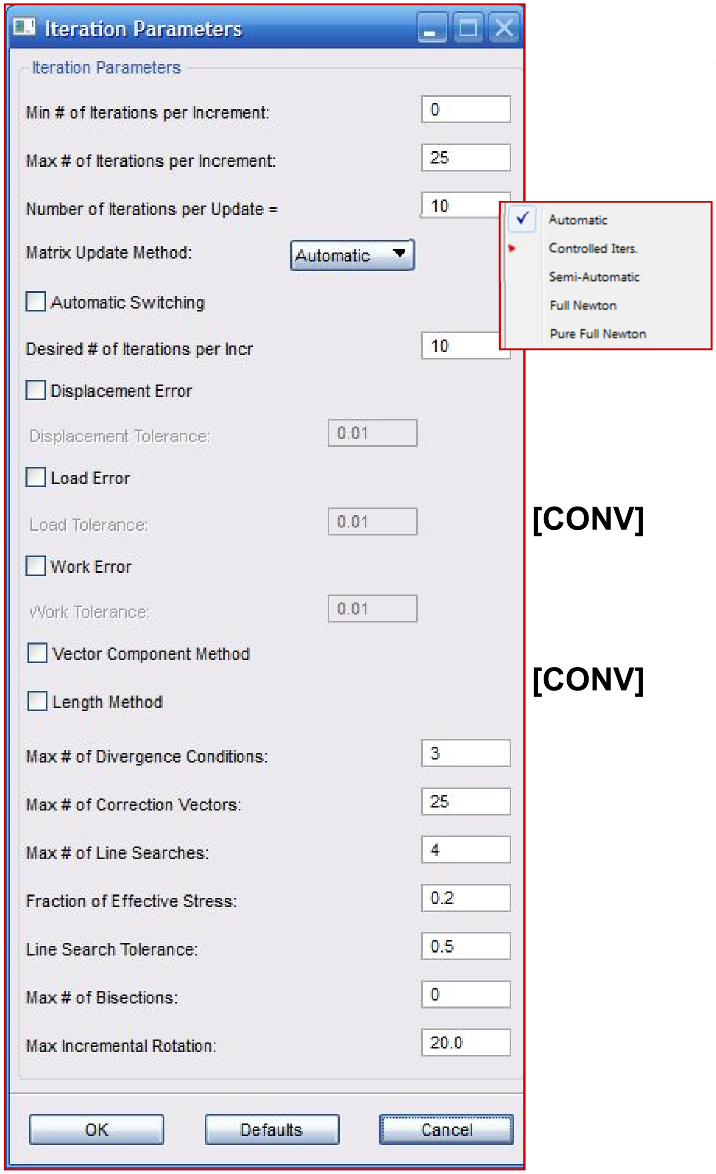
Min Ratio Arc Length/Initial Arc Length:

Max # of Increments: 9999

[Total Time:] 1.0

ITERATION PARAMETERS

- **Controls Solution algorithm**
- **Good place to start is with defaults**
- **“Matrix Update Method”**
  - ‘Automatic’ – program selects the most efficient strategy based on convergence rates
  - ‘Semi-Automatic’ – for each load increment the program performs a single iteration, updates the stiffness matrix and resumes normal Automatic option
  - ‘Controlled Iters’ – user specified frequency of stiffness matrix update
    - Modified Newton Raphson accessed by selecting ‘Controlled Iters’ and setting ‘Max # of Iterations per Increment’ = ‘Number of Iterations per Update’
  - ‘Pure Full Newton’ – same as ‘Full Newton’ with EPSU=-.01,EPSW=-.01, and MAXLS=0 (FNR is EPSU=.1,EPSW=.1, MAXLS=4)



Iteration Parameters

Min # of Iterations per Increment: 0

Max # of Iterations per Increment: 25

Number of Iterations per Update = 10

Matrix Update Method: Automatic

☐ Automatic Switching

Desired # of Iterations per Incr 10

☐ Displacement Error

Displacement Tolerance: 0.01

☐ Load Error

Load Tolerance: 0.01

☐ Work Error

Work Tolerance: 0.01

☐ Vector Component Method

☐ Length Method

Max # of Divergence Conditions: 3

Max # of Correction Vectors: 25

Max # of Line Searches: 4

Fraction of Effective Stress: 0.2

Line Search Tolerance: 0.5

Max # of Bisections: 0

Max Incremental Rotation: 20.0

[CONV]

[CONV]

NLSTEP parameter:

[MINITER]
[MAXITER]
[KSTEP]

[KMETHOD]
[CONV]

[EPSU]

[EPSP]

[EPSW]

[MAXDIV]
[MAXQN]
[MAXLS]
[FSTRESS]
[LSTOL]
[MAXBIS]

NLSTEP BULK DATA ENTRY

- **‘One stop shop’ MSC Nastran input file entry that defines:**
  - Load incrementation parameters
    - FIXED, ADAPTIVE, ALM
  - Iteration parameters
    - Full Newton Raphson, Pure Full Newton Raphson, Modified Newton Raphson,…)
  - Convergence tolerances
    - Type and value
  - Output Frequency
    - Allow reduced data output for highly nonlinear solutions
  - Can be used in Statics and Dynamics
    - Old NLPARM/TSTEPNL method is still available
  - Applies to all analysis disciplines
    - Thermal, Mechanical, Coupled, R-C circuit
    - Only Structural/Mechanical discussed in this course

NLSTEP BULK DATA ENTRY

- **Format: (For SOL 400)**

1	2	3	4	5	6	7	8	9	10
NLSTEP	ID	TOTTIME	CTRLDEF						
	“GENERAL”	MAXITER	MINITER	MAXBIS	CREEP				
	“FIXED”	NINC	NO						
	“ADAPT”	DTINITF	DTMINF	DTMAXF	NDESTR	SFACT	INTOUT	NSMAX	
		IDAMP	DAMP	CRITTID	IPHYS	LIMTAR	RSMALL	RBIG	
		ADJUST	MSTEP	RB	UTOL				
	“ARCLN”	TYPE	DTINITFA	MINALR	MAXALR	SCALEA	NDESIRA	NSMAXA	
	“HEAT”	CONVH	EPSUH	EPSPH	EPSWH	KMETHODH	KSTEPH		
		MAXQNH	MAXLSH	LSTOLH					
	“MECH”	CONV	EPSU	EPSP	EPSW	KMETHOD	KSTEP	MRCONV	
		MAXQN	MAXLS	LSTOL	FSTRESS				
	“COUP”	HGENPLAS	HGENFRIC						
	“RCHEAT”	SOLVER	DRLXCA	ARLXCA	BALENG	DAMPC	GRVCON	CSGFAC	
		NRLOOP	OUTINV	DTIME1					

KEYWORD CTRLDEF – “SMART DEFAULTS”

1	2	3	4	5	6	7	8	9	10
NLSTEP	ID	TOTTIME	**CTRLDEF**						
	GENERAL	MAXITER	MINITER	MAXBIS	CREEP				

- Automatically sets up the entries for the time/load stepping adjustment and convergence tolerance
- Four values available as "QLINEAR", "MILDLY", "SEVERELY" , or blank in SOL400
- Select a value based on judgment of the nonlinearity of the model
- Requires hand editing of the MSC Nastran input file. Can not be created in Patran

KEYWORD CTRLDEF

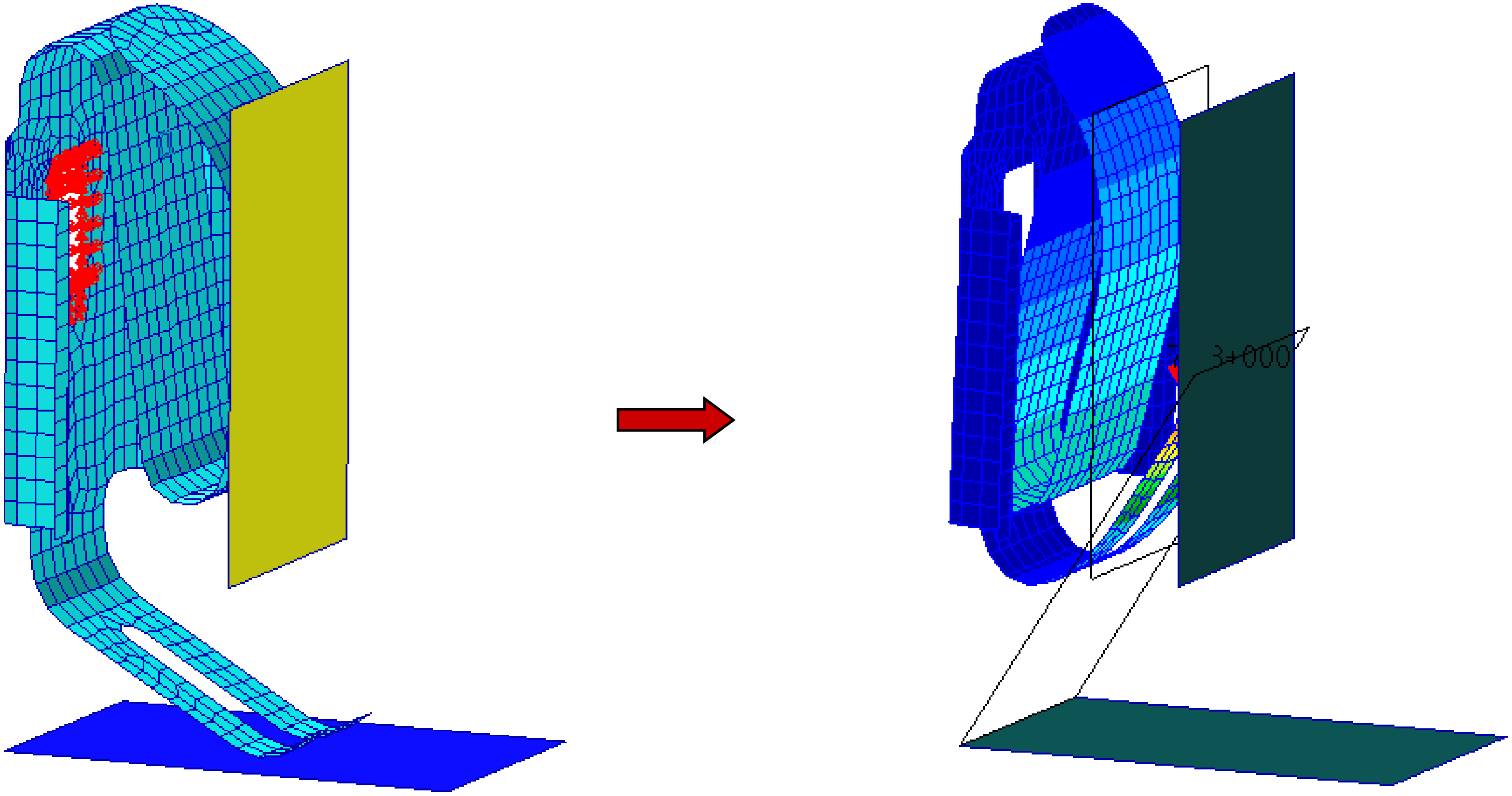
- **CTRLDEF = ‘QLINEAR’ -** For linear analysis but with contact nonlinearity
  - NINC = 1 (FIXED) or DTINITF = 1.0 (ADAPT) ------ 1 increment load stepping
  - CONV = PV, EPSP = 0.001 and KMETHOD = PFNT
  - Good for contact, interference fit and bolt preload
- **CTRLDEF = ‘MILDLY’ -** For mildly nonlinear problems
  - NINC = 10 (FIXED) or DTINITF = 0.1 (ADAPT) – Default is FIXED
  - CONV = PV, EPSP = 0.01 and KMETHOD = PFNT
  - Good for general nonlinear problem including geometry nonlinearity, material nonlinearity, contact nonlinearity, and their combination
- **CTRLDEF = ‘SEVERELY’ -** For severely nonlinear problems
  - NINC = 50 (FIXED) or DTINITF = 0.01 (ADAPT) – Default is FIXED
  - CONV = PV, EPSP = 0.01 and KMETHOD = PFNT
  - Good for nonlinear problem involving severe large displacement, large strain, multi-contact pairs

GENERAL RECOMMENDATIONS

- **NLSTEP is a unified load/time stepping control option and should be used for all nonlinear analysis.**
- **Keyword CTRLDEF is a user-friendly entry, users have only to select a value based on their judgment of the nonlinearity of the model to be analyzed.**
- **Adaptive load stepping is usually more efficient.**
- **For the advanced nonlinear elements and materials and contact analysis, FNT or PFNT with CONV=PV should be used.**
- **For contact with friction, PFNT and UPW has been found to be more precise.**

DEMO

- **Clip model with 3 contact bodies**
- **Touching contact and large deformation**



DEMO RUNTIME

- **NLPARM with FIXED: 900s**
- **NLSTEP with ADAPT: 163s**
- **NLSTEP with MILDLY: 116s**

NONLINEAR OUTPUT CONTROL

NONLINEAR OUTPUT CONTROL

- **Nonlinear analyses require additional solution efforts and can be challenging to achieved ‘converged’ status**
- **Additional information can be helpful in trouble shooting**
- **Sources of additional information:**
  - “.f06” file – Standard MSC Nastran output file
    - SOL 400 automatically writes increment/iteration information
    - Additional information can be requested using the NLOPRM entry if desired.
    - If higher level of diagnostics is desired consider ‘DIAG 51’ entry at ECS level
    - Buffered output so not always up to date of a currently running job
  - “.sts” file - status file will be written for each SOL 400 job
    - Updated in real time so a running job can be monitored

NONLINEAR OUTPUT CONTROL

- **Example's .sts file:**

```
information summary of job:  ./demo_400_nlio_std

  subcase      inc  cycl  sepa cut     cycl   split   separ   cut  rmesh time step   total time  wall time
   cpu time    max resp. type
  /step #       #     #    #    #        #        #      #      #     #     of          of
        1          |--of the inc--|-----------of the analysis-----------|  the inc      the job
        1        0     0    0    0        0       0       0     0     0  0.0000E+00  0.0000E+00       2.00
   1.40   0.0000E+00 disp
        1        1     4    0    0        4       0       0     0     0  1.0000E+00  1.0000E+00       2.00
   1.44  -1.5486E+02 disp
 Job ends with exit number :       0
            total wall time:          2.00
            total cpu  time:          1.47

            exit DEFINITION -----
            = 0  job terminates normally
            = 1  job terminates abnormally (check Fatal Error Message in F06)
```

NONLINEAR OUTPUT CONTROL

```
N O N - L I N E A R   I T E R A T I O N   S O L U T I O N   C O N T R O L   P A R A M E T E R S

     LOOP CONTROLS FOR :   SUBCASE        1,         STEP         1,         SUBSTEP        0


     SOLUTION CONTROL PARAMETERS FROM : NLSTEP                         ID :           1

          Total Time of Loading Case (TOTTIME) ....     1.00E+00

          Maximum Number of Iterations (MAXITER) ..           10
          Minimum Number of Iterations (MINITER) ..            1
          Maximum Number of Bisection (MAXBIS) ....           10
          Creep Option (CREEP) ...................             0

          Number of Fixed Increments (NINC) .......            1
          Interval of Output (NO) .................            1

          Convergence Criteria (CONV) .............     P  V
                   - Displacement    (EPSU) ......    -1.00E-01
          Tolerance  - Residual Force (EPSP) ......     1.00E-03
                   - Work            (EPSW) ......     1.00E-01
          Option of Rotations and Moments (MRCONV).            3
          Matrix Update Option (KMETHOD) .........          PFNT
          Matrix Update Increment (KSTEP) ........             1
          Maximum Quasi-Newton Vectors (MAXQN) ....            0
          Maximum Line Searches (MAXLS) ..........             0
          Line Search Tolerance (LSTOL) ..........      5.00E-01
          Error Tolerance in YF (FSTRESS) ........      2.00E-01

 *** USER INFORMATION MESSAGE 6204 (NL3EMA)
     0.000000E+00 SECONDS REQUIRED TO DECOMPOSE MATRIX.
%1.00000E+00   1  1 1.00E+00 2.69E+01 1.00E+00  1.000  1   1   0  4.92E+04 6.533E+06 1.06E+01 -1.587E+02        6 2  1.00 0  0   0    1
%1.00000E+00   1  2 9.33E-02 1.62E+00 8.19E+02  0.060  0   1   0  2.41E+03 3.233E+06 1.13E+01 -1.558E+02        6 2  1.00 0  0   1    2
%1.00000E+00   1  3 5.83E-03 1.66E-01 2.07E-02  0.063  0   1   0  1.60E+00 3.233E+06 1.12E+01 -1.549E+02        6 2  1.00 0  0   2    3
%1.00000E+00   1  4 3.07E-06 7.29E-07 8.63E-03  0.093  0   1   0  2.71E-05 3.233E+06 1.12E+01 -1.549E+02        6 2  1.00 0  0   3    4

 *** JOB CONVERGES FOR THE CURRENT STEP.
 *** SUBCASE      1 STEP      1 IS COMPLETED.
```

Standard .f06 Output

NONLINEAR OUTPUT CONTROL

```
        LOAD  NO.       - - ERROR FACTORS - -    CONV  ITR MAT NO.    AVG     TOTL       - - - - - DISP - - - - - -  LINE_S  NO. TOT TOT
        STEP  INC ITR  DISP     LOAD      WORK   RATE  DIV DIV BIS  R_FORCE   WORK      AVG        MAX     AT GRID C FACT NO QNV KUD ITR

%1.00000E+00   1  1 1.00E+00 2.69E+01 1.00E+00  1.000  1   1   0  4.92E+04 6.533E+06 1.06E+01 -1.587E+02        6 2  1.00 0  0   0    1
```

- **LOAD STEP**
  - step number minus 1 plus fraction of step, i.e. 0.08 = 8% of first step
- **NO. INC**
  - increment number
- **ITR**
  - iteration number
- **DISP LOAD WORK**
  - displacement, load and energy errors, must be smaller than the tolerances EPSU, EPSP and EPSW
- **CONV RATE**
  - should be between 0 and 1; bigger than 1 means the solution will never converge

NONLINEAR OUTPUT CONTROL

```
        LOAD  NO.       - - ERROR FACTORS - -    CONV  ITR MAT NO.    AVG     TOTL       - - - - - DISP - - - - - -  LINE_S  NO. TOT TOT
        STEP  INC ITR  DISP     LOAD      WORK   RATE  DIV DIV BIS  R_FORCE   WORK      AVG        MAX     AT GRID C FACT NO QNV KUD ITR

%1.00000E+00   1  1 1.00E+00 2.69E+01 1.00E+00  1.000  1   1   0  4.92E+04 6.533E+06 1.06E+01 -1.587E+02        6 2  1.00 0  0   0    1
```

- **ITR DIV**
  - divergence counter, > MAXDIV triggers the divergence process
- **MAT DIV**
  - divergence counter for element and material routines
- **NO. BIS**
  - number of bisections
- **AVG R_FORCE**
  - average residual force (forces and moments); should be small
- **TOTL WORK**
  - approximate total work
- **DISP: AVG MAX AT GRID C**
  - average, maximum displacement at grid in direction c

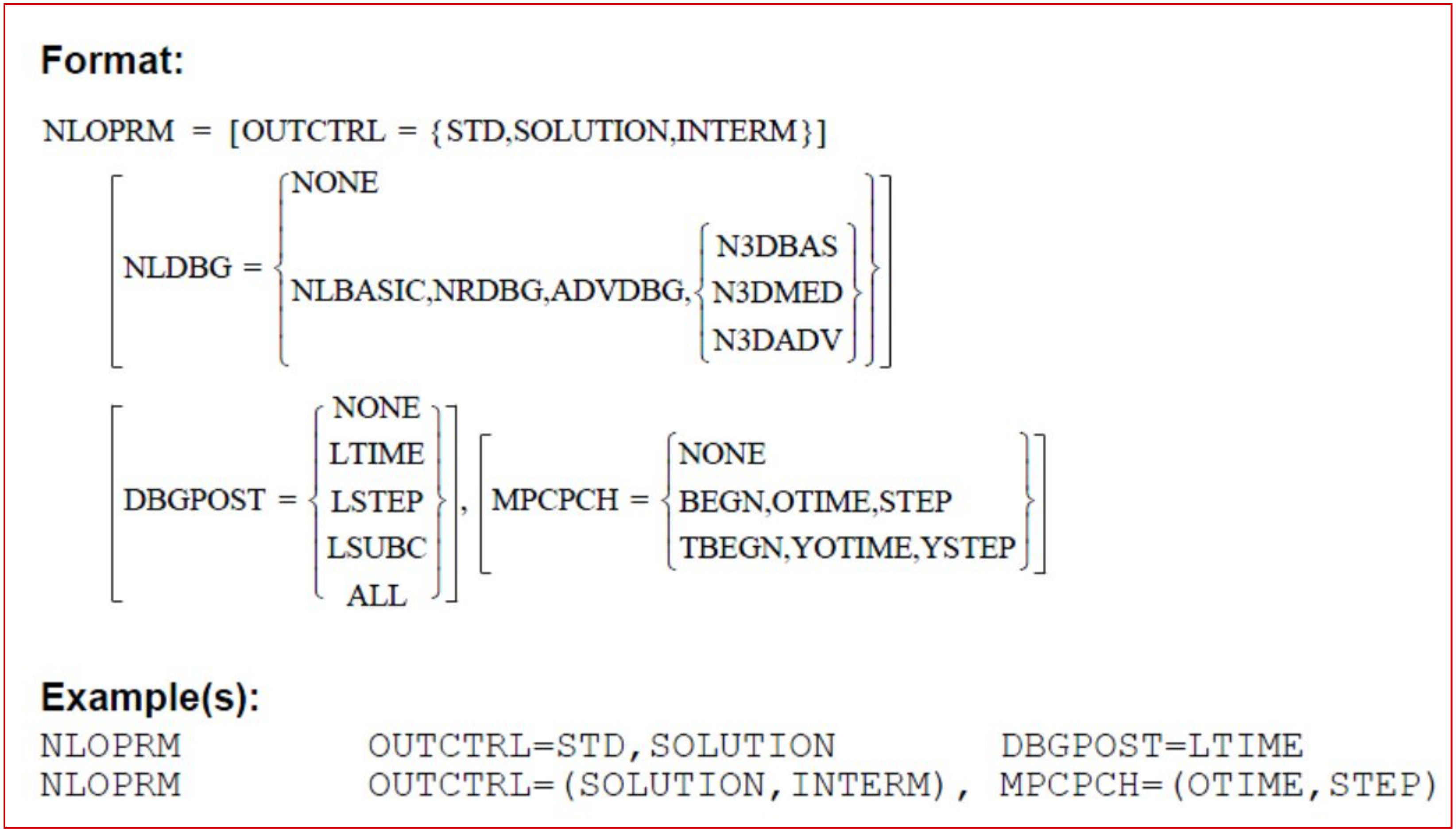
NONLINEAR OUTPUT CONTROL

```
        LOAD  NO.       - - ERROR FACTORS - -    CONV  ITR MAT NO.    AVG     TOTL        - - - - - DISP - - - - - -  LINE_S  NO. TOT TOT
        STEP  INC ITR  DISP     LOAD      WORK   RATE  DIV DIV BIS  R_FORCE   WORK      AVG         MAX     AT GRID C FACT NO QNV KUD ITR

%1.00000E+00   1  1 1.00E+00 2.69E+01 1.00E+00  1.000  1   1   0  4.92E+04 6.533E+06 1.06E+01 -1.587E+02         6 2  1.00 0  0   0    1
```

- **LINE_S: FACT NO**
  - line search factor α and number of line searches
- **NO. QNV**
  - number of Quasi Newton Vectors
- **TOT KUD / ITR**
  - total of stiffness updates / iterations

NONLINEAR OUTPUT CONTROL

- **Case Control command NLOPRM is also available for increased diagnostic output:**



Format:

$$\text{NLOPRM} = \left[\text{OUTCTRL} = \{\text{STD,SOLUTION,INTERM}\}\right]$$

$$\left[\text{NLDBG} = \begin{Bmatrix} \text{NONE} \\ \text{NLBASIC,NRDBG,ADVDBG,} \begin{Bmatrix} \text{N3DBAS} \\ \text{N3DMED} \\ \text{N3DADV} \end{Bmatrix} \end{Bmatrix}\right]$$

$$\left[\text{DBGPOST} = \begin{Bmatrix} \text{NONE} \\ \text{LTIME} \\ \text{LSTEP} \\ \text{LSUBC} \\ \text{ALL} \end{Bmatrix}\right], \left[\text{MPCPCH} = \begin{Bmatrix} \text{NONE} \\ \text{BEGN,OTIME,STEP} \\ \text{TBEGN,YOTIME,YSTEP} \end{Bmatrix}\right]$$

Example(s):

```
NLOPRM         OUTCTRL=STD,SOLUTION          DBGPOST=LTIME
NLOPRM         OUTCTRL=(SOLUTION,INTERM),  MPCPCH=(OTIME,STEP)
```

NONLINEAR OUTPUT CONTROL

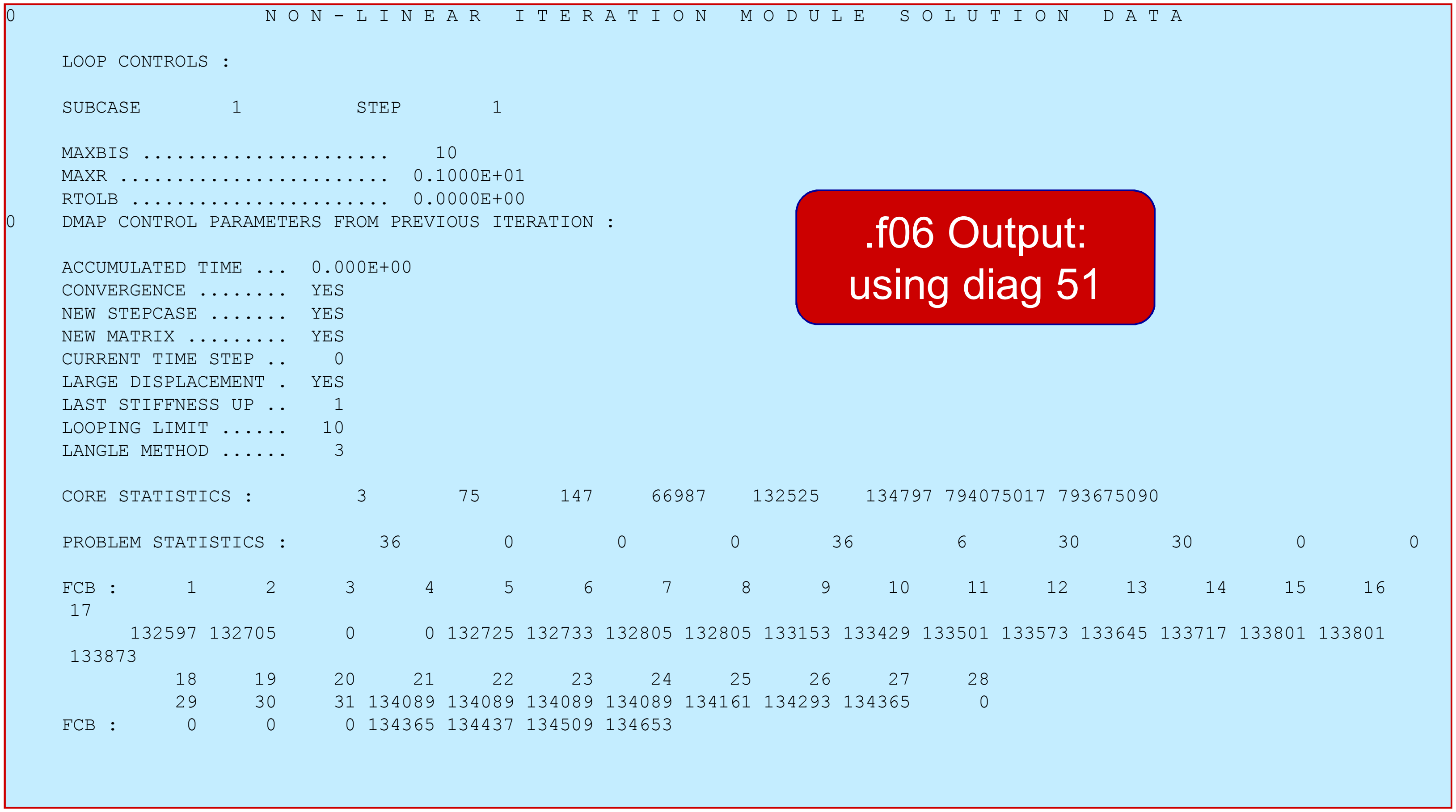
- **Example using: nloprm nldbg=advdbg**

```
 *** USER INFORMATION MESSAGE 6204 (NL3EMA)
     0.000000E+00 SECONDS REQUIRED TO DECOMPOSE MATRIX.

             maximum residual force   at node         6 degree of freedom  1 is equal to    5.566E+05
             maximum reaction force   at node         1 degree of freedom  1 is equal to    2.072E+04
             residual convergence ratio    2.687E+01

             maximum residual moment at node         5 degree of freedom  6 is equal to    4.656E+04
             maximum reaction moment at node         1 degree of freedom  6 is equal to    9.982E+04
             residual convergence ratio    4.664E-01

             maximum displacement  change   at node         6 degree of freedom  2 is equal to    1.587E+02
             maximum displacement increment at node         6 degree of freedom  2 is equal to    1.587E+02
             displacement convergence ratio   1.000E+00

             maximum   rotation    change   at node         6 degree of freedom  6 is equal to    2.381E-01
             maximum   rotation   increment at node         6 degree of freedom  6 is equal to    2.381E-01
             rotation convergence ratio   1.000E+00

             strain energy change at this iteration is    6.53309E+06
             strain energy change at this increment is    6.53309E+06
             relative energy error is                     1.00000E+00
%1.00000E+00   1  1 1.00E+00 2.69E+01 1.00E+00  1.000  1   1   0  4.92E+04 6.533E+06 1.06E+01 -1.587E+02        6  2  1.00  0   0   0
    1
```

.f06 Output:
First iteration

NONLINEAR OUTPUT CONTROL

- **Matrix level output can be accessed with *diag 51* request in the Executive Control section**
  - Before CEND entry



```
0                      N O N - L I N E A R   I T E R A T I O N   M O D U L E   S O L U T I O N   D A T A

     LOOP CONTROLS :

     SUBCASE        1          STEP        1

     MAXBIS ......................    10
     MAXR ........................  0.1000E+01
     RTOLB .......................  0.0000E+00
0    DMAP CONTROL PARAMETERS FROM PREVIOUS ITERATION :

     ACCUMULATED TIME ...  0.000E+00
     CONVERGENCE ........  YES
     NEW STEPCASE .......  YES
     NEW MATRIX .........  YES
     CURRENT TIME STEP ..    0
     LARGE DISPLACEMENT .  YES
     LAST STIFFNESS UP ..    1
     LOOPING LIMIT ......   10
     LANGLE METHOD ......    3

     CORE STATISTICS :         3        75       147     66987     132525     134797 794075017 793675090

     PROBLEM STATISTICS :        36         0         0         0        36         6        30        30         0         0

     FCB :      1      2      3      4      5      6      7      8      9     10     11     12     13     14     15     16
      17
           132597 132705      0      0 132725 132733 132805 132805 133153 133429 133501 133573 133645 133717 133801 133801
      133873
               18     19     20     21     22     23     24     25     26     27     28
               29     30     31 134089 134089 134089 134089 134161 134293 134365      0
     FCB :      0      0      0 134365 134437 134509 134653
```

.f06 Output:
using diag 51

NONLINEAR OUTPUT CONTROL

```
- Inside the Line Search Loop

    NL3LSL: DISPLACEMENT VECTOR (P-SET)

    1 :   0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00 -8.8914D+00  0.0000D+00  0.0000D+00
   11 :   0.0000D+00 -8.5714D-02  0.0000D+00 -3.3021D+01  0.0000D+00  0.0000D+00  0.0000D+00 -1.5238D-01  0.0000D+00 -6.8579D+01
   21 :   0.0000D+00  0.0000D+00  0.0000D+00 -2.0000D-01  0.0000D+00 -1.1176D+02  0.0000D+00  0.0000D+00  0.0000D+00 -2.2857D-01
   31 :   0.0000D+00 -1.5874D+02  0.0000D+00  0.0000D+00  0.0000D+00 -2.3810D-01

    NL3LSL: NONLINEAR INTERNAL FORCE VECTOR (P-SET)

    1 :  -2.0717D+04  1.0193D+03  0.0000D+00  0.0000D+00  0.0000D+00  9.9817D+04 -1.3046D+05  1.7284D+04  0.0000D+00  0.0000D+00
   11 :   0.0000D+00 -3.8105D+03 -1.7305D+05  3.9327D+04  0.0000D+00  0.0000D+00  0.0000D+00 -1.5121D+04 -1.4869D+05  4.4360D+04
   21 :   0.0000D+00  0.0000D+00  0.0000D+00 -3.1892D+04 -8.3709D+04  2.8614D+04  0.0000D+00  0.0000D+00  0.0000D+00 -4.6557D+04
   31 :   5.5663D+05 -1.3061D+05  0.0000D+00  0.0000D+00  0.0000D+00 -2.6159D+04

    NL3LSL: LOAD VECTOR (P-SET)

    1 :   0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00
   11 :   0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00
   21 :   0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00
   31 :   0.0000D+00 -1.0000D+02  0.0000D+00  0.0000D+00  0.0000D+00  0.0000D+00

    NL3DEL: LOADING ERROR VECTOR(FE-SET)

    1 :   1.3046D+05 -1.7284D+04  0.0000D+00  0.0000D+00  0.0000D+00  3.8105D+03  1.7305D+05 -3.9327D+04  0.0000D+00  0.0000D+00
   11 :   0.0000D+00  1.5121D+04  1.4869D+05 -4.4360D+04  0.0000D+00  0.0000D+00  0.0000D+00  3.1892D+04  8.3709D+04 -2.8614D+04
   21 :   0.0000D+00  0.0000D+00  0.0000D+00  4.6557D+04 -5.5663D+05  1.3051D+05  0.0000D+00  0.0000D+00  0.0000D+00  2.6159D+04
```

```
%1.00000E+00   1   1 1.00E+00 2.69E+01 1.00E+00  1.000   1    1    0   4.92E+04 6.533E+06 1.06E+01 -1.587E+02         6  2  1.00  0   0    0     1
```

RESTARTS

RESTARTS

- **Why restart?**
  - To eliminate the need to re-run portions of a run that are known to be ‘good’
  - Unconverged solutions – make changes as appropriate to enhance convergence and allow code to continue
  - perform additional load conditions
- **Procedure**
  - MSC Nastran restarts by using the MASTER/DBALL files from an initial run. This file must be saved if restarts are desired (i.e. do not use ‘scr=yes’ on command line)
  - In the restart file include an ASSIGN statement pointing to the MASTER/DBALL file
  - In the restart file include a NLRESTART statement to indicate at what point of the initial run that the restart is to begin from

RESTART

- **Restart via Case Control NLRESTART**

Required entries for restart

```
ASSIGN MASTER=‘cold.MASTER’
RESTART
SOL 400
CEND
NLRESTART SUBCASE 1, STEP 2, TIME 0.3    $ before any SUBCASE command
$
SUBCASE 1
        TSTEPNL = 10
        STEP 1              $ Case control of cold start with STEP 1 and 2
             LOAD = 10
        STEP 2              $ Restart begins at time 0.3 of this step
             TSTEPNL = 30
             LOAD = 20
        STEP 3
             LOAD = 30
BEGIN BULK
TSTEPNL,30, ,0.01,5
ENDDATA
```

RESTART

NLRESTART (SOL 400) **Nonlinear Restart Request**

Request a RESTART execution at a specified point for SOL 400.

Format:

$$\text{NLRESTART}\left[\text{SUBCASE } i\left[, \text{STEP } j\left[, \begin{matrix}\text{LOADFAC } f\\ \text{TIME } t\end{matrix}\right]\right]\right]$$

Example:

```
NLRESTART   SUBCASE 1, STEP 2, LOADFAC 0.3
```

Describer	Meaning
i	Specifies the identification number of a previously executed SUBCASE (Integer, Default is the first SUBCASE).
j	Specifies the identification number of a previously executed STEP (Integer, Default is the first STEP).
f	Specified the load factor of a previously executed load increment in nonlinear static analysis (Real, $0.0 \leq f \leq 1.0$, Default = 0.0).
t	Specified the time of a previously executed time step in nonlinear transient analysis (Real, $t_0 \leq t \leq t_n$, where t_0 is the initial time of STEP j, and t_n is the last time of STEP j; Default = t_0).

GENERAL GUIDELINES & LIMITATIONS

GENERAL GUIDELINES

- **Convergence of a nonlinear problem is not just about the convergence tolerance values or the criteria specified… It is an overall issue of model integrity and representation of reality**
- **It is strongly recommended that small tests be performed to gain experience of unknown (to you) element and solution types:**
  - To understand its limitations
  - To insure that it represents the behavior for the actual simulation
  - To prevent expensive "surprises" at the end of a project

GENERAL GUIDELINES

- **Single element tests are preferable (where possible) :**
  - It is much quicker and easier to verify the input and
  - Evaluate the response with only a few degrees of freedom
- **There are a number of sources of examples and benchmarks available that may help in this regard:**
  - The MD User's Guide
  - The Implicit Nonlinear Analysis Guide
  - This SOL 400 Seminar

GENERAL GUIDELINES

- **Perform the analysis and scrutinize the results from a static linear analysis to check the integrity/behavior of the basic model:**
  - For material nonlinearity simply increase the failure criteria
  - For geometrically nonlinear analyses turn off large strain and displacement options
    - param, lgdisp, -1 (no large displacements and rotations)
    - nlmopts, lrgstrn, 0 (small strains)
    - mdlprm, nldiff, 1 (no differential stiffness)
  - For contact, turn off friction for the first run (if physics permits it)

GENERAL GUIDELINES

- **Add each of the nonlinearities one by one to determine their effect on the solution and its convergence behavior**
  - For instance, start with contact, adding next any geometric nonlinearity and then finally any material nonlinearity, etc.
  - For contact analyses, contact conditions can be set to GLUED
  - Next step would be TOUCHING, but with a separation force of 1E20

GENERAL GUIDELINES

- The deformed shape often gives obvious clues as to why the simulation is not converging
- Exaggerating the deformation is a good way to pick up cracks in the model or localized effects from incorrect contact definition
- Animating with a reasonable deformation exaggeration can also be of help
- Caution - exaggerated deformation may be misleading for large deformation types of simulations with contact. Exaggeration may imply contact penetration, only ‘true scale’ plotting will show penetration.

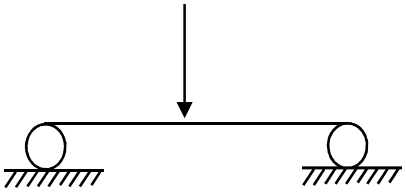
GENERAL GUIDELINES

- **Ensure consistent units are used throughout the model**
- **If experiencing convergence difficulties, reduce the time step or load. There may be significant nonlinearity occurring at the beginning.**
  - Usual for contact
  - May suggest an incorrect yield value for material nonlinearity
  - Buckling or significant rotation may have occurred
  - Excessive element distortion

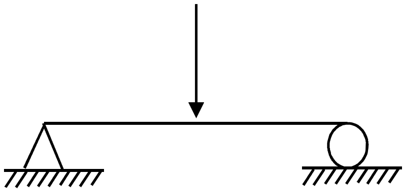
GENERAL GUIDELINES

- **If using the “fixed” load incrementation, change to the “adaptive” scheme and include the “automatic” criteria**

- **Check that each component of the structure is restrained against rigid body motion**
  - SOL 103 – normal modes – can help identify inadequate constraints – free body modes will be calculated in the direction of missing constraint



Unstable



Stable

GENERAL LIMITATIONS

- **Omitted degrees-of-freedom (o-set) may not be used.**
- **For Perturbation, buckling is not supported**
- **For Arc Length Method SPCD is not supported**

SECTION 4

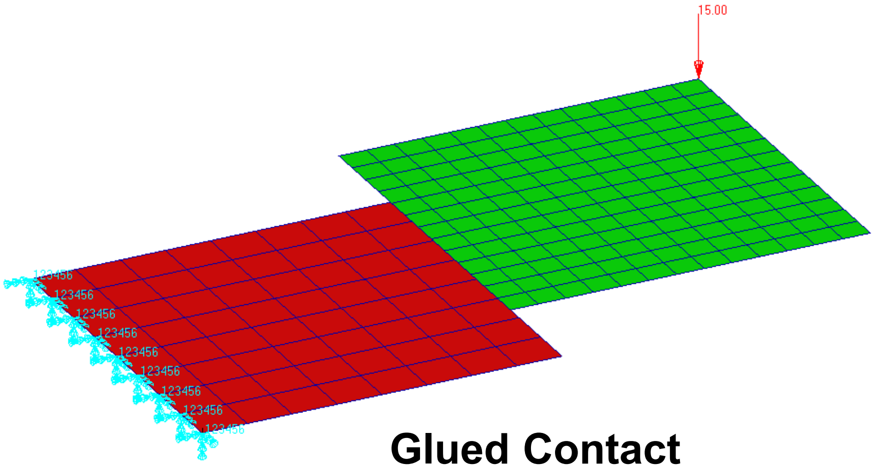
CONTACT IN MSC NASTRAN SOL400

OVERVIEW

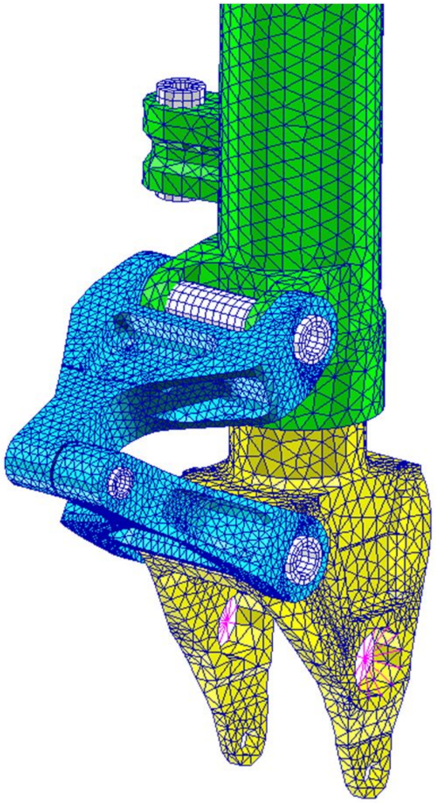
- **What is and when to consider contact**
- **Contact Bodies** (Rubber Door Seal Workshop)
- **Contact Pairs/Tables** (Deformable to Rigid Contact Workshop)
- **Contact Detection**
  - Node to segment
  - Segment to segment
- **Special Features**
  - Interference (Interference Fit Workshop)
  - Stress free initial contact
  - Glued contact (Contact Pairs Workshop)
  - Contact with Shells
  - Friction
- **Convergence and Controls with contact**

WHAT IS CONTACT ANALYSIS?

- **Contact analysis is the analysis of contact bodies (deformable or rigid) interacting with each other**
- **Contact can be deformable-deformable or rigid-deformable**
- **Contact analysis types**
  - Touching Contact
  - Glued Contact



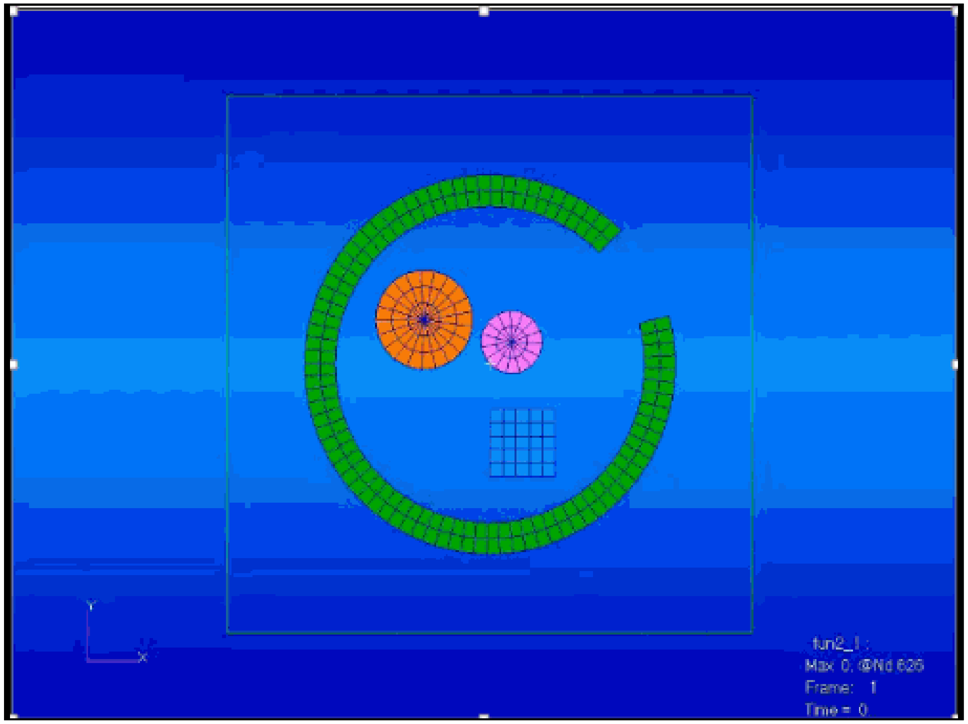
Glued Contact

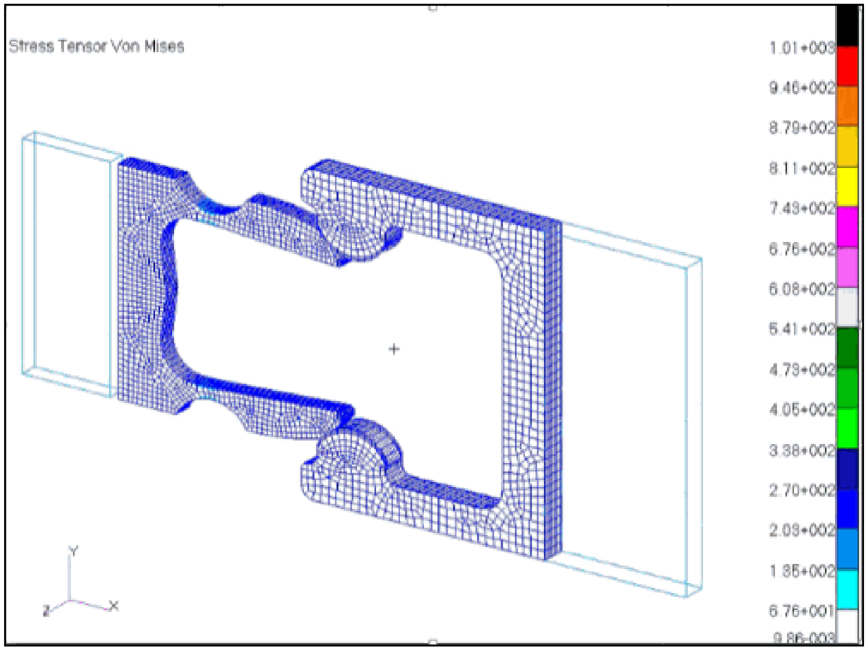


Touching Contact

WHEN TO CONSIDER CONTACT

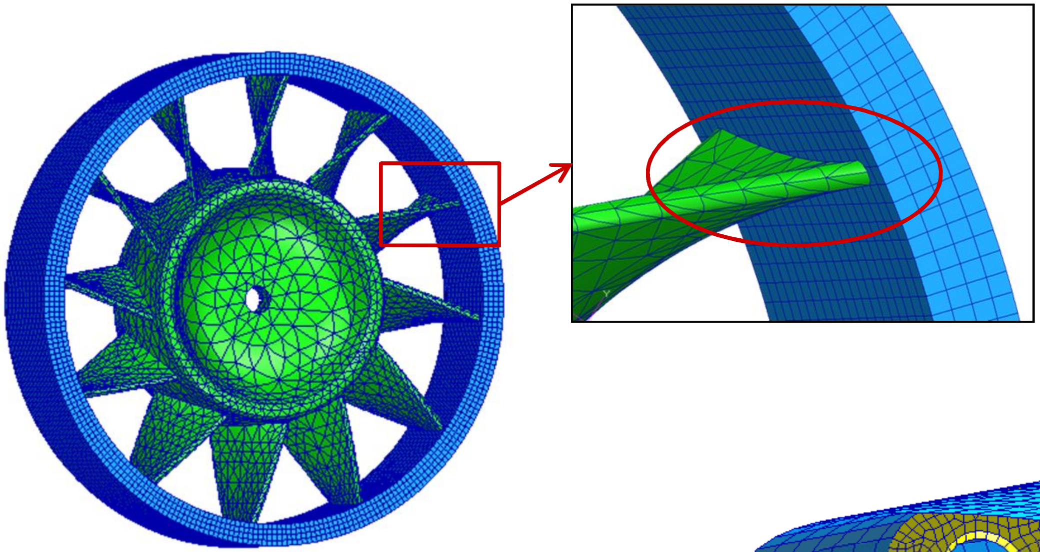
- **Intermittent interaction between unique components**



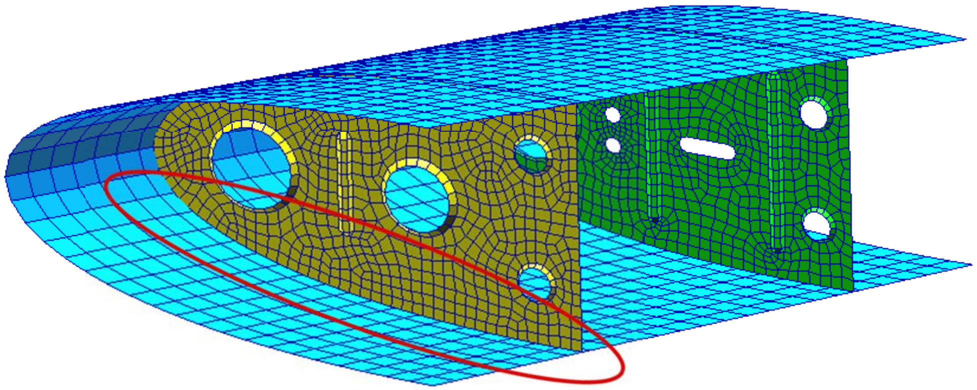


WHEN TO CONSIDER CONTACT

- **Connections between dissimilar meshes**



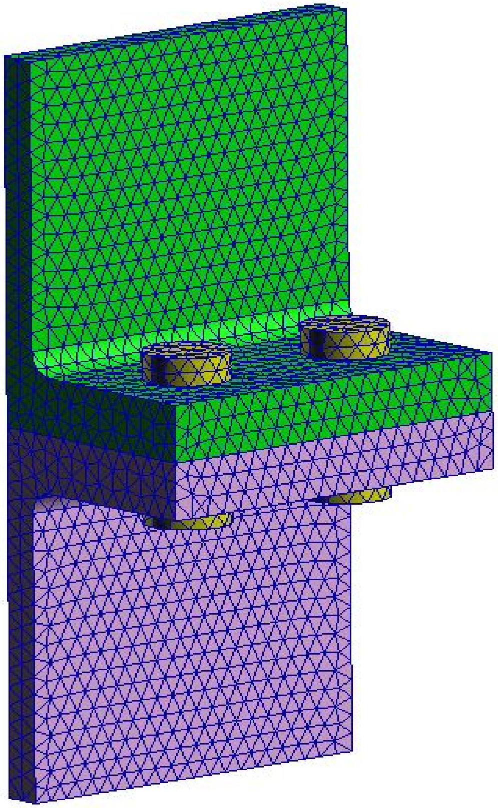
Solid Face-to-Face



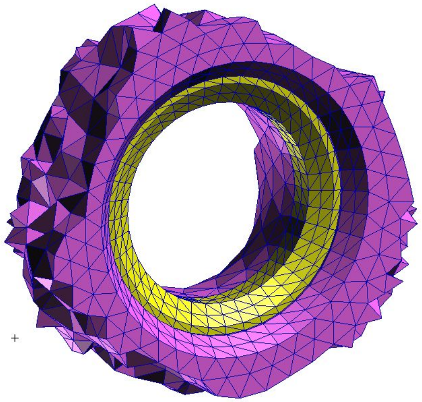
Shell Edge-to-Face

CONTACT ANALYSIS EXAMPLES

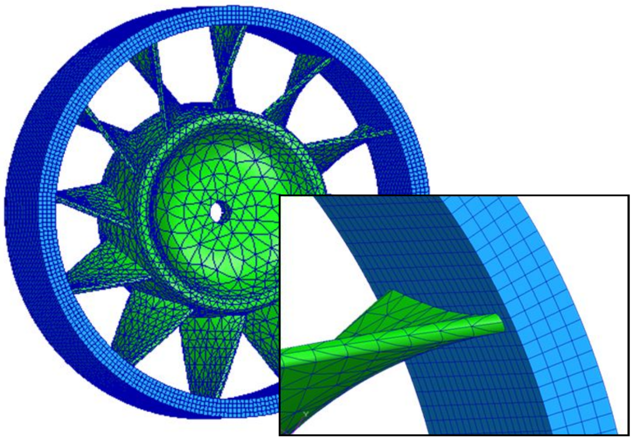
- **Solid-to-solid contact examples**



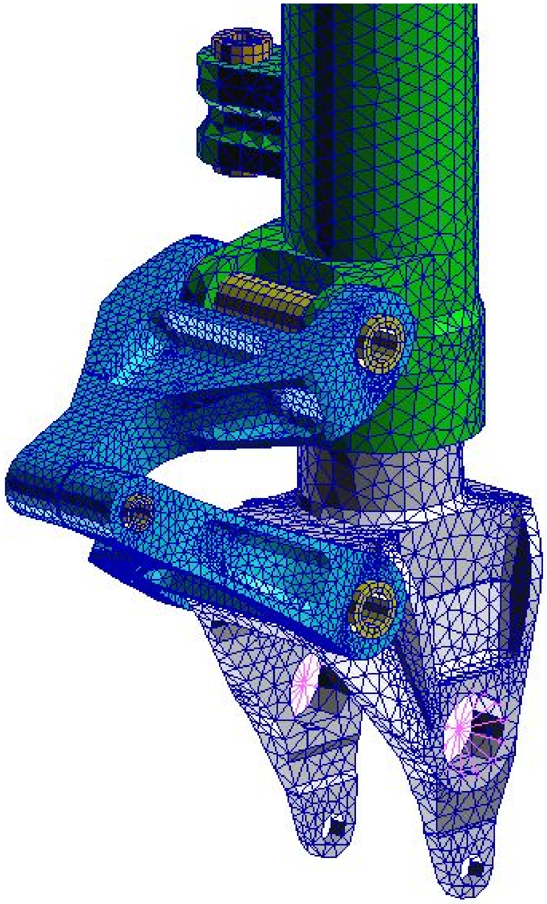
Preloaded Bolted Joint



Interference Fit



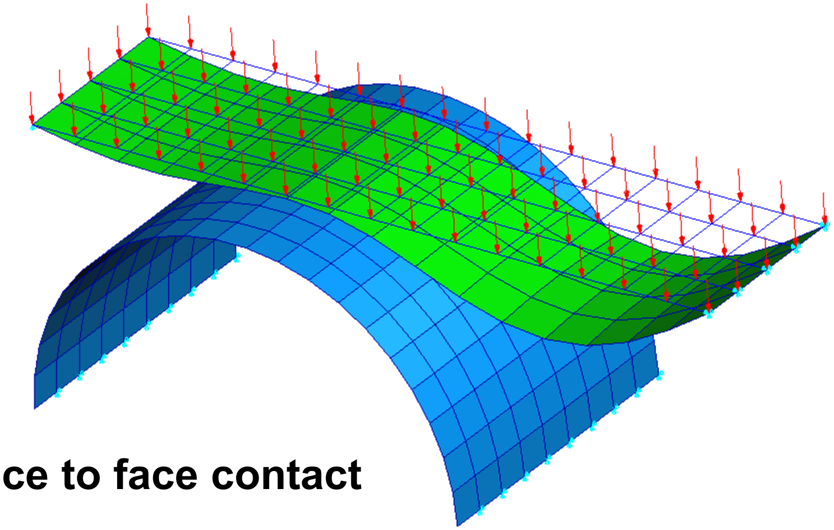
Glued Assembly



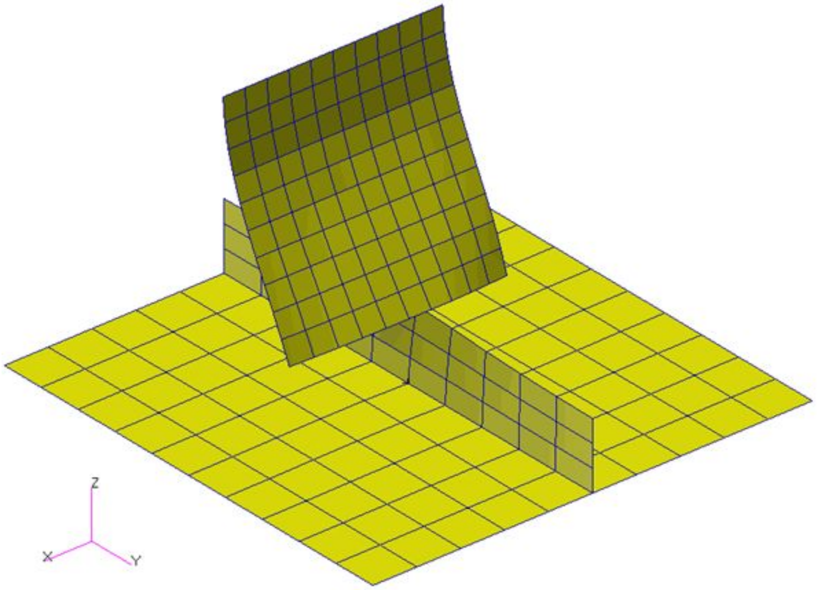
Lug-Clevis-Pin

CONTACT ANALYSIS EXAMPLES

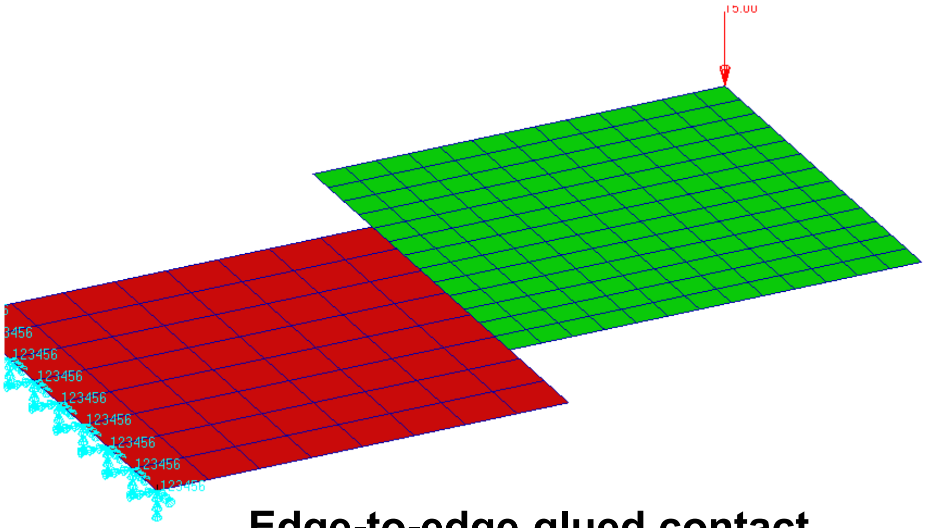
- **Shell-to-shell contact examples**



Face to face contact



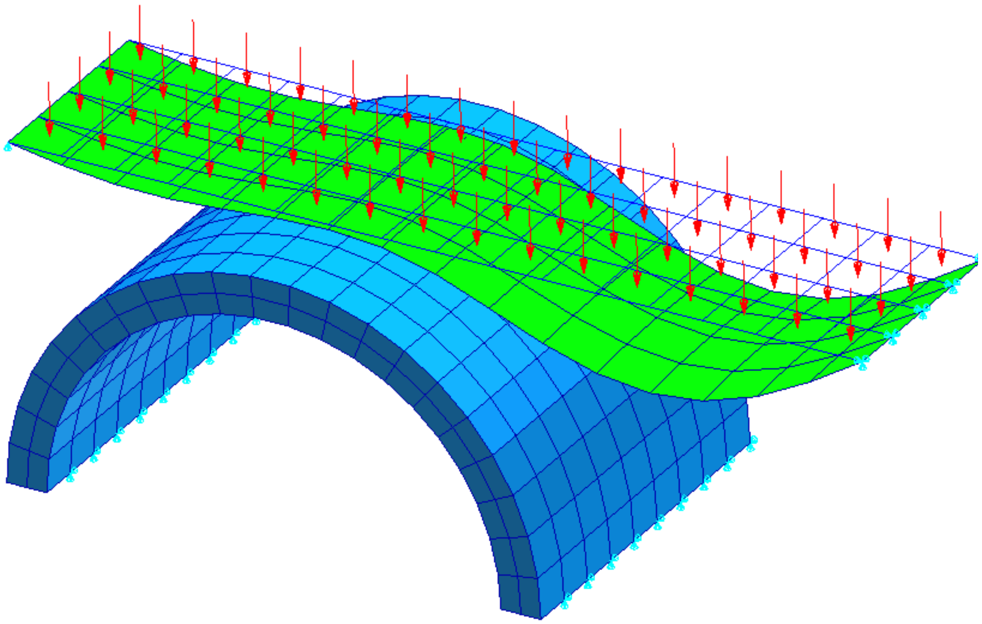
Edge-to-edge contact



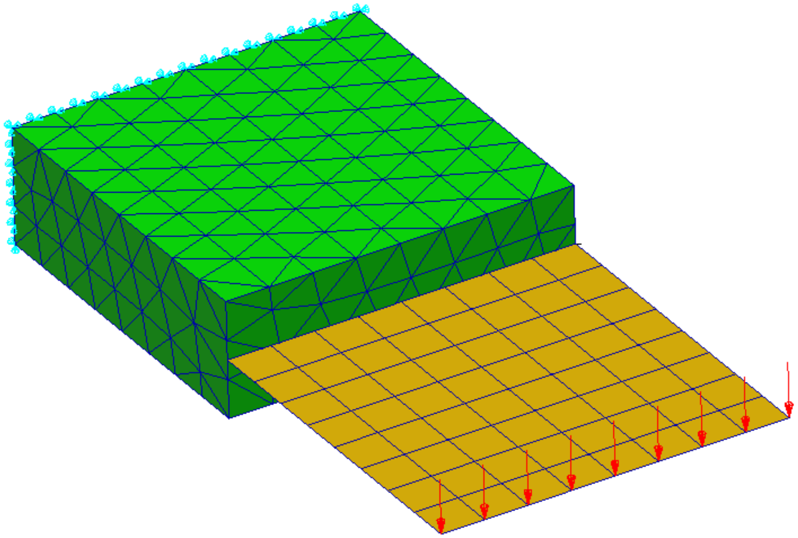
Edge-to-edge glued contact

CONTACT ANALYSIS EXAMPLES

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



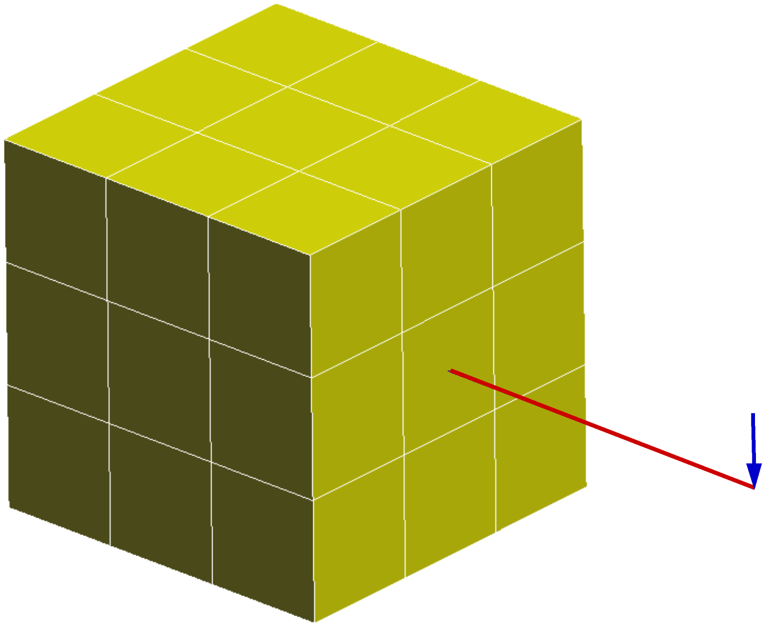
Face-to-face contact



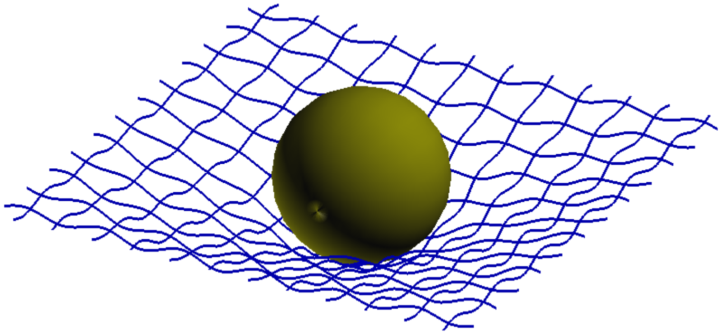
Edge-to-face glued contact

CONTACT ANALYSIS EXAMPLES

- **Beam contact examples**



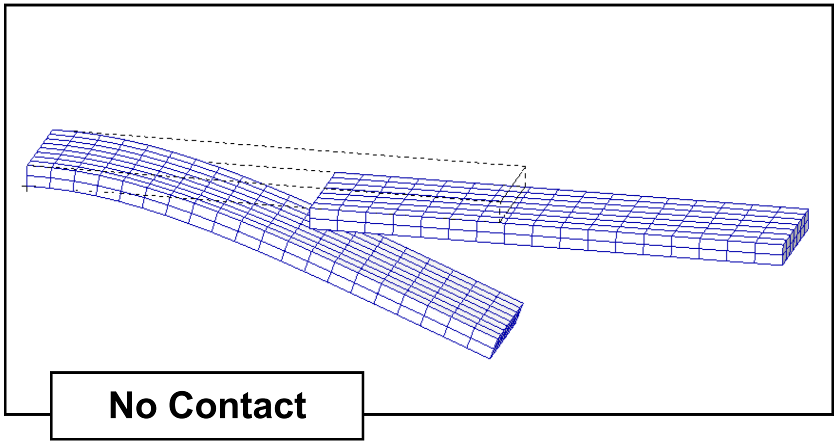
Beam-to-solid glued contact



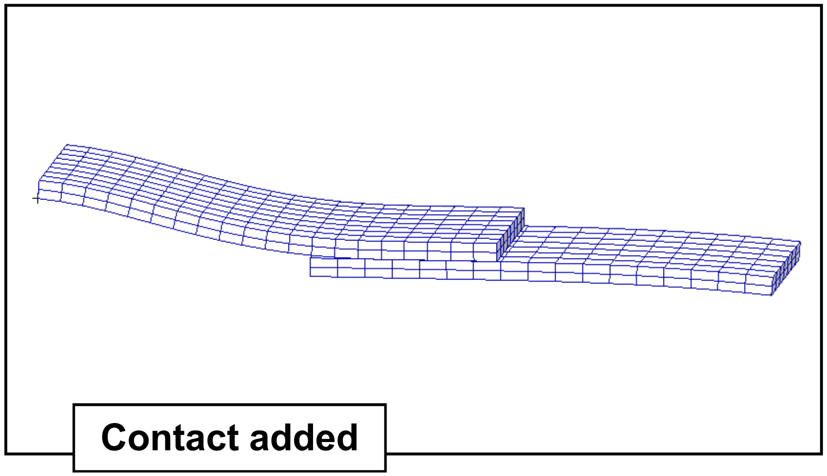
Beam-to-beam contact

CASE STUDY

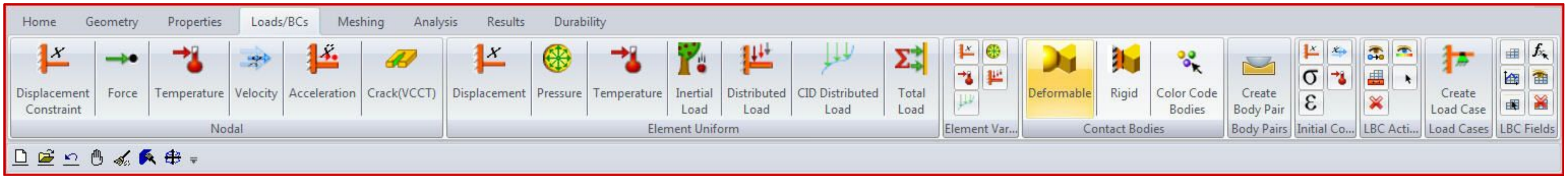
- **Initial Pass: No Contact definition**
  - Parts don’t see each other
  - Parts pass through each other



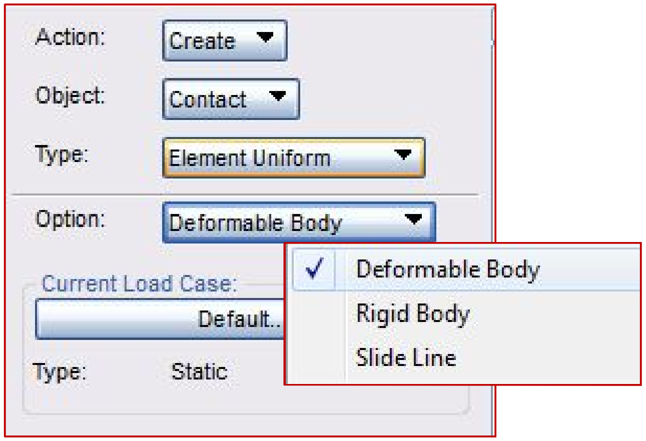
- **Second Pass: Add Contact Body definition**
  - Parts now contact each other



CASE STUDY



- **Easy contact body setup:**



Action: Create

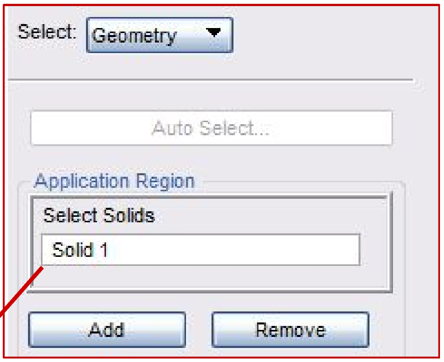
Object: Contact

Type: Element Uniform

Option: Deformable Body

Current Load Case: Default

Type: Static

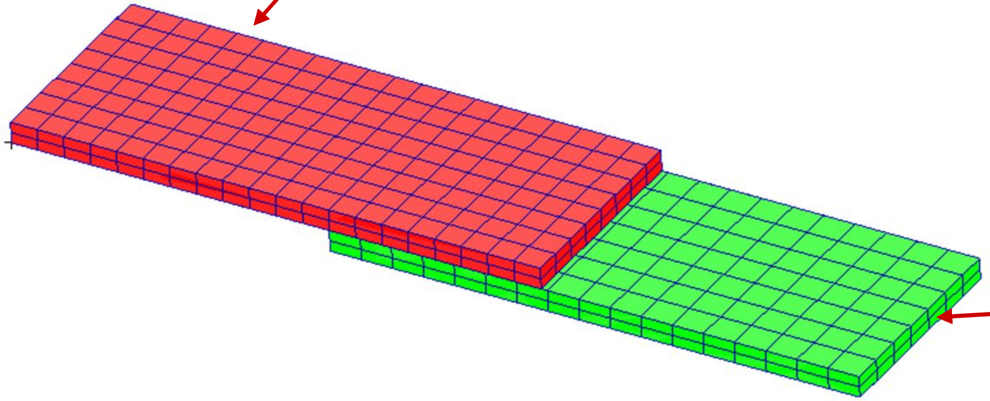


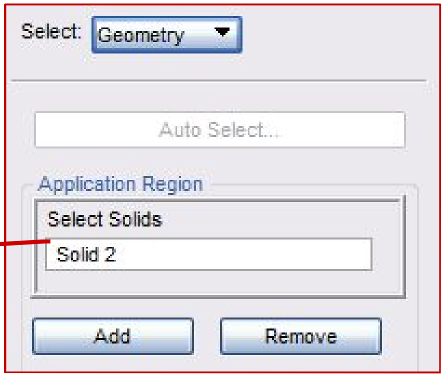
Select: Geometry

Application Region

Select Solids

Solid 1





Select: Geometry

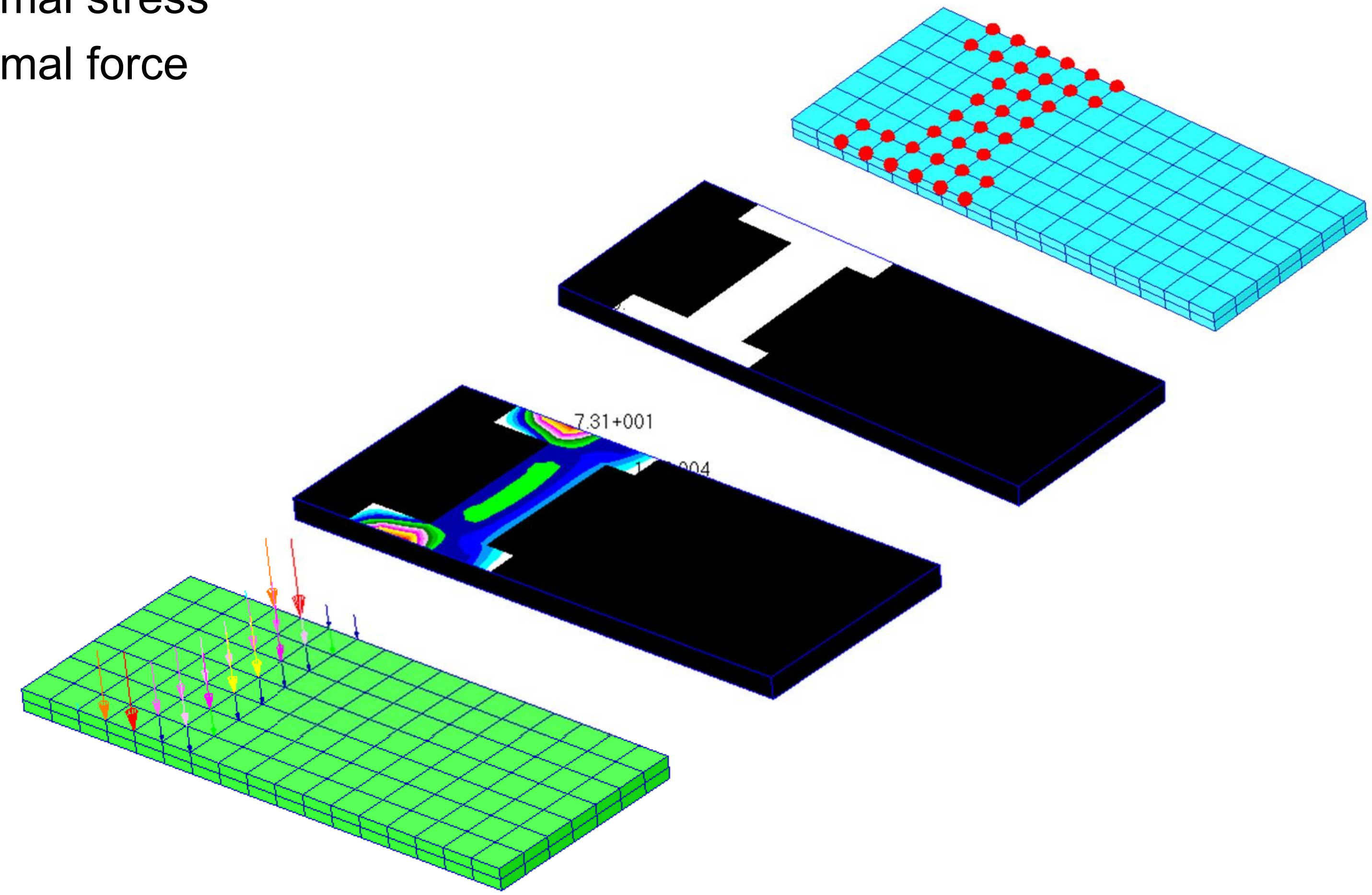
Application Region

Select Solids

Solid 2

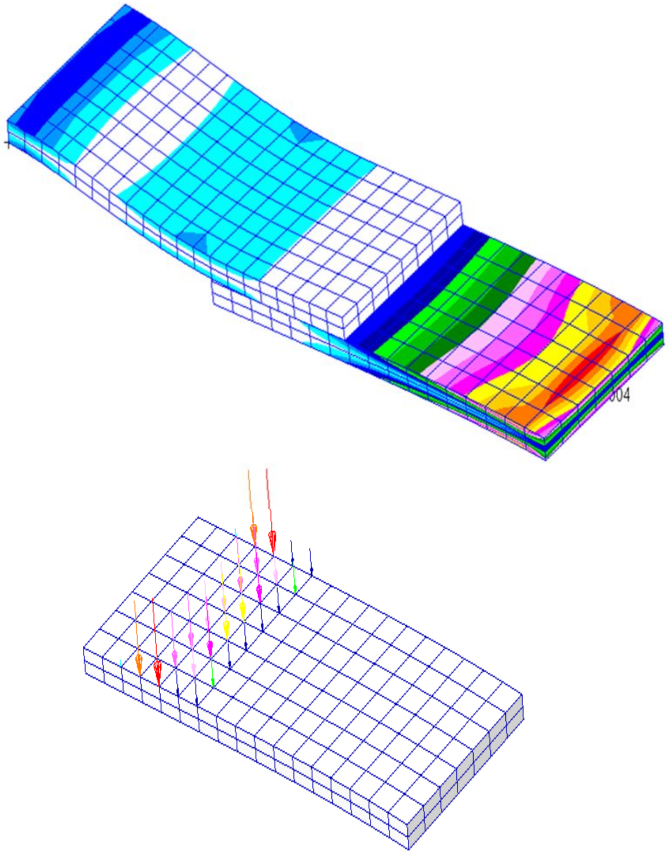
CASE STUDY

- **Comprehensive contact results plots:**
  - Contact status
  - Contact normal stress
  - Contact normal force



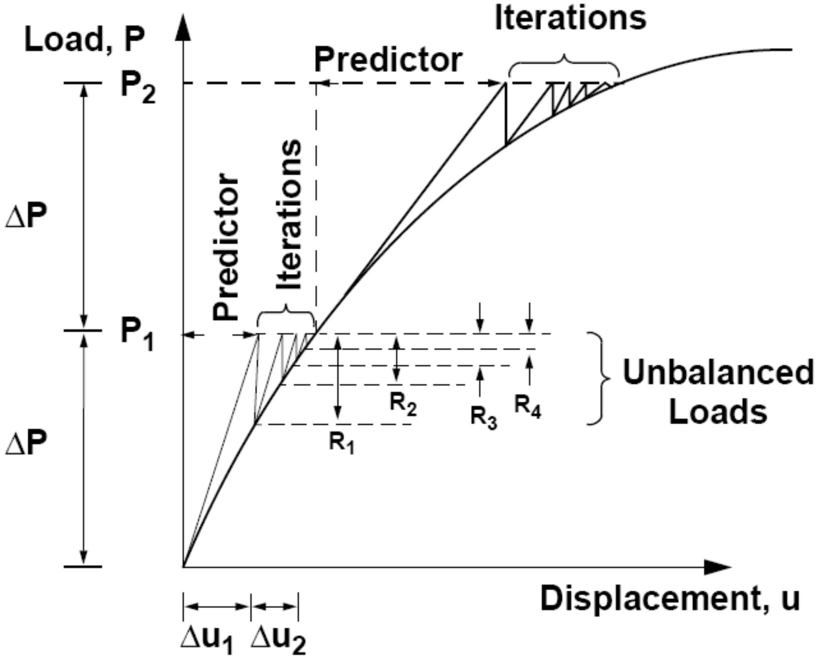
CASE STUDY

- **Summary of Case Study**
  - Studied contact between two parts with dissimilar meshes
  - Demonstrated easy contact setup
  - Plotted contact normal stress and force to gain insight into load path and contact status



CONTACT ANALYSES

- **Its important to consider that contact, by its nature, is non-linear**
  - Superposition is no longer valid
  - Solutions are load sequence dependent
  - Since the system of equations has become nonlinear, an iteration strategy is needed – more details on the iterative strategy is given in the section of this course on ‘Nonlinear Solution Strategies’:



TYPES OF CONTACT IN NASTRAN

- **Linear – SOL101**
  - Small deflection theory
  - Sliding neglected
  - All aspects of the simulation are ‘linear’ with the exception structural contact
  - Glued – SOL101/103/105/108/109/111/112

- **Nonlinear – SOL400 – the focus of this course**
  - Large deformation
    - Allows sliding between element edges/faces
  - No limitations

CONTACT IN SOL 400 and SOL 101

- **Two versions of contact are available**
  - Node to Segment
  - Segment to Segment
- **Two Friction Models are supported**
  - Bilinear Coulomb Friction
  - Bilinear Shear Friction
- **Glued Contact can be permanent or general, controlled via NLGLUE on BCPARA**
  - NLGLUE=0, permanent
  - NLGLUE=1, general

CONTACT IN OTHER SOLUTION SEQUENCES

- **SOL 400 has general 2D and 3D contact capability**
- **SOL 101 has only 3D Contact capability**
- **The contact in SOL 101 is linear in the sense that materials and geometry stay linear**
- **Higher order elements are supported**
- **In other solution sequences only permanent glued contact is available. This is true for SOLs 103, 105, 107, 108, 109, 110, 111, 112 and 200.**
  - Permanent Glued Contact is a special case of glued contact
  - Primarily used to join 2 dissimilar meshes
  - Contact must initially be true (bodies should be in contact initially)
  - When edges or grids are to be glued, gluing can also be done for the rotational DOFs (Moment Carrying Glue)
  - Permanent contact constraint MPC equations are used. No nonlinear increments or iterations involved

OVERVIEW

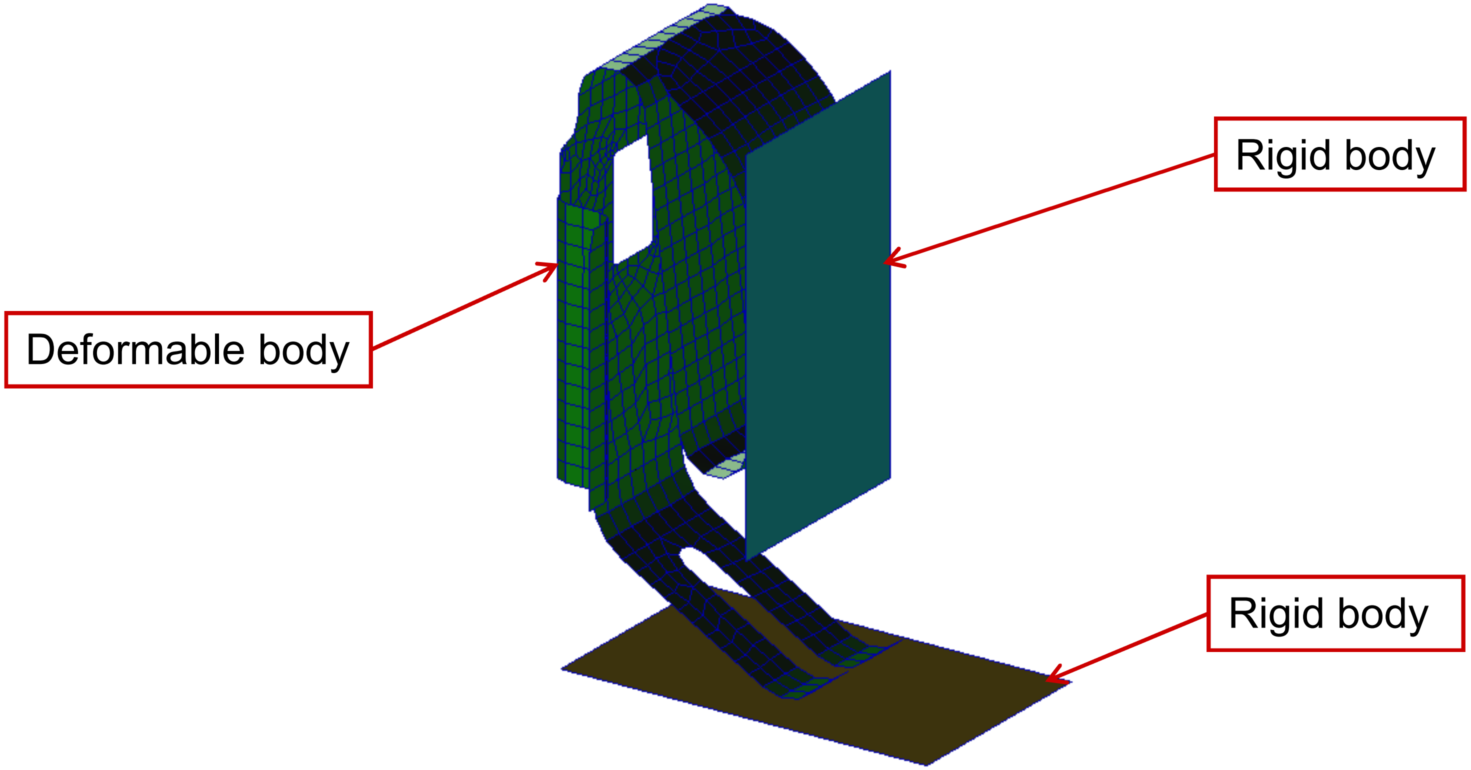
- What is and when to consider contact
- **Contact Bodies (Workshop - Rubber Door Seal)**
- Contact Pairs/Tables (Workshop – Deformable to Rigid Contact)
- Contact Detection
  - Node to segment
  - Segment to segment
- Special Features
  - Interference (Workshop – Interference Fit)
  - Stress free initial contact
  - Glued contact (Workshop – Contact Pairs)
  - Contact with Shells
  - Friction
- Convergence and Controls with contact

CONTACT BODIES

- **Contact evaluation requires the creation of ‘Contact Bodies’**
- **Contact Bodies are groups of elements or geometries that are to be evaluated for contact with other Contact Bodies**
- **There are 2 types of Contact Bodies:**
  - Deformable
  - Rigid
- **Contact can occur between Deformable-Deformable or Deformable-Rigid**

CONTACT BODY TYPES

- **Contact bodies can be deformable or rigid**
- **A deformable body is defined by element IDs or element properties**
- **A rigid body is defined by geometry (curves and surfaces) or 4-node patches**



DEFINITION OF DEFORMABLE BODIES

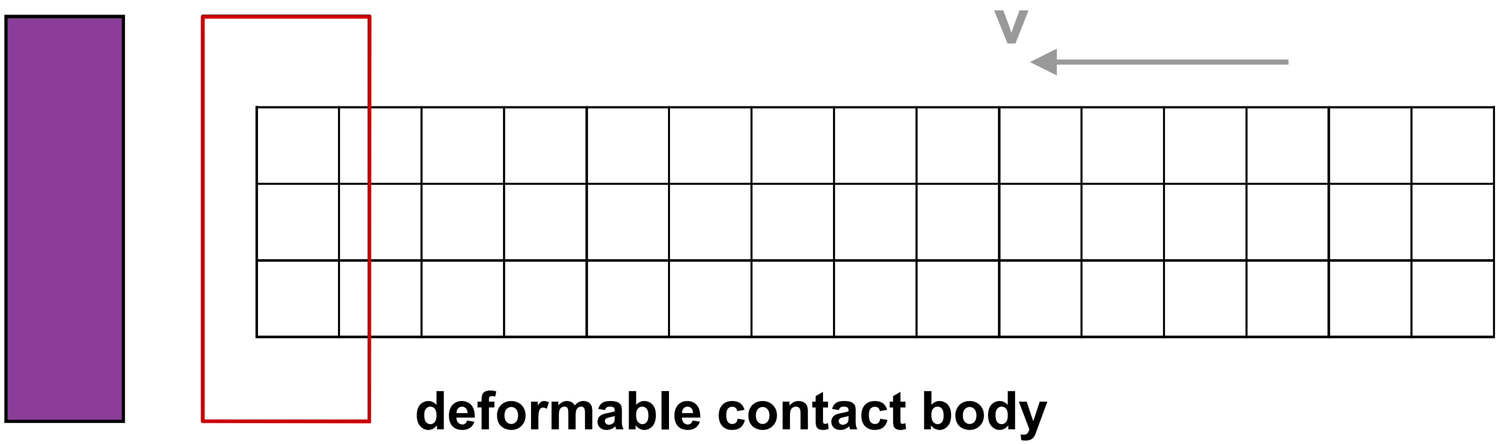
- **A deformable body is a collection of finite elements**
- **A deformable body must contain elements of the same class:**

Linear shell	CTRIA3/CQUAD4
Quadratic shell	CTRIA6/CQUAD8
Linear solid	CHEXA/CPENTA/CTETRA
Quadratic solid	CHEXA/CPENTA/CTETRA
Beams	CBAR/CBEAM/CROD

- **Quadratic contact is supported and is automatically activated when contact body contains higher order elements**

DEFORMABLE BODIES

- 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:

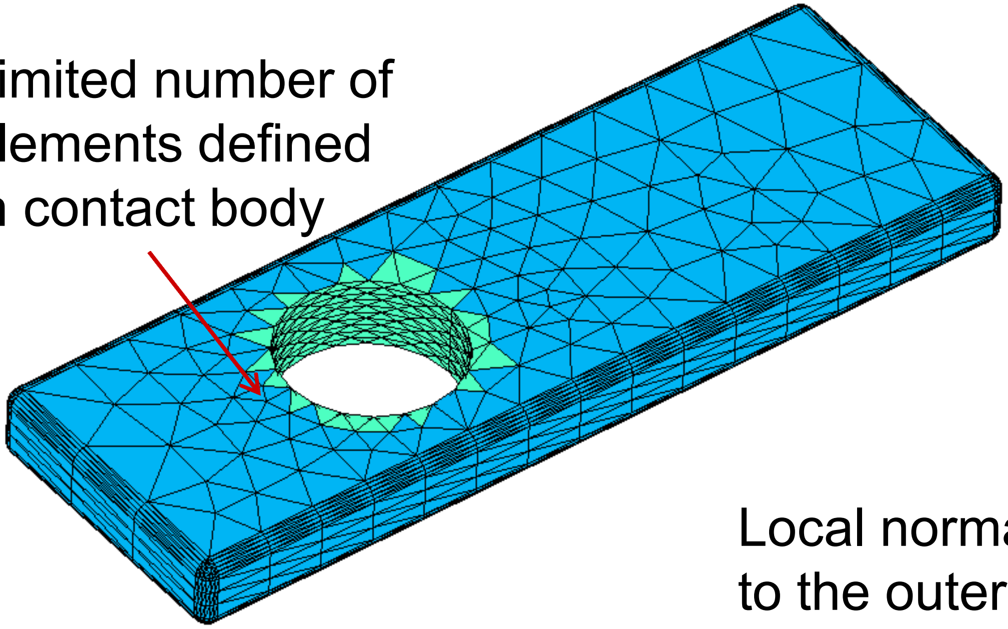


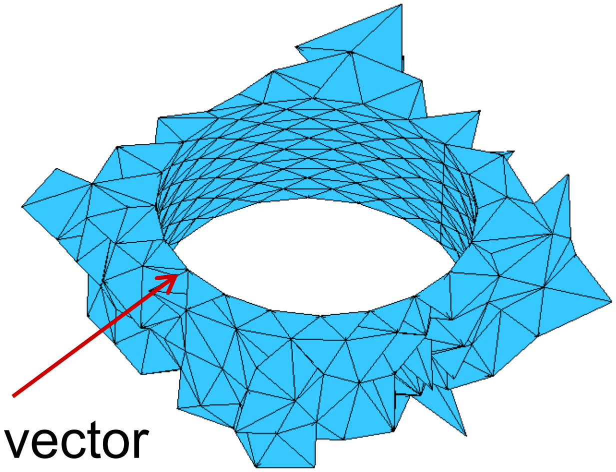
deformable contact body

DEFINITION OF DEFORMABLE BODIES

- **On the other hand, be careful with a subset of elements**
  - When using tet elements, discontinuous element normals cause numerical problems. Hence the use of a subset of tet elements should be avoided:

Limited number of elements defined in contact body

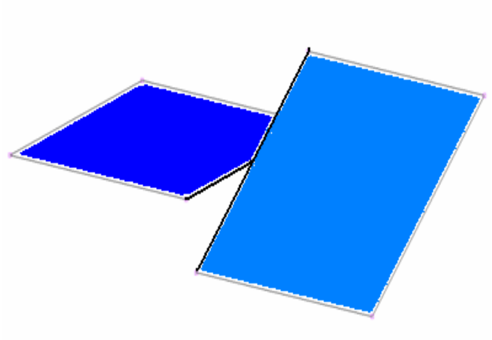


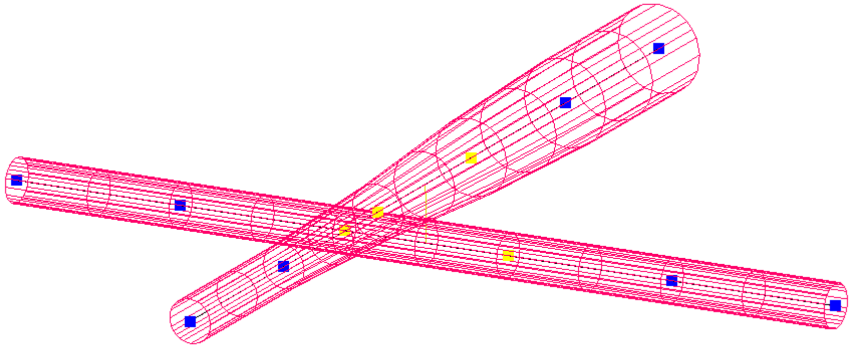


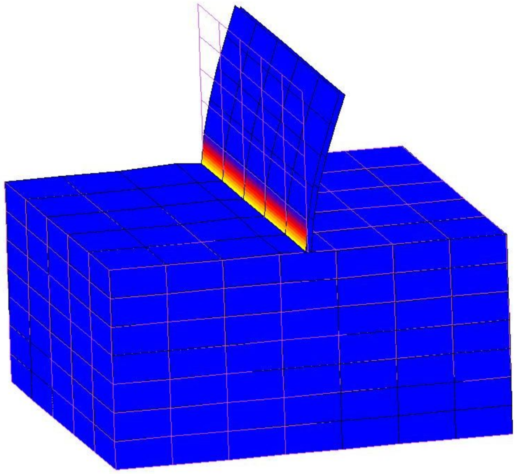
Local normal vector to the outer boundary may be completely wrong

DEFORMABLE-DEFORMABLE CONTACT

- **Further divided into the following types:**
  - Surface to Surface
  - Edge to Surface
  - Edge to Edge
  - Beam to Beam

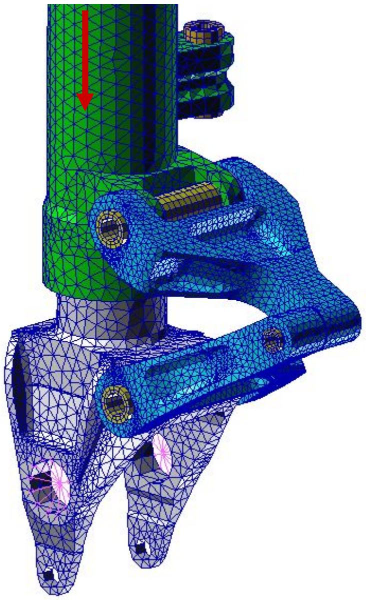


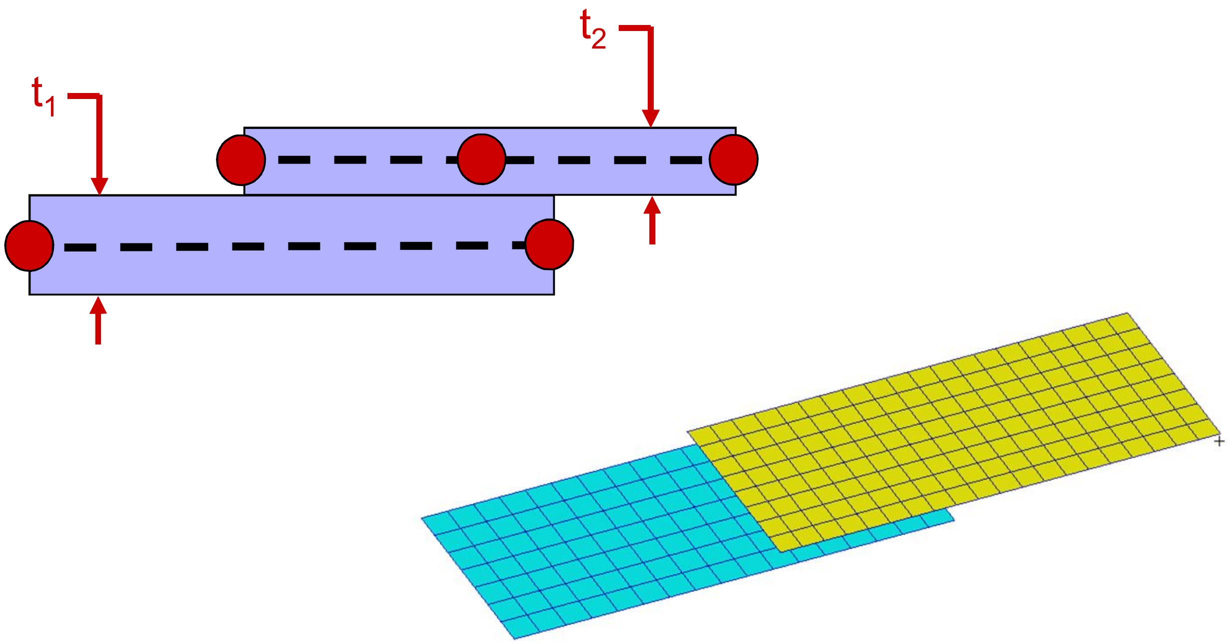




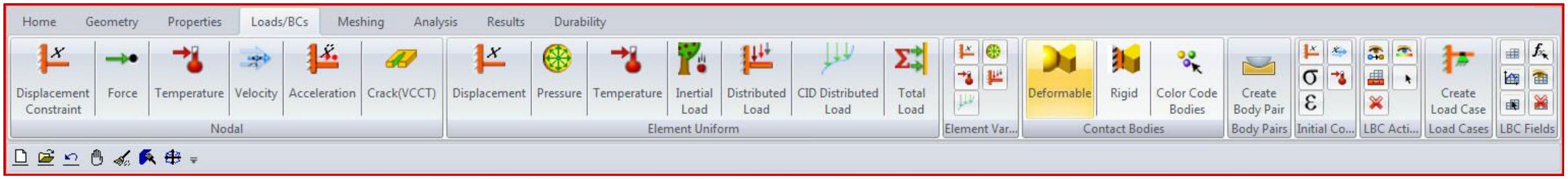
DEFINITION OF DEFORMABLE BODIES

- 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

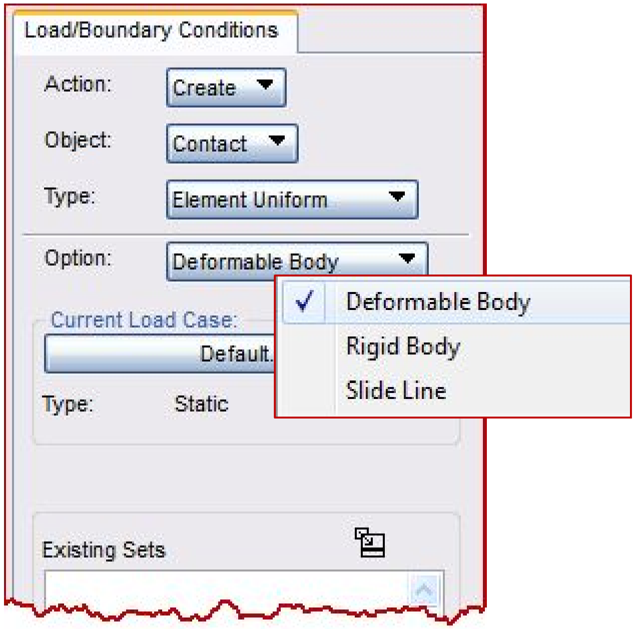




CREATING A DEFORMABLE BODY



- **Under the Patran Loads/BCs tab:**
  - Click Deformable
  - Enter New Set Name
  - Pick Target Element Type

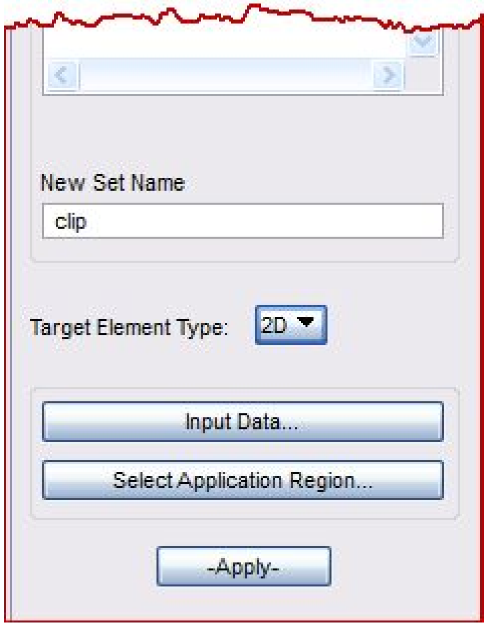

Load/Boundary Conditions

Action: Create

Object: Contact

Type: Element Uniform

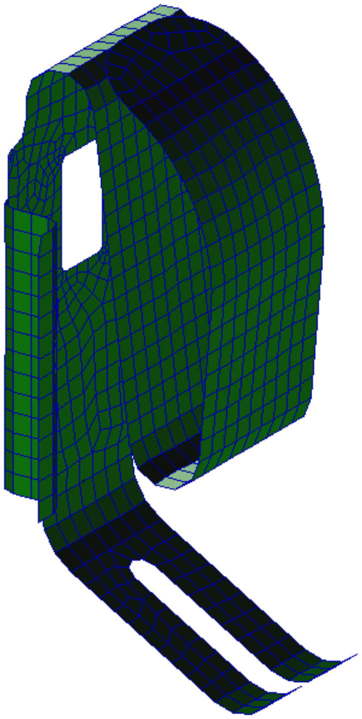
Option: Deformable Body

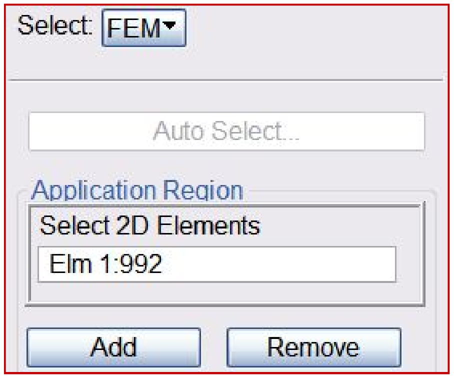

New Set Name: clip

Target Element Type: 2D

CREATING A DEFORMABLE BODY

- **On Select Application Region form: Select elements**
  - Directly
  - Based on geometry (if elements are associated with geometry) or
  - Based on associated property



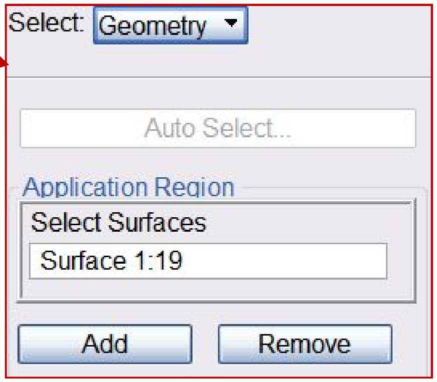


Select: FEM

Select 2D Elements

Elm 1:992

or

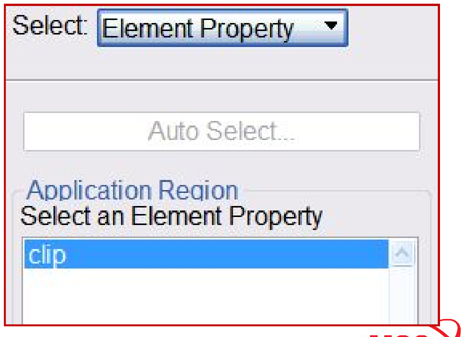


Select: Geometry

Select Surfaces

Surface 1:19

or



Select: Element Property

Select an Element Property

clip

CREATING A DEFORMABLE BODY

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

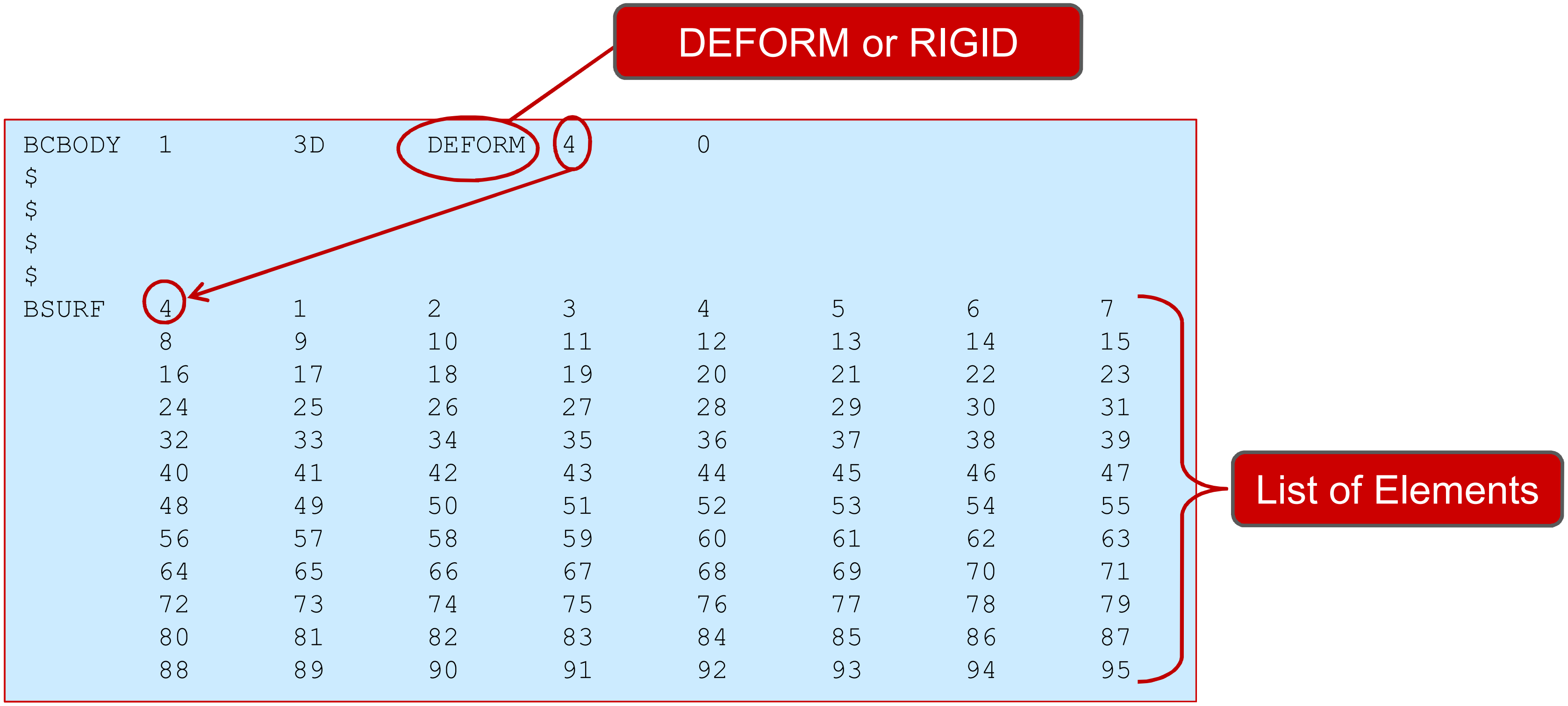
BCBODY	BID	DIM	BEHAV	BSID	ISTYP	FRIC	IDSPL	CONTROL	
	NLOAD	ANGVEL	DCOS1	DCOS2	DCOS3	VELRB1	VELRB2	VELRB3	
	“ADVANCE”	SANGLE	COPTB						
	“RIGID”	CGID	NENT	--- Rigid Body Name ---					
	“GROW”	GE1	GE2	GE3	TAB-GE1	TAB-GE2	TAB-GE3		

BSURF	ID	ELID1	ELID2	ELID3	ELID4	ELID5	ELID6	ELID7	

- **BSURF – Defines a contact body by Element IDs**
  - Referenced by BSID in BCBODY

CREATING A DEFORMABLE BODY

- **Sample MSC Nastran input file:**



DEFORM or RIGID

```
BCBODY  1       3D      DEFORM  4       0
$
$
$
$
BSURF   4       1       2       3       4       5       6       7
        8       9       10      11      12      13      14      15
        16      17      18      19      20      21      22      23
        24      25      26      27      28      29      30      31
        32      33      34      35      36      37      38      39
        40      41      42      43      44      45      46      47
        48      49      50      51      52      53      54      55
        56      57      58      59      60      61      62      63
        64      65      66      67      68      69      70      71
        72      73      74      75      76      77      78      79
        80      81      82      83      84      85      86      87
        88      89      90      91      92      93      94      95
```

List of Elements

CREATING A DEFORMABLE BODY

- **Property-based MSC Nastran entry:**
  - BCPROP: Defines a contact body to Element Property (Referenced by BSID in BCBODY)

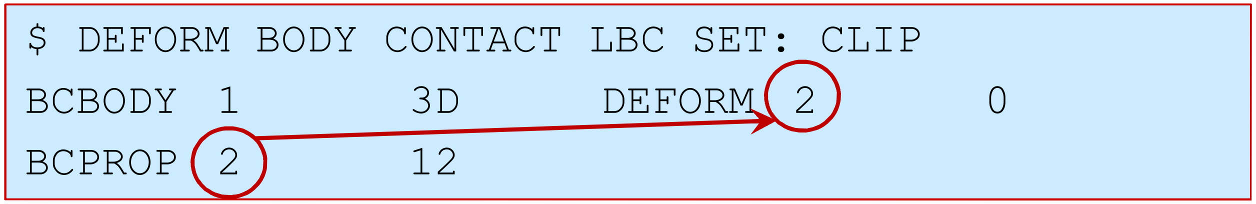
1	2	3	4	5	6	7	8	9	10
BCPROP	ID	IP1	IP2	IP3	IP4	IP5	IP6	IP7	
	IP8	IP9	etc.						

- **Example**

BCPROP	1	101	20	301					

CREATING A DEFORMABLE BODY

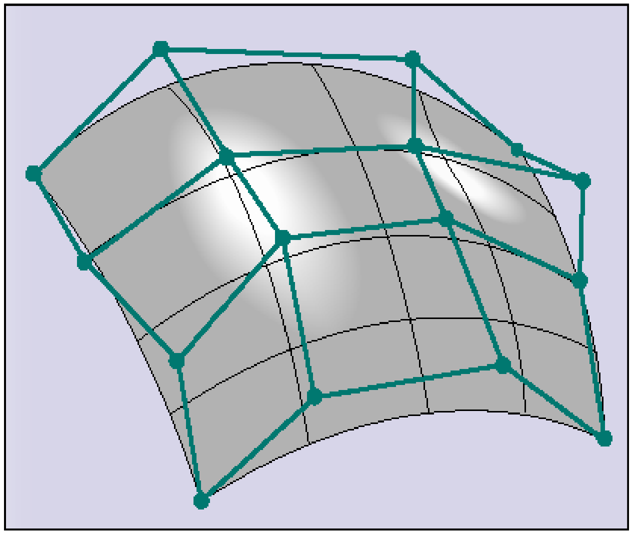
- **Sample MSC Nastran input file:**



```
$ DEFORM BODY CONTACT LBC SET: CLIP
BCBODY  1      3D      DEFORM 2       0
BCPROP  2      12
```

RIGID BODIES

- **Rigid Bodies can be modeled with geometry or finite elements**
  - Curves for 2D contact (NURBS2D)
    - Allows for 3 DOF – UX, UY, and ROTZ
  - Surfaces for 3D contact (NURBS) →
    - Allows for 6 DOF – UX, UY, UZ, ROTX, ROTY, and ROTZ
  - 2D elements (PATCH3D), 4-node patches only, triangular patches are not supported



- **It is recommended to use Bezier or NURBs:**
  - Continuity of the normal vector along the surface
  - A mathematical description
  - Robustness of the Contact Algorithm

DEFINITION OF RIGID BODIES

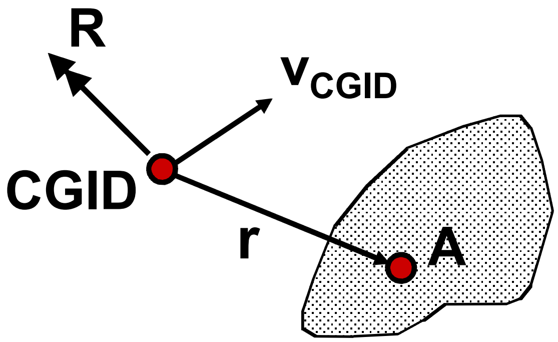
- **Rigid bodies can be stationary (default) or moved in space**
- **There are three different methods of controlling rigid body movements. The input is done via CONTROL on the BCBODY entry**
  - 0, velocity controlled
  - -1, position controlled
  - Positive Integer, load controlled

1	2	3	4	5	6	7	8	9	10
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 ---					

DEFINITION OF RIGID BODIES

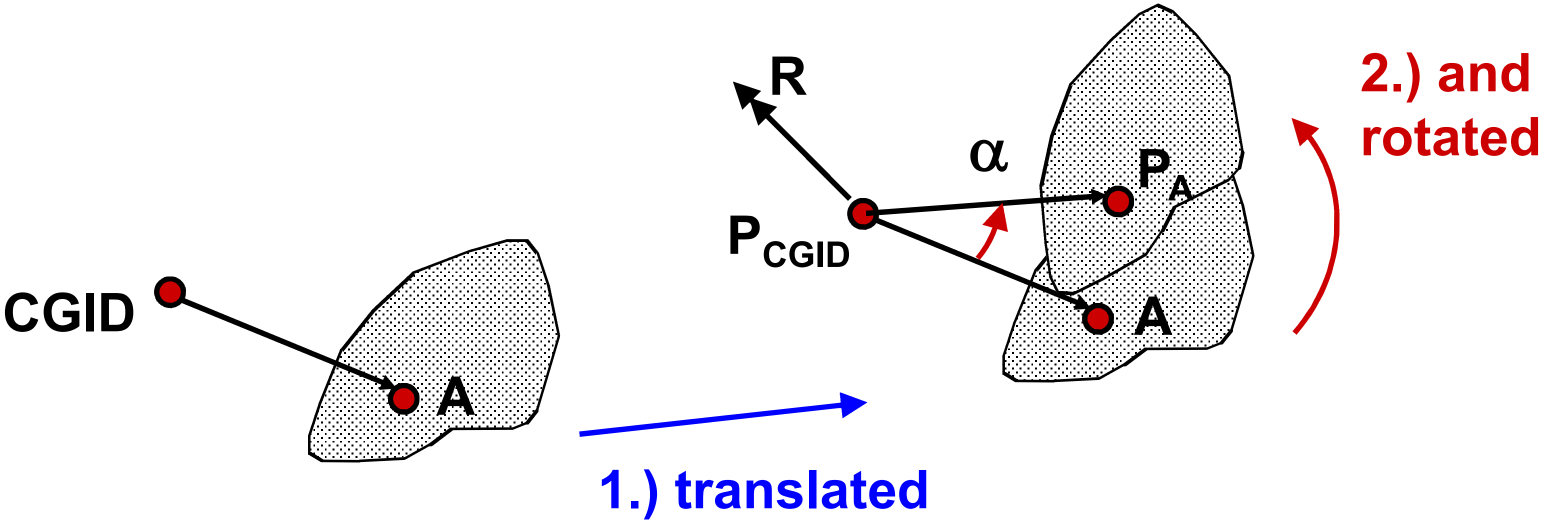
- **For velocity control, on the BCBODY entry define:**
  - CONTROL = 0
  - ANGVEL, angular velocity w about the local axis R through the center of rotation CGID, in radians/time
  - DCOSi, direction cosines of local axis R
  - VELRBi, translational velocity vCGID of the center of rotation CGID in direction i
- **The velocity of point A of the rigid body then reads:**

$$v_A = v_{CGID} + \omega R \times r$$



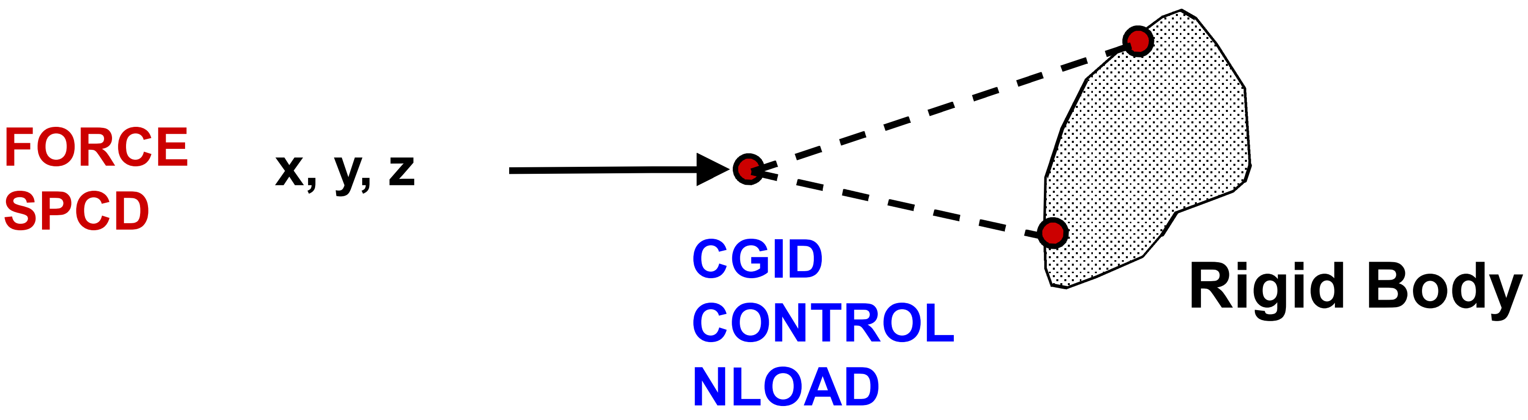
DEFINITION OF RIGID BODIES

- **For position control, on the BCBODY entry define:**
  - CONTROL = -1
  - ANGVEL, angular position a about the local axis R through the center of rotation, in radians
  - DCOSi, direction cosines of local axis R
  - VELRBi, target position PCGID of the center of rotation in direction i

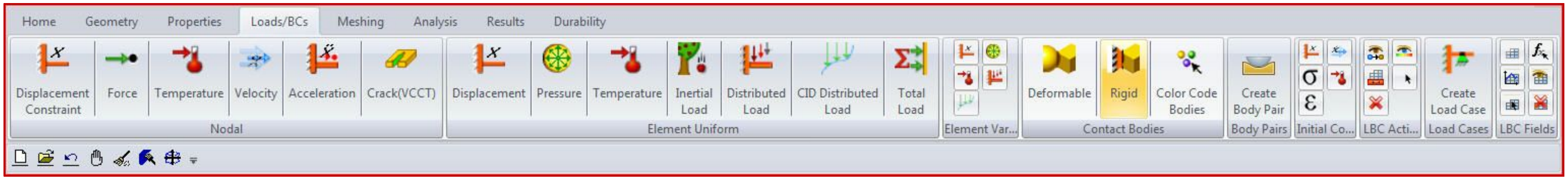


DEFINITION OF RIGID BODIES

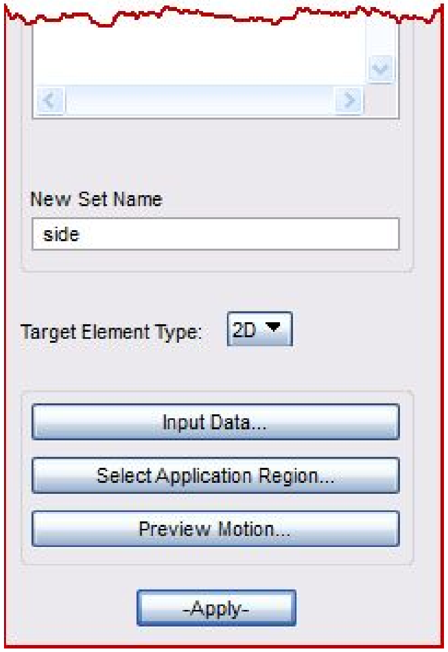
- **For load control, on BCBODY card define:**
  - CONTROL = ID of the grid point to which translational loads or SPCDs are applied and at which the translations of CGID are reported
  - NLOAD = ID of the grid point to which the rotational loads or SPCDs are applied and at which the rotations of CGID are reported



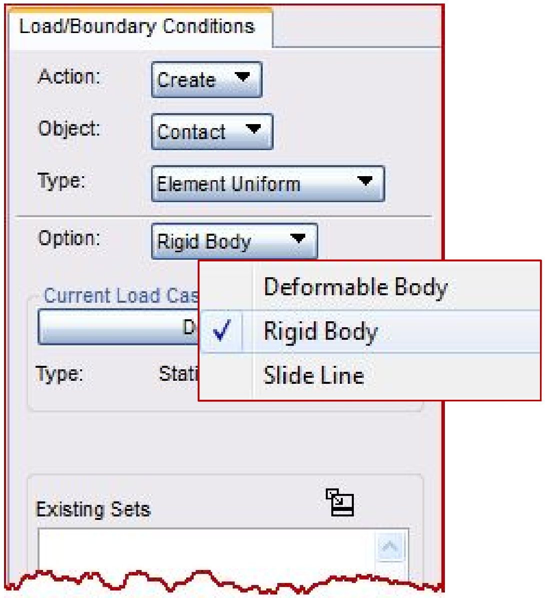
CREATING A RIGID BODY



- **Under the Patran Loads/BCs tab:**
  - Click Rigid
  - Enter New Set Name
  - Pick Target Element Type


New Set Name: side

Target Element Type: 2D



Action: Create

Object: Contact

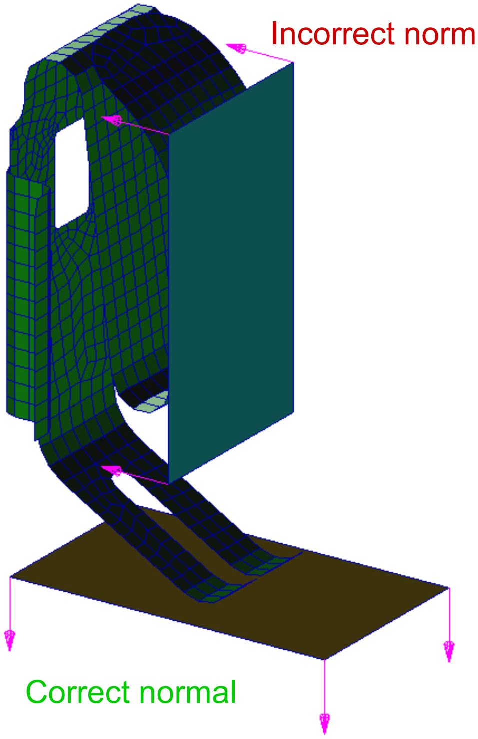
Type: Element Uniform

Option: Rigid Body

CREATING A RIGID BODY

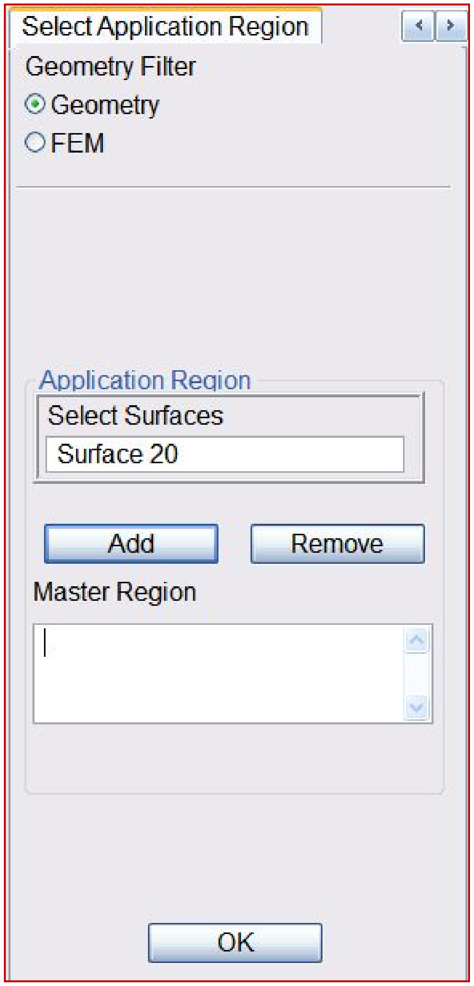
- **On Select Application Region form:**
  - Select Geometry directly (to create NURBs) or
  - Based on FEM (to create PATCH3D)

The rigid body normal should point away from the deformable body to be contacted.



Incorrect normal

Correct normal



Select Application Region

Geometry Filter

● Geometry

○ FEM

Application Region

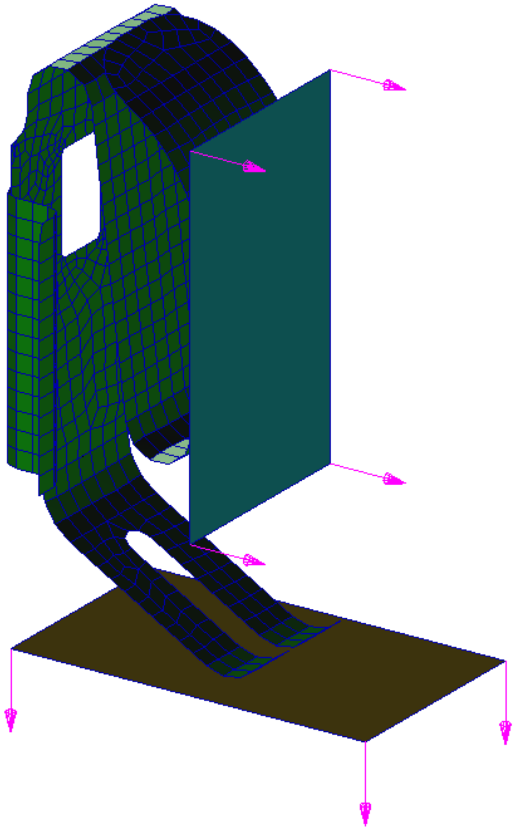
Select Surfaces

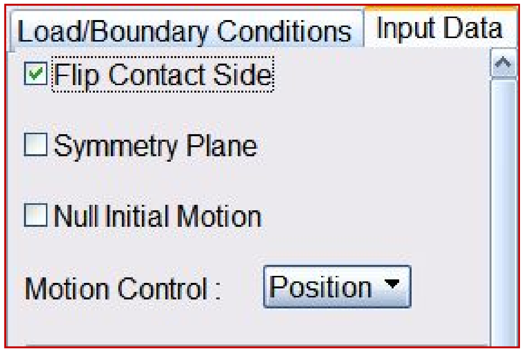
Surface 20

Master Region

CREATING A RIGID BODY

- **To change rigid body normal, check Flip Contact Side on Input Data form.**





☑ Flip Contact Side

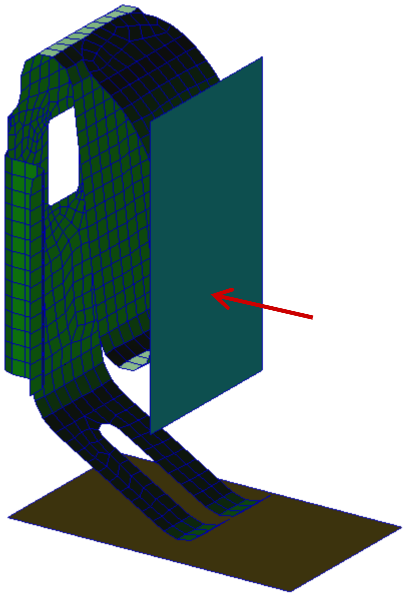
☐ Symmetry Plane

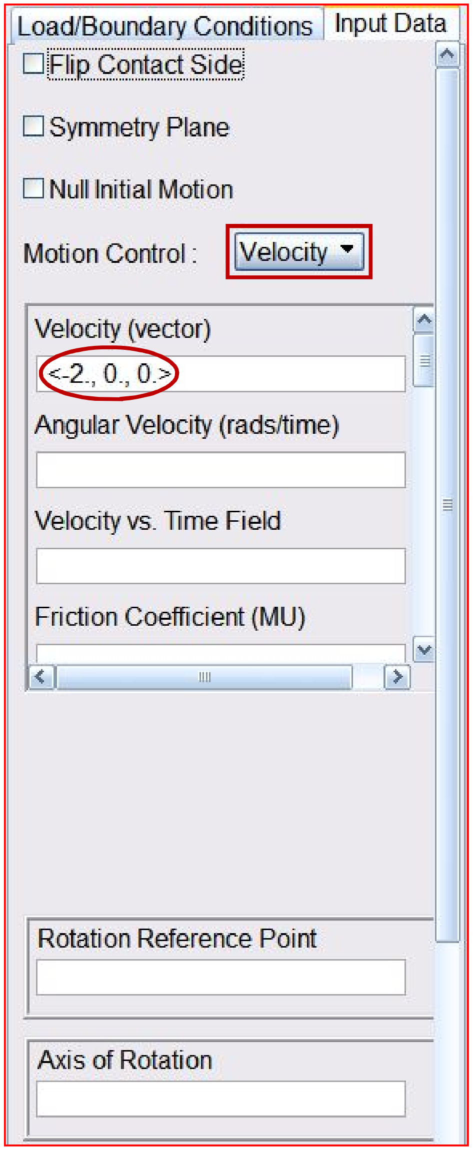
☐ Null Initial Motion

Motion Control : Position

CREATING A RIGID BODY

- **Setup velocity controlled rigid body motion**
  - Select Velocity motion control on Input Data form
  - Specify rigid body translational velocity vector
  - If the rigid body rotates about a local axis, then specify
    - Angular velocity
    - Rotation reference point (center of rotation)
    - Axis of rotation





☐ Flip Contact Side

☐ Symmetry Plane

☐ Null Initial Motion

Motion Control : Velocity

Velocity (vector)
<-2., 0., 0.>

Angular Velocity (rads/time)

Velocity vs. Time Field

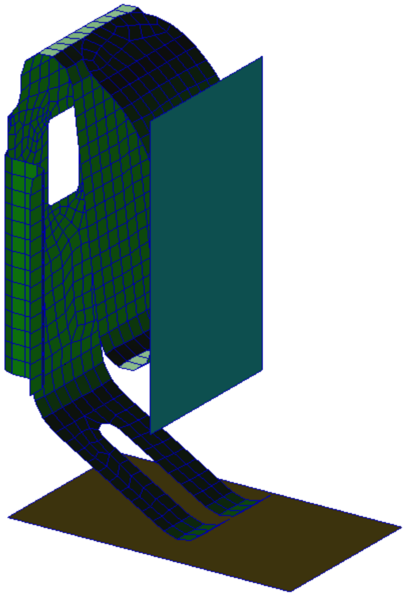
Friction Coefficient (MU)

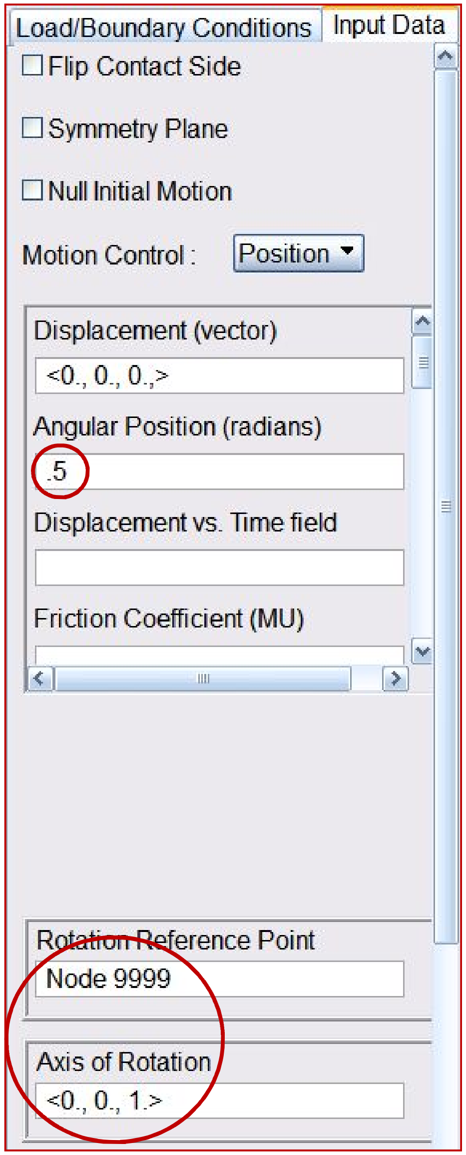
Rotation Reference Point

Axis of Rotation

CREATING A RIGID BODY

- **Setup position controlled rigid body motion**
  - Select Position motion control on Input Data form
  - Specify rigid body translational displacement vector
  - If the rigid body rotates about a local axis, then specify
    - Angular position
    - Rotation reference point (center of rotation)
    - Axis of rotation





☐ Flip Contact Side

☐ Symmetry Plane

☐ Null Initial Motion

Motion Control : Position

Displacement (vector): <0., 0., 0.,>

Angular Position (radians): .5

Displacement vs. Time field:

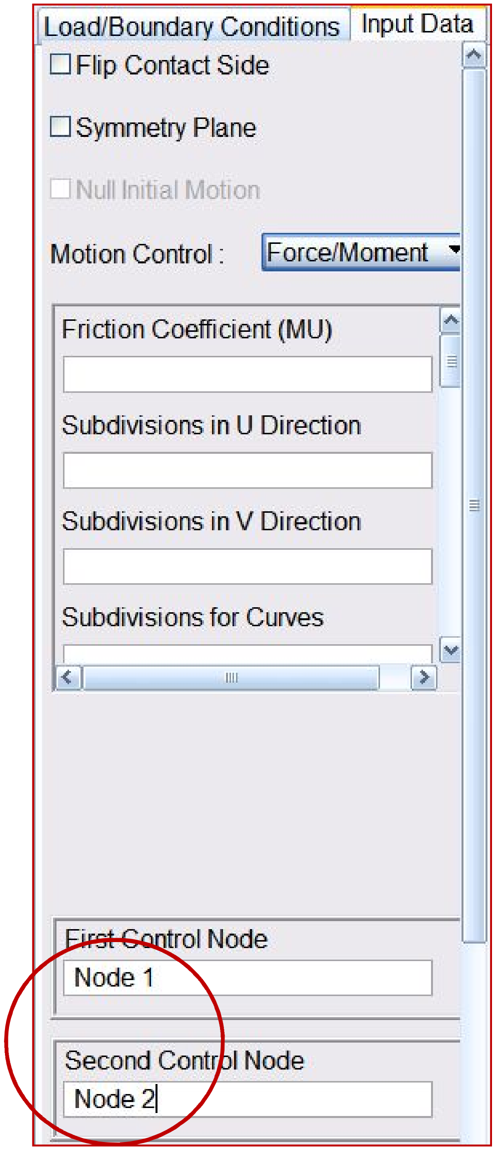
Friction Coefficient (MU):

Rotation Reference Point: Node 9999

Axis of Rotation: <0., 0., 1.>

CREATING A RIGID BODY

- **Setup load controlled rigid body motion**
  - Select Force/Moment motion control on Input Data form
  - Specify first control node (center of rotation), this node represents translational DOFs of the rigid body
  - Specify second control node (location is irrelevant), this node represents rotational DOFs of the rigid body
  - Apply force or translational displacement to the first control node
  - Apply moment or rotational displacement to the second control node
  - If no load or SPCD applied to the control nodes, the rigid body is free to move



☐ Flip Contact Side

☐ Symmetry Plane

☐ Null Initial Motion

Motion Control : Force/Moment

Friction Coefficient (MU)

Subdivisions in U Direction

Subdivisions in V Direction

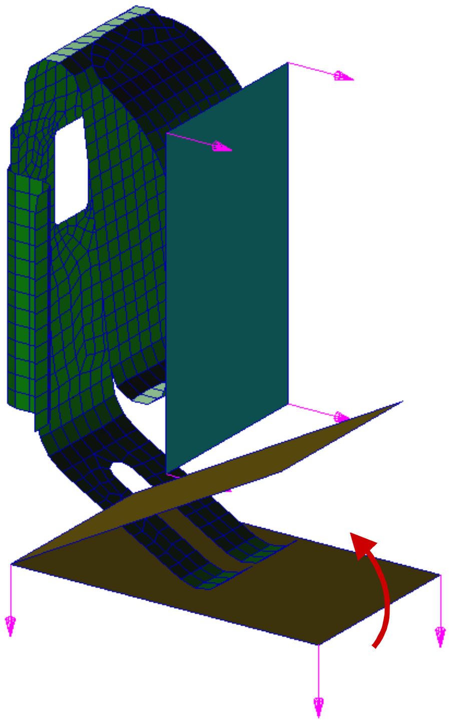
Subdivisions for Curves

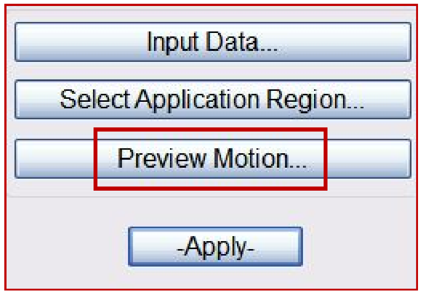
First Control Node: Node 1

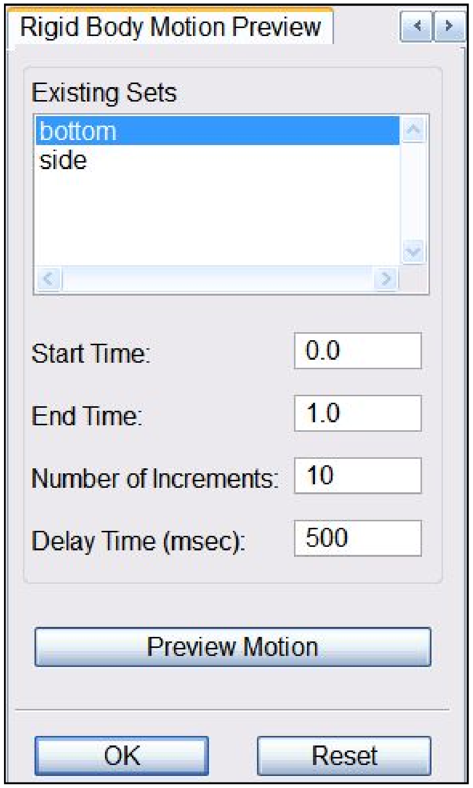
Second Control Node: Node 2

CREATING A RIGID BODY

- **Preview rigid body motion**
  - Available only for velocity and position control
  - On Preview Motion form
    - Select a rigid body from existing sets
    - Click Preview Motion







Start Time: 0.0

End Time: 1.0

Number of Increments: 10

Delay Time (msec): 500

CREATING A RIGID BODY

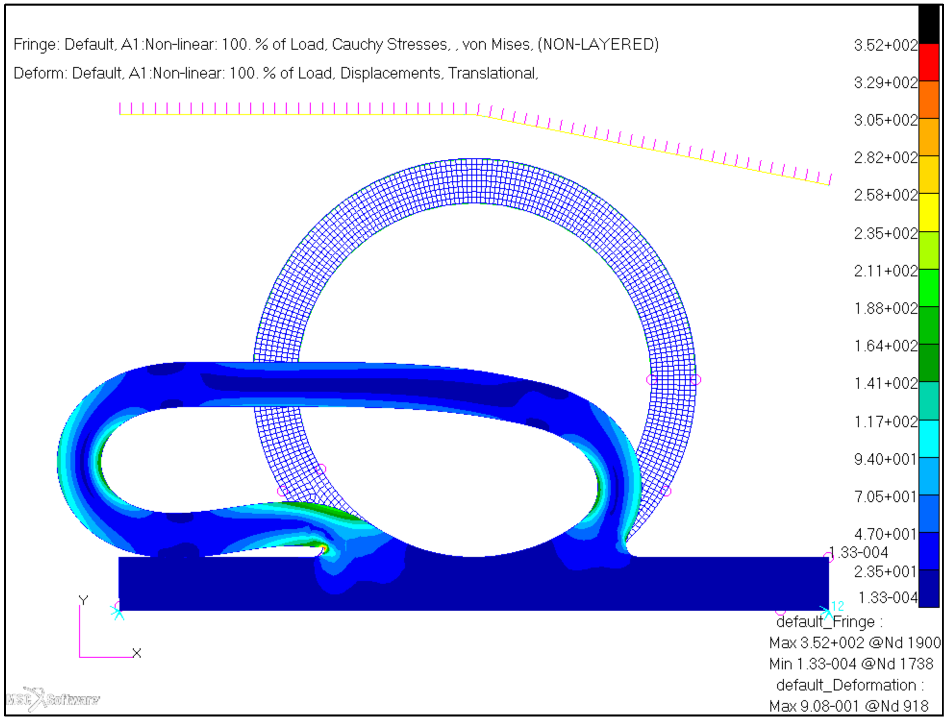
- **Sample MSC Nastran input file:**

Don’t try this at home!!

```
$ Rigid Body Contact LBC set: side
BCBODY  3       3D      RIGID           0               1       0
        0       0.      0.      0.      0.      -2.     0.      0.
        RIGID   0       1       SIDE
        NURBS   -2      2       2       2       50      50      4
                5.1016  5.3644-8-2.9    5.1016  9.9     -2.9
                5.1016  5.3644-82.9     5.1016  9.9     2.9
```

WORKSHOP 2 – RUBBER DOOR SEAL

- **Workshop Objectives**
  - Large displacement/ large strain analysis
  - Contact analysis using rigid-deformable contact
  - Hyperelastic material model



OVERVIEW

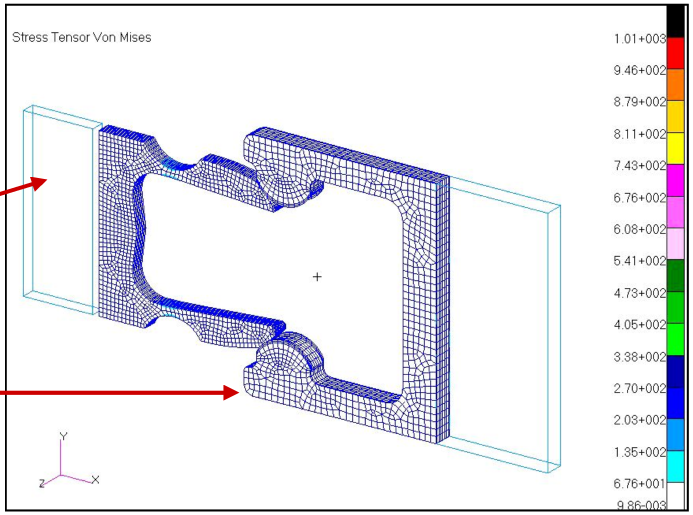
- What is and when to consider contact
- Contact Bodies (Workshop – Rubber Door Seal)
- **Contact Pairs/Tables** (Workshop – Deformable to Rigid Contact)
- Contact Detection
  - Node to segment
  - Segment to segment
- Special Features
  - Interference (Workshop – Interference Fit)
  - Stress free initial contact
  - Glued contact (Workshop – Contact Pairs)
  - Contact with Shells
  - Friction
- Convergence and Controls with contact

CONTACT PAIRS/TABLES

- **Patran/MSC Nastran allows 2 methods to define which contact bodies are to be considered for contact with which other bodies**
  - ‘Contact Table’
    - As the name implies, this method allows the user to populate a ‘table’ which defines which bodies contact which other bodies
    - Easy to use
    - Only practical for relatively small (typically <10) numbers of contact bodies
  - ‘Contact Pair’
    - Explicit definition of which bodies are to touch which other bodies
    - Allows ease of use when many (>10) contact bodies exist
- **Furthermore, each method allows for specification of the contact type as**
  - Touching – allows intermittent contact
  - Glued – enforces a permanent connection between contact bodies
- **Both methods define the same characteristics/properties, the choice of which method is be used comes down to which is ‘easier’, typically determined by the number of contact bodies**

CONTACT PAIR

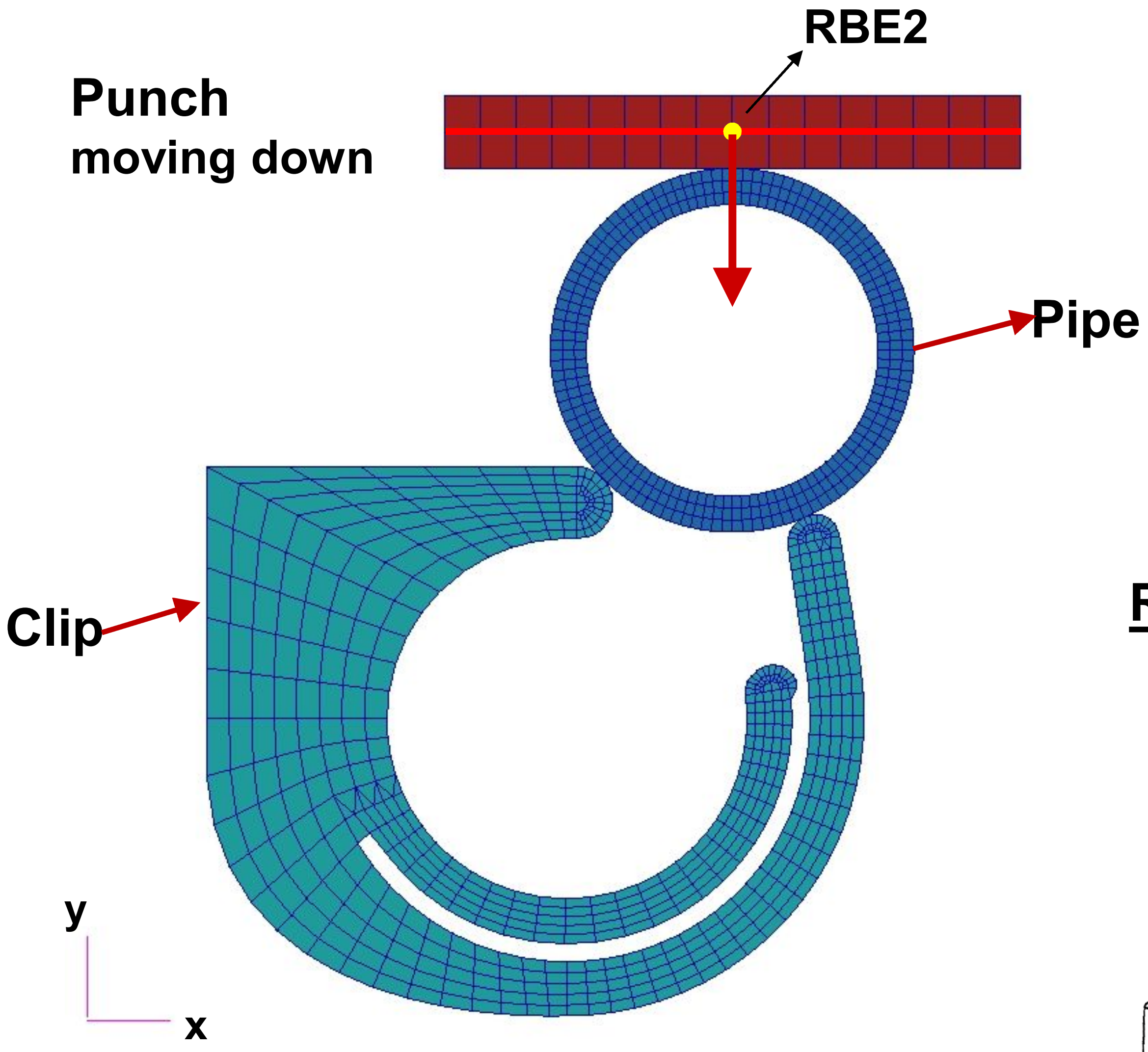
- **Determines which Contact Bodies interact with each other and how**
  - TOUCH or GLUE



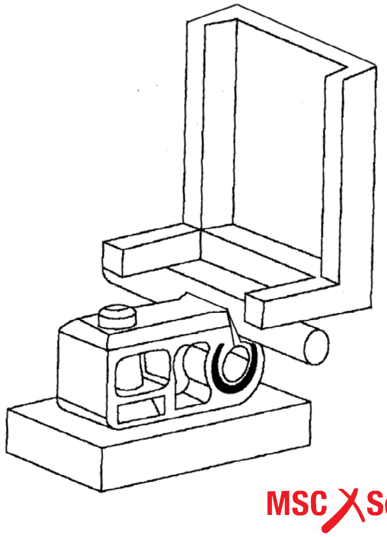
This rigid body does not need to interact with this deformable body

- **Also allows 'tweaking' of contact body parameters on a Contact Body pair basis in the event of convergence or other difficulties**
  - Contact order
  - Distance Tolerance
  - Separation Force

PIPE TO CLIP CONTACT EXAMPLE

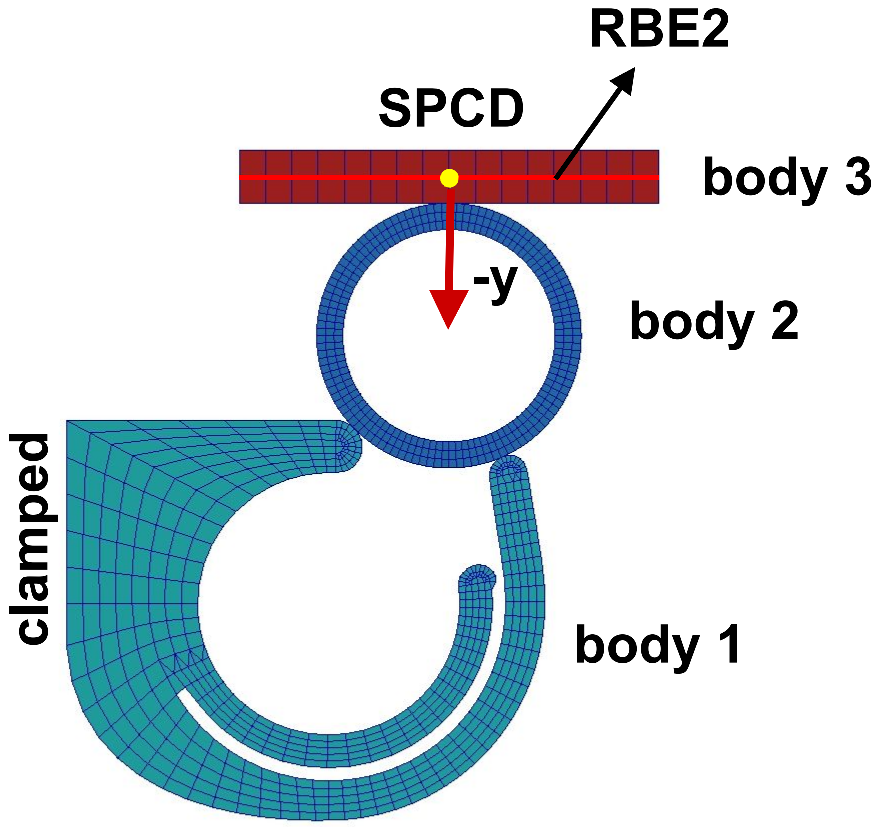


Related Test



MODEL DESCRIPTION

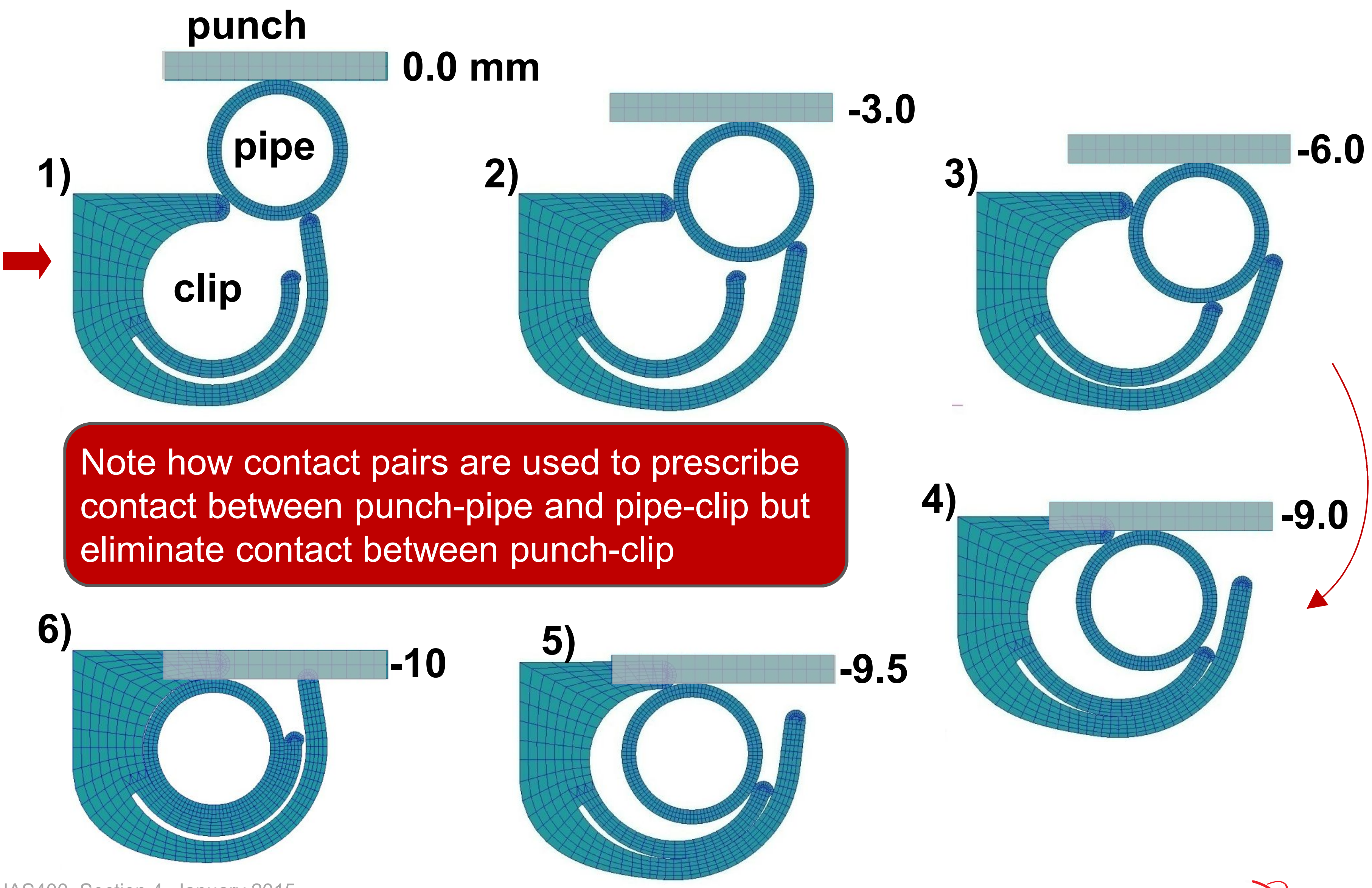
- **Clip:** **plastics, E=2100., n=0.3**
  - 2D solids, inner ∅ = 10 mm
- **Pipe:** **steel, E=2.1E5, n=0.3**
  - 2D solids, outer ∅ = 10.1 mm
- **Punch:** **steel, like pipe**
  - 2D solids
- **Loading:**
  - Punch pushed in
  - Punch is pulled out
- **Boundary Conditions:**
  - Left side of clip is clamped
  - Punch and pipe vertically connected
  - To exclude dynamic effects (snap & eject)



Friction:

body	**1-1, μ=0.25**
bodies	**1-2, μ=0.15**
bodies	**2-3, μ=0**

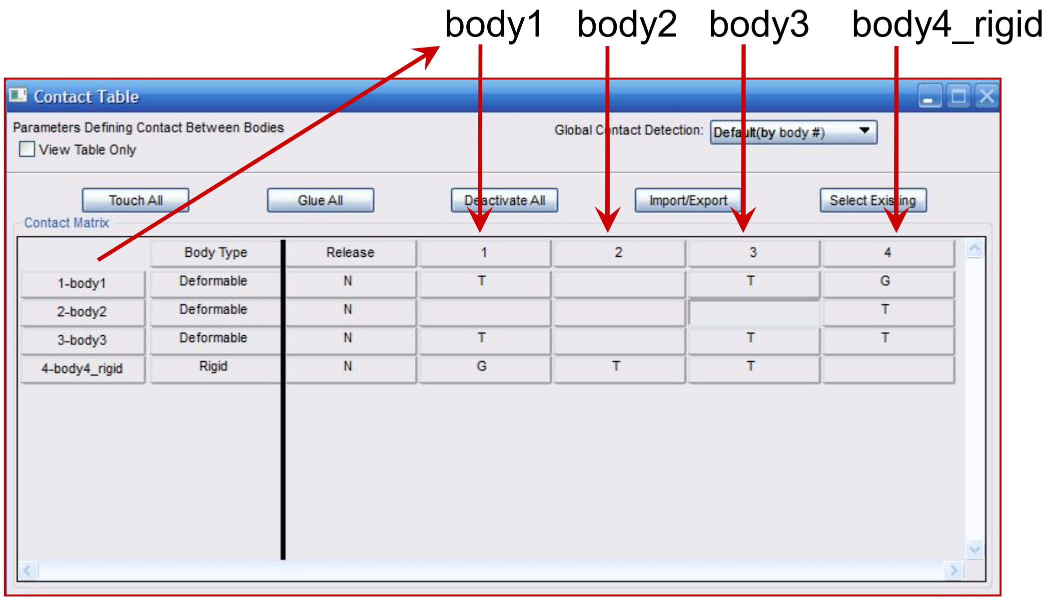
PUSHING DEFORMATIONS



Note how contact pairs are used to prescribe contact between punch-pipe and pipe-clip but eliminate contact between punch-clip

CONTACT TABLES

- **Best described using example:**



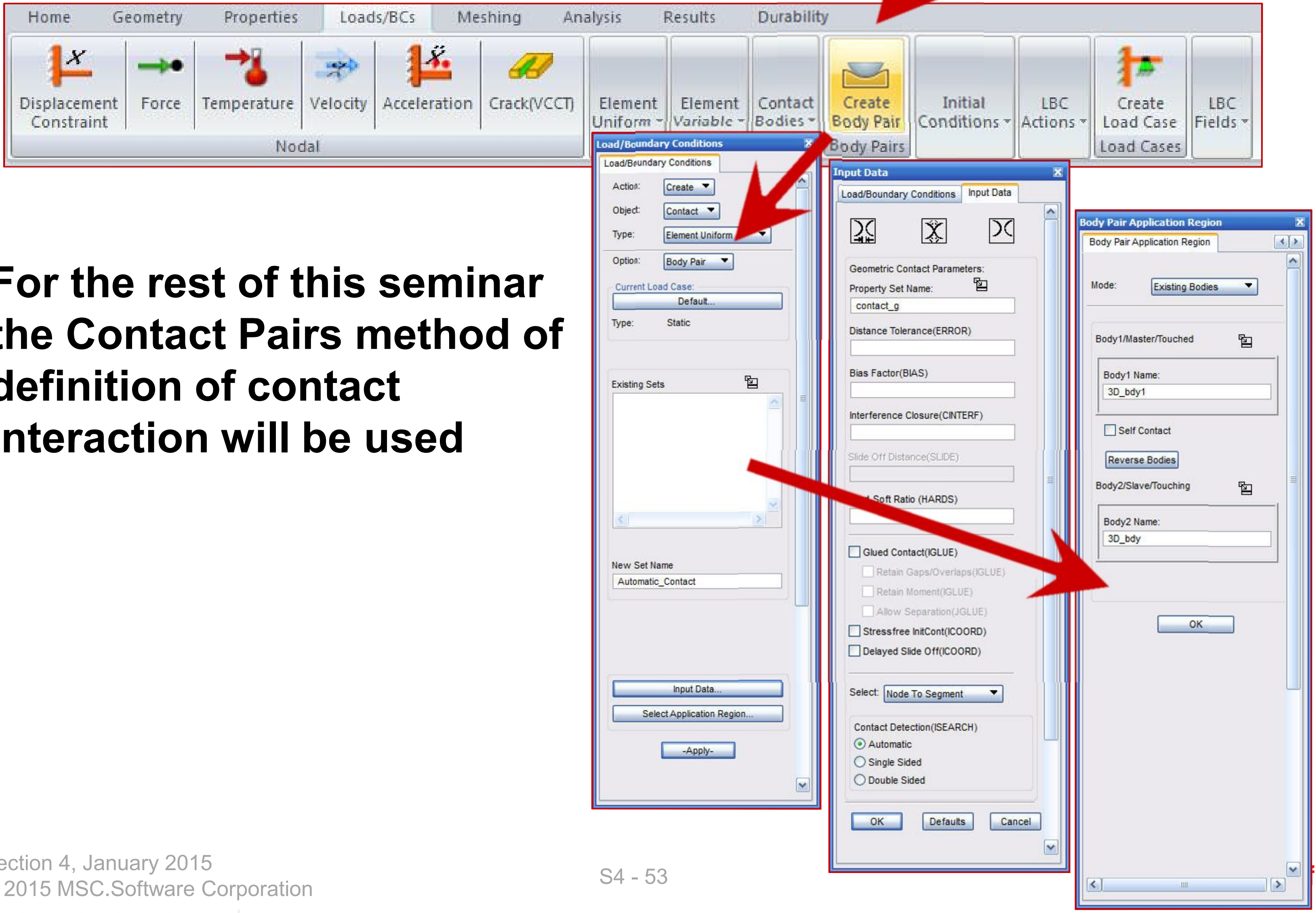
	Body Type	Release	1	2	3	4
1-body1	Deformable	N	T		T	G
2-body2	Deformable	N				T
3-body3	Deformable	N	T		T	T
4-body4_rigid	Rigid	N	G	T	T	

- **In this example:**
  - ‘body1’ is only allowed to ‘touch’ itself and ‘body3’
  - ‘body1’ is ‘glued’ to ‘body4_rigid’
  - ‘body2’ is only allowed to’ touch’ ‘body4_rigid’
  - ‘body3’ is only allowed to ‘touch’ itself and ‘body4_rigid’

CONTACT PAIR

- **Best described using example:**

- **For the rest of this seminar the Contact Pairs method of definition of contact interaction will be used**



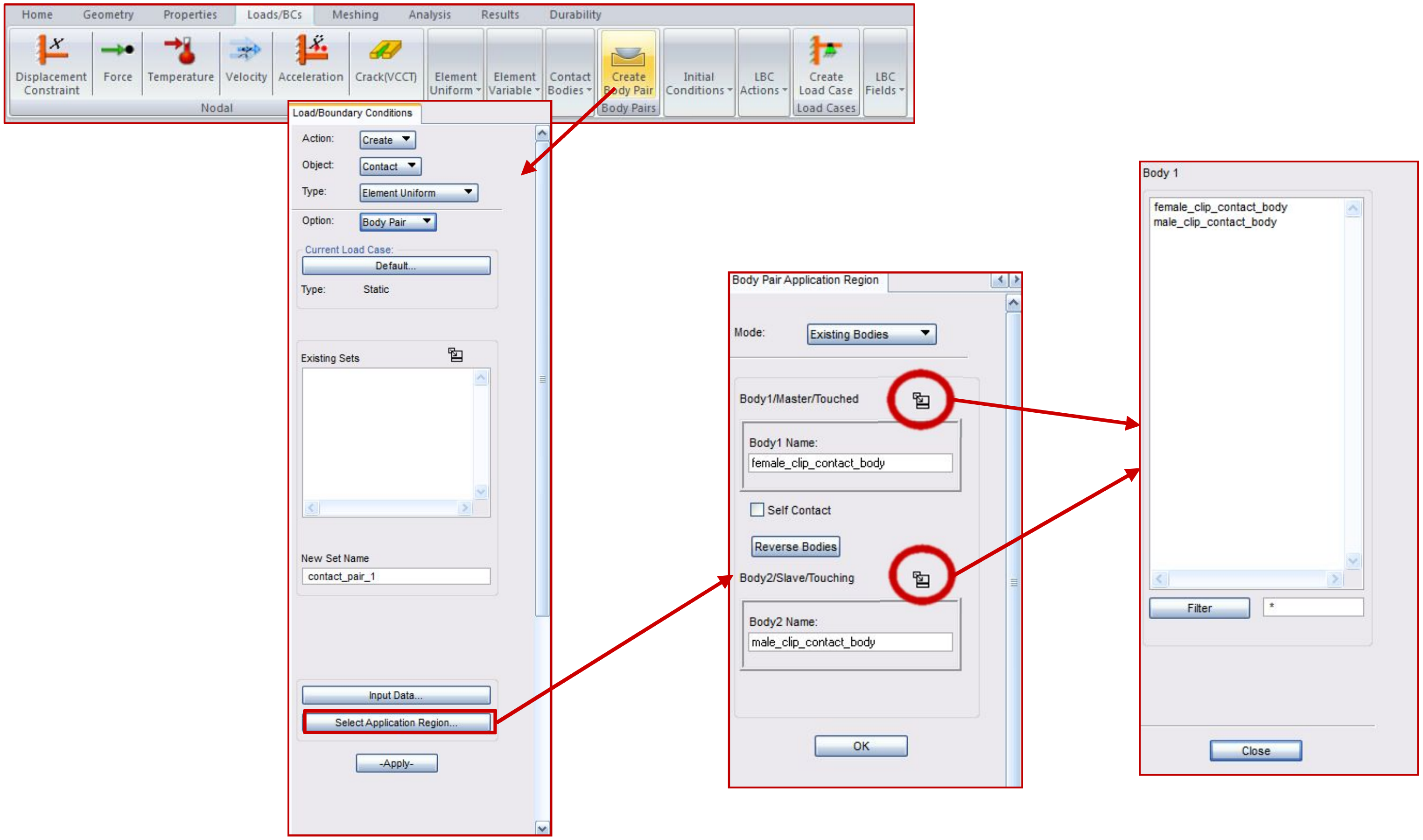
Action: Create
Object: Contact
Type: Element Uniform
Option: Body Pair
Current Load Case: Default...
Type: Static
New Set Name: Automatic_Contact

Property Set Name: contact_g
Distance Tolerance(ERROR)
Bias Factor(BIAS)
Interference Closure(CINTERF)
Slide Off Distance(SLIDE)
Soft Ratio (HARDS)
☐ Glued Contact(IGLUE)
☐ Retain Gaps/Overlaps(IGLUE)
☐ Retain Moment(IGLUE)
☐ Allow Separation(JGLUE)
☐ Stressfree InitCont(ICOORD)
☐ Delayed Slide Off(ICOORD)
Select: Node To Segment
Contact Detection(ISEARCH)
● Automatic
○ Single Sided
○ Double Sided

Mode: Existing Bodies
Body1 Name: 3D_bdy1
☐ Self Contact
Body2 Name: 3D_bdy

CONTACT PAIR

- **A ‘CONTACT PAIR’ is defined like other Loads/BCs and defines which bodies are to interact.**



Action: Create
Object: Contact
Type: Element Uniform
Option: Body Pair
Current Load Case: Default...
Type: Static
New Set Name: contact_pair_1

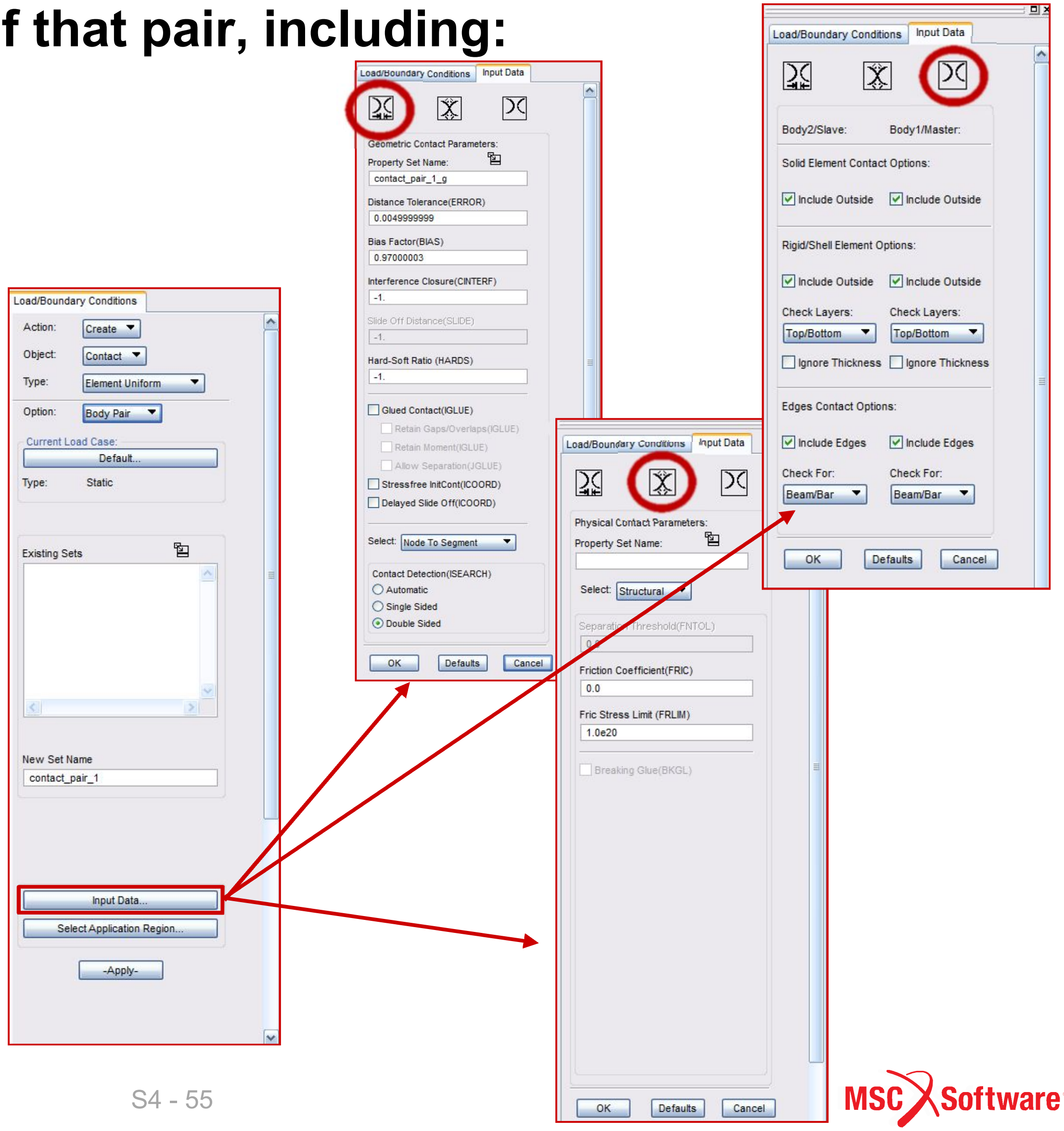
Mode: Existing Bodies
Body1 Name: female_clip_contact_body
Body2 Name: male_clip_contact_body

Body 1:
female_clip_contact_body
male_clip_contact_body
Filter: *

- **Like other Loads/BCs the contact pair can active/inactive as desired by the user in each loadcase**

CONTACT PAIR

- **Each CONTACT PAIR can have a unique set of parameters that will affect the behavior of that pair, including:**
  - Distance tolerance
  - Bias factor
  - Separation force
  - Glue
  - Stress free initial contact
  - Order behavior
- **3 sub-forms are available via the ‘Input Data’ button to define the various parameters.**
- **Like other Loads/BCs the contact pair can be made active/inactive as desired by the user in each loadcase**



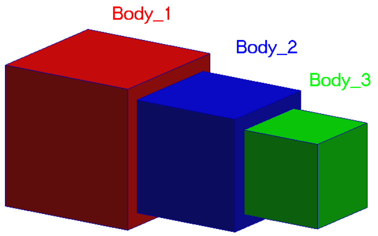
Action: Create
Object: Contact
Type: Element Uniform
Option: Body Pair
Current Load Case: Default...
Type: Static
Existing Sets
New Set Name: contact_pair_1

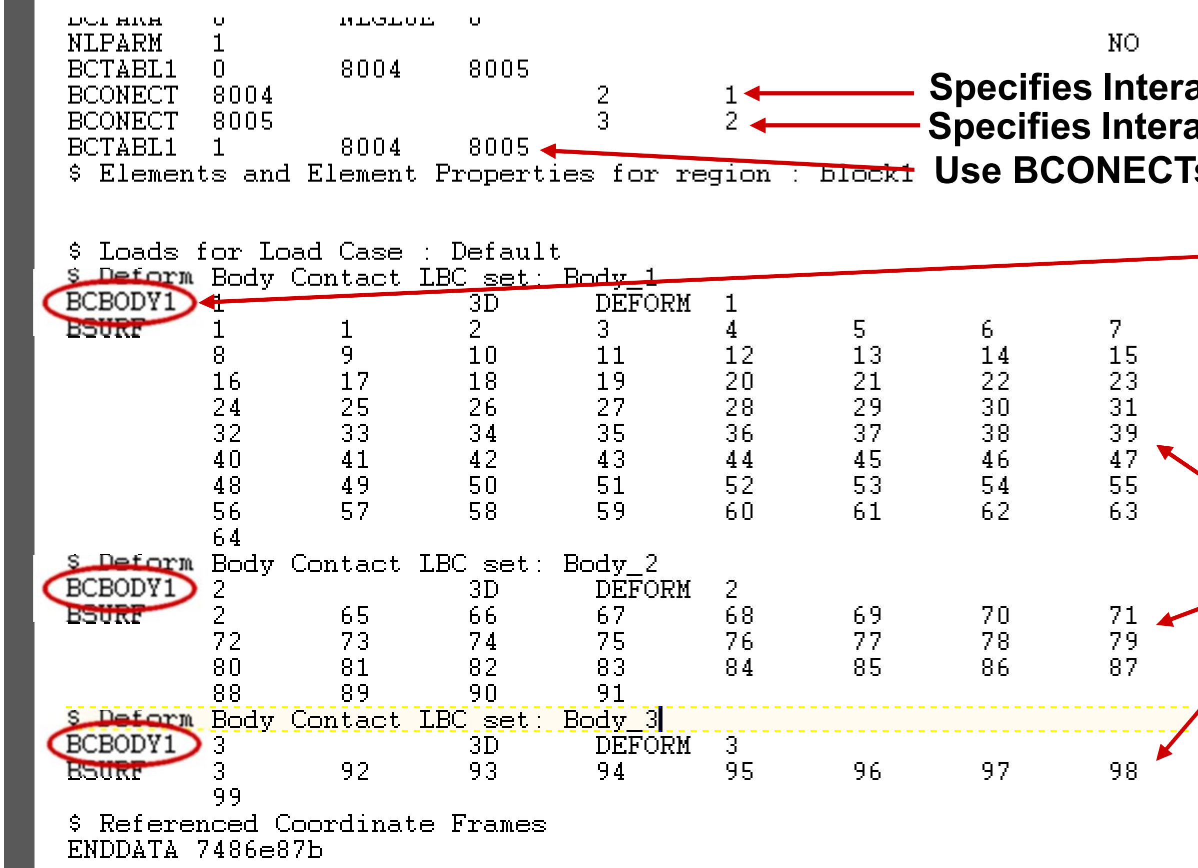
Geometric Contact Parameters:
Property Set Name: contact_pair_1_g
Distance Tolerance(ERROR): 0.0049999999
Bias Factor(BIAS): 0.97000003
Interference Closure(CINTERF): -1.
Slide Off Distance(SLIDE): -1.
Hard-Soft Ratio (HARDS): -1.
☐ Glued Contact(IGLUE)
☐ Retain Gaps/Overlaps(IGLUE)
☐ Retain Moment(IGLUE)
☐ Allow Separation(JGLUE)
☐ Stressfree InitCont(ICOORD)
☐ Delayed Slide Off(ICOORD)
Select: Node To Segment
Contact Detection(ISEARCH)
○ Automatic
○ Single Sided
● Double Sided

Physical Contact Parameters:
Property Set Name:
Select: Structural
Separation Threshold(FNTOL): 0.0
Friction Coefficient(FRIC): 0.0
Fric Stress Limit (FRLIM): 1.0e20
☐ Breaking Glue(BKGL)

Body2/Slave: | Body1/Master:
Solid Element Contact Options:
☑ Include Outside | ☑ Include Outside
Rigid/Shell Element Options:
☑ Include Outside | ☑ Include Outside
Check Layers: Top/Bottom | Check Layers: Top/Bottom
☐ Ignore Thickness | ☐ Ignore Thickness
Edges Contact Options:
☑ Include Edges | ☑ Include Edges
Check For: Beam/Bar | Check For: Beam/Bar

CONTACT PAIRS WRITTEN FROM PATRAN





```
BCPARA  0       NLGLUE  0
NLPARM  1                                                       NO
BCTABL1 0       8004    8005
BCONECT 8004                            2       1
BCONECT 8005                            3       2
BCTABL1 1       8004    8005
$ Elements and Element Properties for region : block1


$ Loads for Load Case : Default
$ Deform Body Contact LBC set: Body_1
BCBODY1 1               3D      DEFORM  1
BSURF   1       1       2       3       4       5       6       7
        8       9       10      11      12      13      14      15
        16      17      18      19      20      21      22      23
        24      25      26      27      28      29      30      31
        32      33      34      35      36      37      38      39
        40      41      42      43      44      45      46      47
        48      49      50      51      52      53      54      55
        56      57      58      59      60      61      62      63
        64
$ Deform Body Contact LBC set: Body_2
BCBODY1 2               3D      DEFORM  2
BSURF   2       65      66      67      68      69      70      71
        72      73      74      75      76      77      78      79
        80      81      82      83      84      85      86      87
        88      89      90      91
$ Deform Body Contact LBC set: Body_3
BCBODY1 3               3D      DEFORM  3
BSURF   3       92      93      94      95      96      97      98
        99
$ Referenced Coordinate Frames
ENDDATA 7486e87b
```

Specifies Interaction between Contact Body 1 and 2

Specifies Interaction between Contact Body 2 and 3

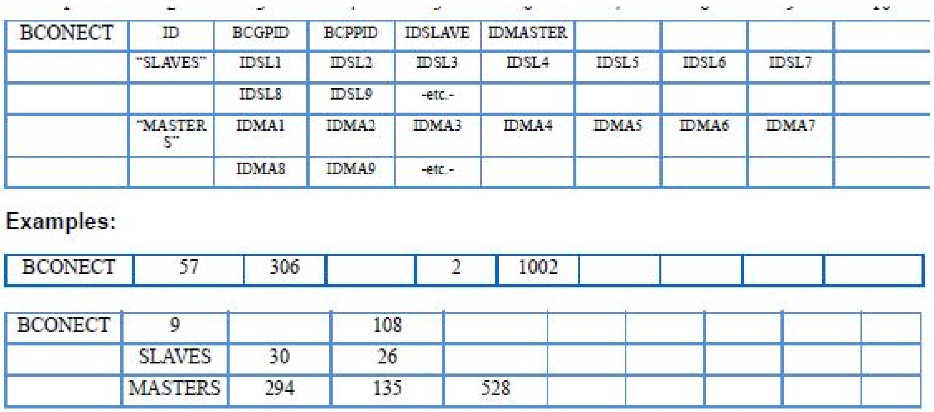
Use BCONECTs 8004 and 8005

Note the use of BCBODY1 entry (not BCBODY), required when using contact pairs

Contact Body Definition

CONTACT PAIRS IN MSC NASTRAN

- **BCONECT – MSC Nastran entry to define which contact bodies interact with which other contact body**
- **In its simplest form this will be a single line specifying 2 bodies**
- **Lists of bodies may also be defined if desired**



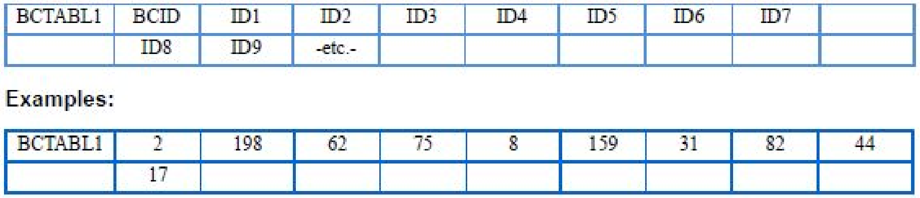
BCONECT	ID	BCGPID	BCPPID	IDSLAVE	IDMASTER				
	"SLAVES"	IDSL1	IDSL2	IDSL3	IDSL4	IDSL5	IDSL6	IDSL7	
		IDSL8	IDSL9	-etc.-					
	"MASTERS"	IDMA1	IDMA2	IDMA3	IDMA4	IDMA5	IDMA6	IDMA7	
		IDMA8	IDMA9	-etc.-					

Examples:

BCONECT	57	306		2	1002				

BCONECT	9		108						
	SLAVES	30	26						
	MASTERS	294	135	528					

- **BCTABL1 – specifies which BCONECTs are to be a simultaneously and is referenced at the Case Control level by BCONTACT:**



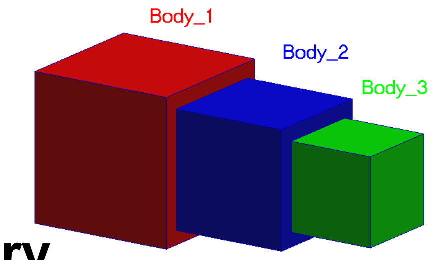
BCTABL1	BCID	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

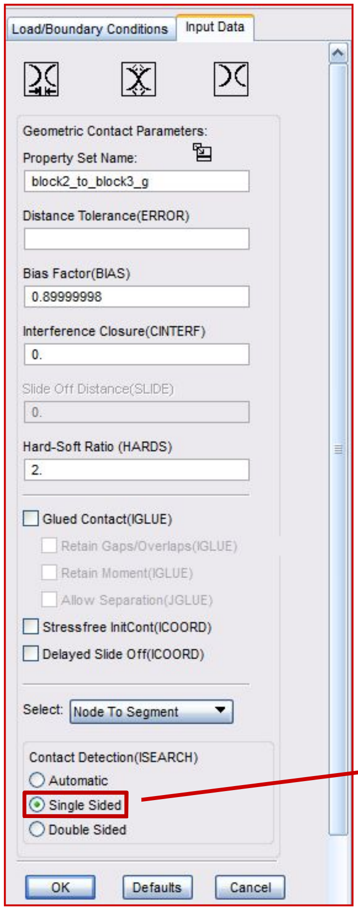
Examples:

BCTABL1	2	198	62	75	8	159	31	82	44
	17								

CONTACT PAIRS IN MSC NASTRAN

- **Example – bodies 1, 2 and 3 in a sequence**
- **User specified ‘single sided’ search sets appropriate value in MSC Nastran BCONPRG entry**





Geometric Contact Parameters:

Property Set Name: block2_to_block3_g

Distance Tolerance(ERROR):

Bias Factor(BIAS): 0.89999998

Interference Closure(CINTERF): 0.

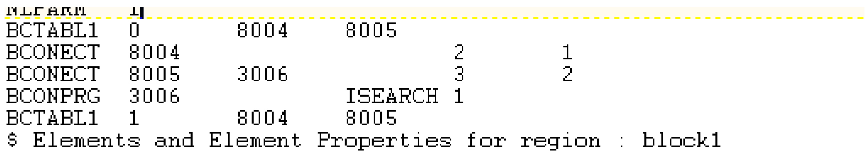
Slide Off Distance(SLIDE): 0.

Hard-Soft Ratio (HARDS): 2.

☐ Glued Contact(IGLUE)
  ☐ Retain Gaps/Overlaps(IGLUE)
  ☐ Retain Moment(IGLUE)
  ☐ Allow Separation(JGLUE)
☐ Stressfree InitCont(ICOORD)
☐ Delayed Slide Off(ICOORD)

Select: Node To Segment

Contact Detection(ISEARCH)
○ Automatic
● Single Sided
○ Double Sided



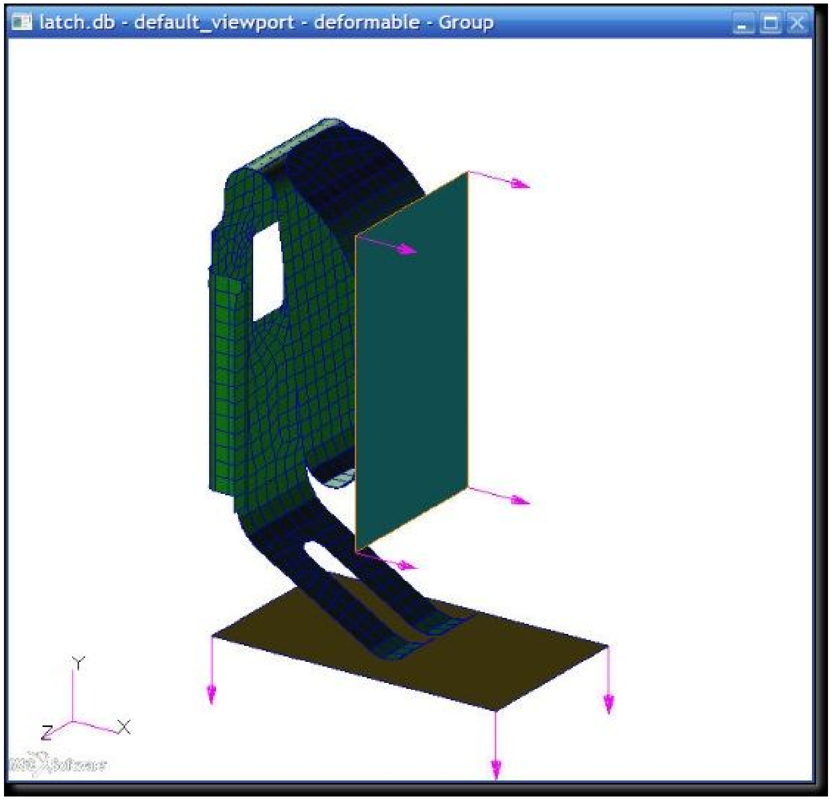
```
NLPARM  1
BCTABL1 0       8004    8005
BCONECT 8004                    2       1
BCONECT 8005    3006            3       2
BCONPRG 3006            ISEARCH 1
BCTABL1 1       8004    8005
$ Elements and Element Properties for region : block1
```

BCONTACT = ALLBODY

- **With this Case Control command, the definition of contact pairs is eliminated**
  - All bodies can potentially contact each other
- **Care should be exercised**
  - Good for checking out runs
  - Convergence can be difficult
  - Run time can be longer

WORKSHOP 3 – DEFORMABLE TO RIGID CONTACT

- Perform a start to finish contact analysis with MSC Nastran that uses both deformable and rigid contact bodies
- Gain experience in the definition of the motion of rigid contact bodies
- Gain familiarity with the Patran contact analysis GUI



OVERVIEW

- What is and when to consider contact
- Contact Bodies (Workshop – Rubber Door Seal)
- Contact Pairs/Tables (Workshop – Deformable to Rigid Contact)
- **Contact Detection**
  - Node to segment
  - Segment to segment
- Special Features
  - Interference (Workshop – Interference Fit)
  - Stress free initial contact
  - Glued contact (Workshop – Contact Pairs)
  - Contact with Shells
  - Friction
- Convergence and Controls with contact

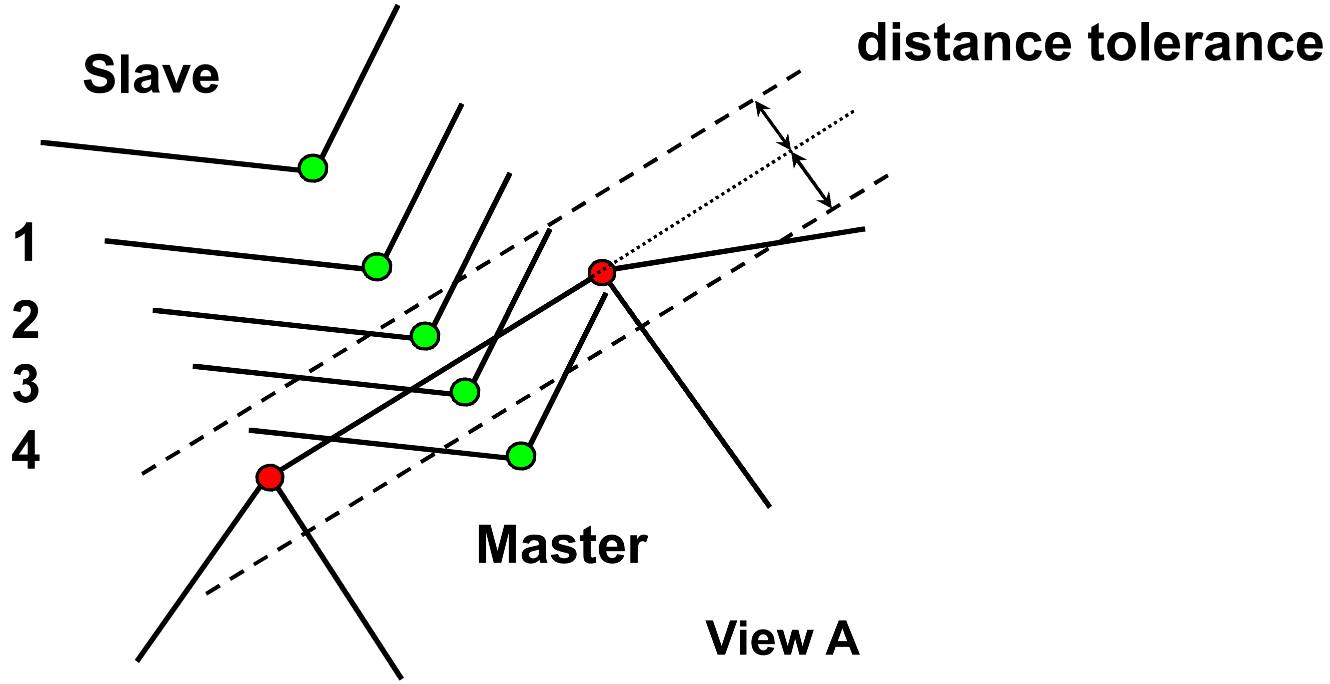
CONTACT DETECTION

- **Term used to define how Nastran determines whether 2 contact bodies are interacting**
- **2 types of contact detection are available in Nastran**
  - Node to Segment
  - Segment to Segment
- **Each has its own advantages, discussed in the following sections**

NODE TO SEGMENT CONTACT

- This type of contact had been developed and tested for many years and is a proven technique.
- However, it has some drawbacks. Segment to Segment contact was recently developed to overcome these drawbacks (discussed later).
- For Node to Segment contact a node can come into contact with a segment.
- For Segment to Segment contact a segment can contact another segment (discussed later).

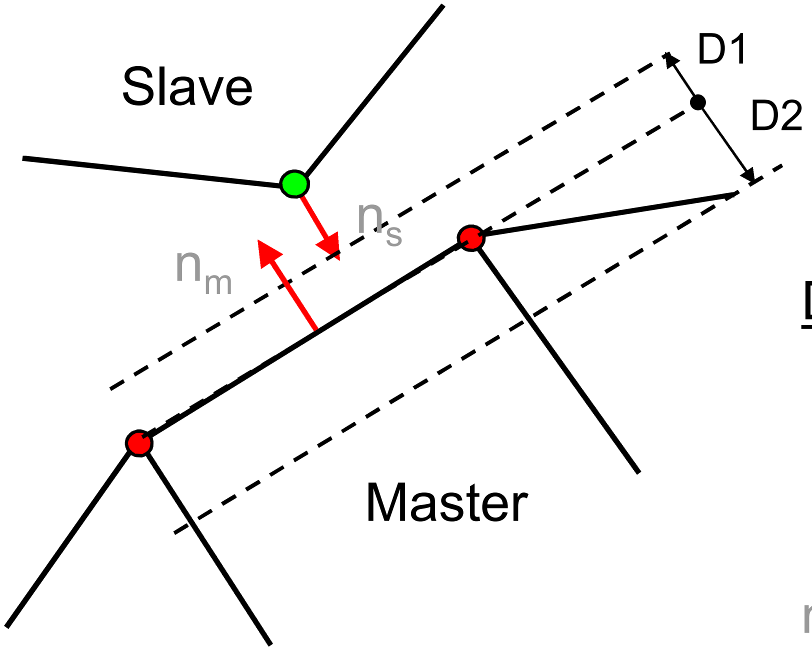
POSSIBLE CONTACT SCENARIOS



1) Node outside element patch, outside distance tolerance
2) Node outside element patch, inside distance tolerance
3) Node inside element patch, inside distance tolerance
4) Node inside element patch, outside distance tolerance

CONTACT DETECTION – SCENARIO 1

- **Slave grid <u>outside</u> master and <u>outside</u> D1**
  - bodies are not in contact
  - nothing has to be done, most simple case



<u>Distance Tolerances</u>

D1=(1 – BIAS)xERROR

D2=(1 + BIAS)xERROR

n – contact normals

DISTANCE TOLERANCES

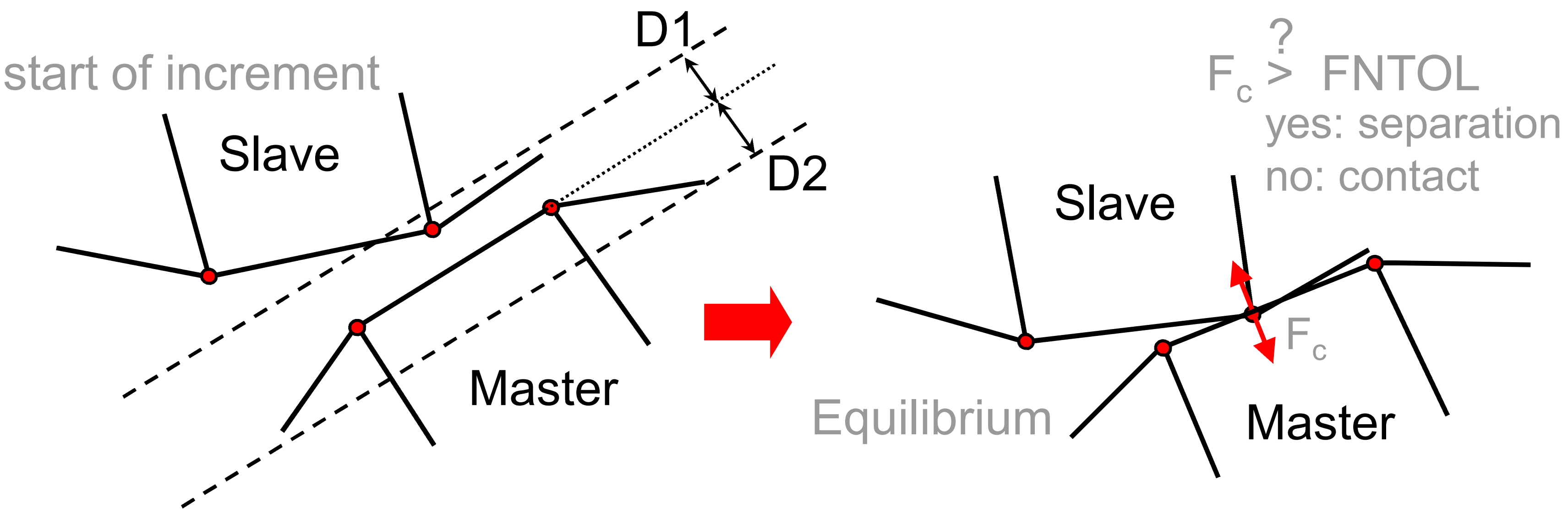
- **Input on BCPARA,0 or BCONPRG (discussed later)**

BCPARA	0	ERROR	0.25	BIAS	0.9	ERRBAS	1		

- **By default, ERROR is evaluated from:**
  - $^{1}/_{20}$ x “smallest element edge” for continuum elements
  - $^{1}/_{4}$ x “smallest thickness” for beam and shell elements
  - Measured globally or pair wise (ERRBAS=0/1)
- **BIAS**
  - The range is between 0.0 and 1.0
  - Default value is 0.9
  - For glued contact it will be 0.0

CONTACT DETECTION – SCENARIO 2

- **Slave grid outside master but inside D1**
  - Only at the start of an increment slave is considered for contact. Apply contact constraint and iterate until the solution converges (equilibrium).
  - Then check the contact force. In case of separation continue iterating. Otherwise go to the next increment.



SEPARATION CHECK

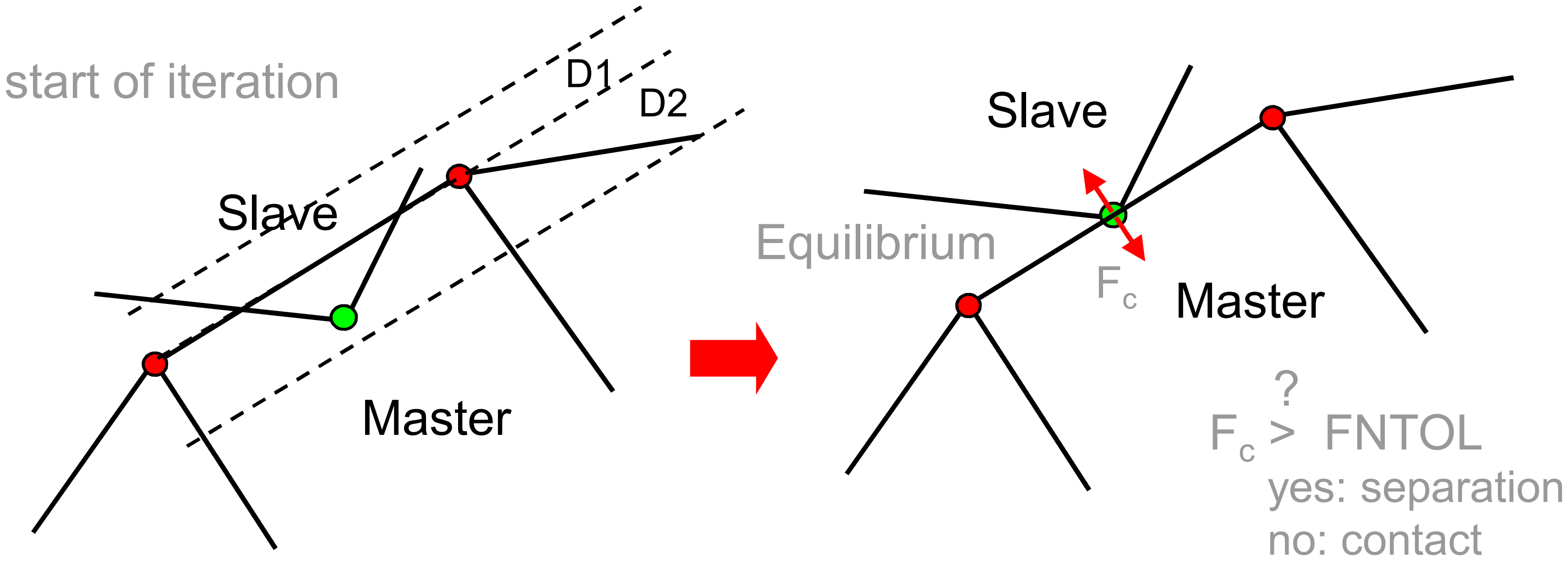
- **BCPARA / IBSEP – separation force flag (default=0)**
  - 0: separation if the contact pulling force exceeds FNTOL (input in BCPARA or BCTABLE. Default: maximum residual force in the complete model).
  - 1: separation if the contact pulling fake stress (pulling force divided by its nodal area) exceeds FNTOL (default: maximum stress at a reaction node in the model times the convergence tolerance).
  - 2: separation if the contact pulling stress (from extrapolating and averaging integration point values) exceeds FNTOL (like 1)
  - 3: separation if the contact pulling fake stress exceeds FNTOL (default=0.1) times the maximum contact stress in the model
  - 4: separation if the contact pulling stress exceeds FNTOL (default=0.1) times the maximum contact stress in the model
  - Notice that for quadratic contact, only options 2 or 4 can be used since the nodal forces do not provide proper values

SEPARATION CONTROL

- **BCPARA / ICSEP – separation flag (default=0)**
  - 0: if the force on a node is greater than the separation force, the node separates and an iteration occurs
  - 1: if a node, which was in contact at the end of the previous increment, has a force greater than the separation force, the node does not separate in this increment, but separates at the beginning of the next increment
  - 2: if a new node comes into contact during this increment, it is not allowed to separate during this increment (prevents chattering)
  - 3: both 1 and 2 are in effect

CONTACT DETECTION – SCENARIO 3

- **Slave grid <u>inside</u> master and <u>inside</u> D2**
  - Apply contact constraint and iterate until the solution converges (equilibrium)
  - Then check the contact force. In case of separation continue iterating. Otherwise go to the next increment.



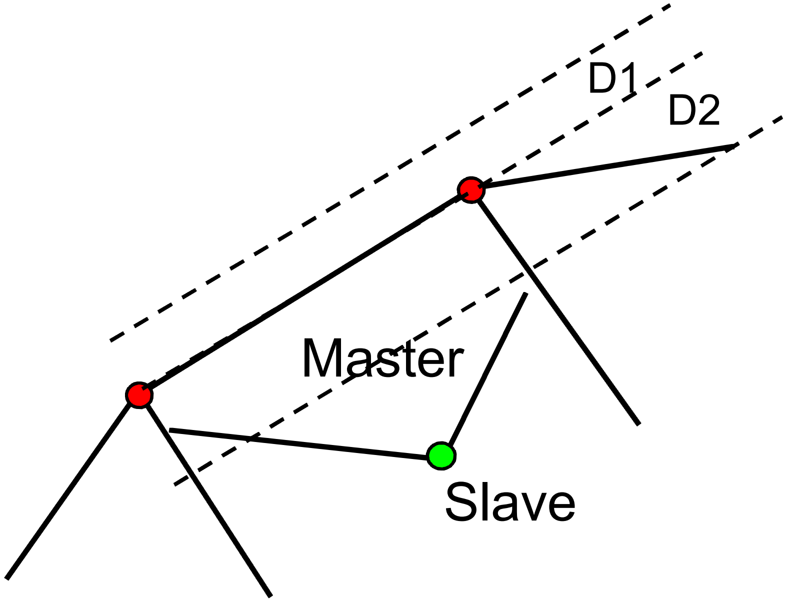
start of iteration

Equilibrium

$F_c \overset{?}{>}$ FNTOL
yes: separation
no: contact

CONTACT DETECTION – SCENARIO 4

- **Slave grid <u>inside</u> master but <u>outside</u> D2**
  - Slave grid has penetrated
    - For statics: repeat iteration with scaled back Dui (maximum penetration: 10% of D2). If converged check for separation. If separated continue iterating. Otherwise go to the next increment.
    - For dynamics: scale back the timestep appropriately.
  - If this situation occurs at the beginning of the analysis, contact will not be found.
    - Prepare your model so that this situation does not occur initially



EFFECT OF CONTACT TOLERANCES

- **The sizes of the contact tolerances D1 and D2 have a significant impact on the computational costs and the accuracy of the solution**
- **Contact tolerances too small:**
  - Detection of contact is difficult, leading to higher costs. Initial contact might not be detected.
- **Contact tolerance too large:**
  - Nodes are considered in contact prematurely, resulting in a loss of accuracy
  - Nodes might “penetrate” the surface by a large amount

CONTACT SEARCH ORDER

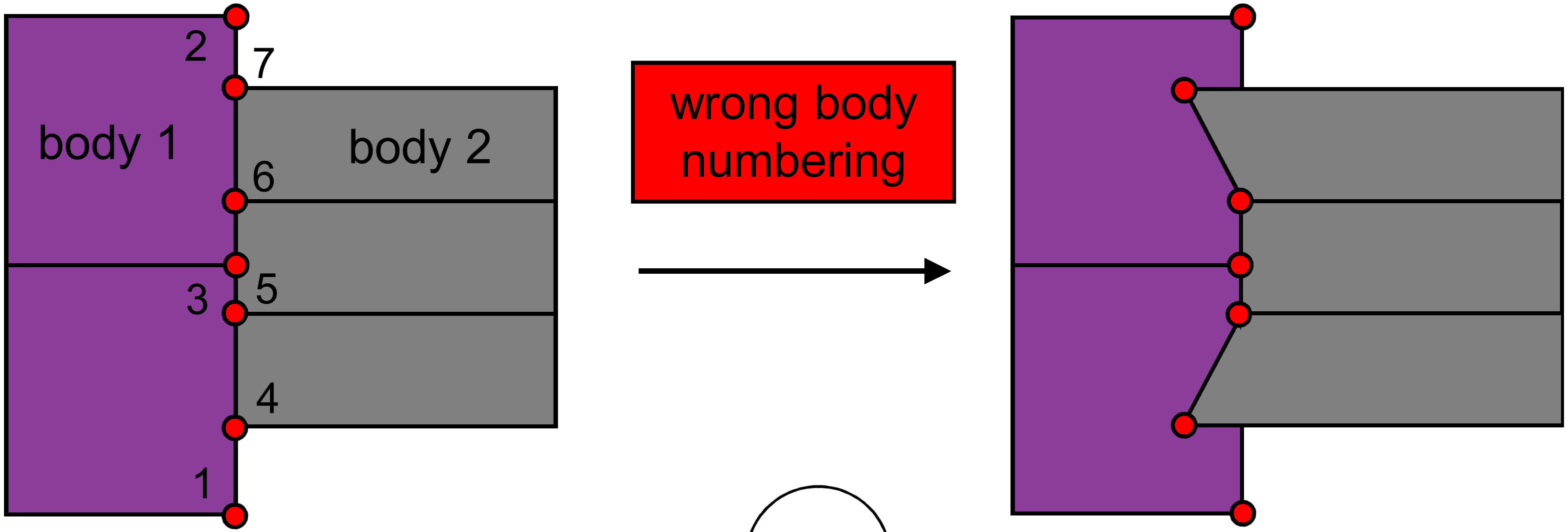
- The contact search order is very important for node-to-segment contact – this is the largest disadvantage of Node to Segment contact
- There are various options to control how the search is carried out (order in which bodies attempt to contact each other).
- It is important to understand how each search strategy works when used with contact bodies of different mesh densities, materials, etc.
- Simply using the defaults can result in contacts which, while they converge, do not well model the situation.

CONTACT SEARCH ORDER

- **Single order search (ISTYP=1) on BCBODY**
  - by default, the lower numbered body is checked with itself and the higher numbered bodies. For instance, body 1 is checked against bodies 1, 2, 3, ... Body 2, however, is only checked against bodies 2, 3, ...
- **Double order search (ISTYP=0, default)**
  - checks possible contact between any two surfaces (surface i is checked for contact with surface j, and surface j is also checked for contact with surface i, where i, j = 1, 2, 3, ..., total number of surfaces in the problem)
- **In both cases, penetrations can occur (discussed next)**

CONTACT SEARCH ORDER

- **Check Body 1:**



wrong body numbering

1.

$$u_3 = u(\underbrace{u_5, u_6})$$

dependent

independent

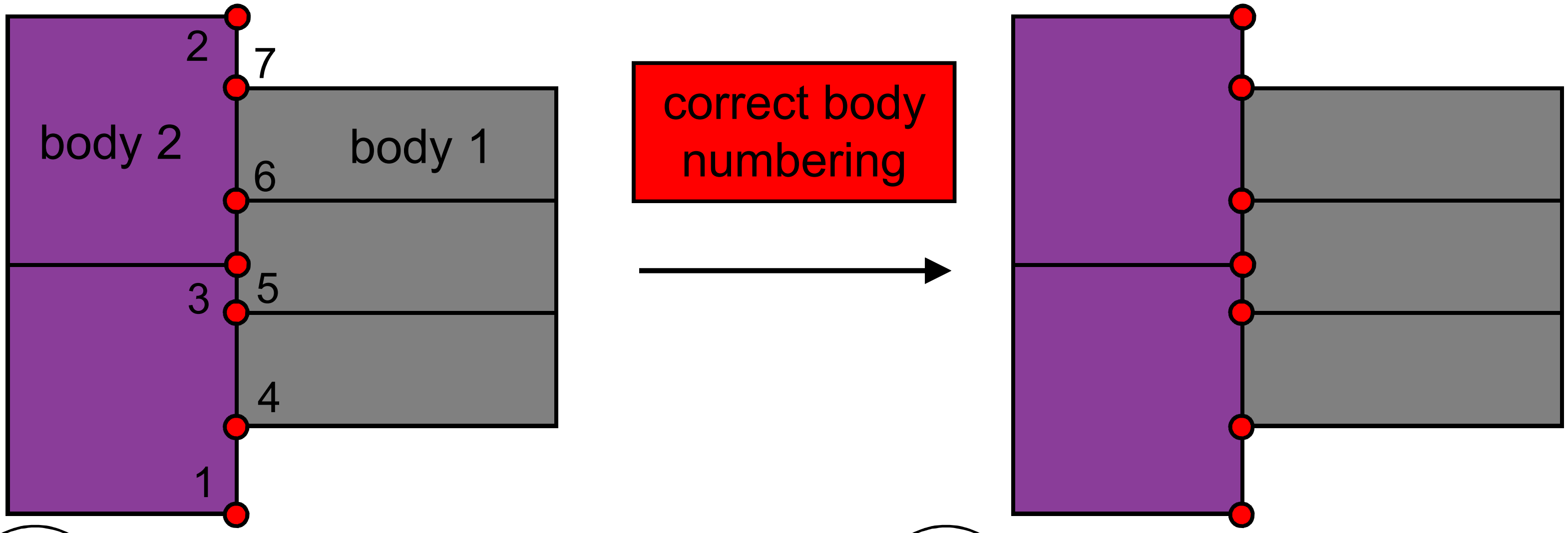
2. **Check Body 2:**

$$\left.\begin{aligned} u_4 &= u(u_1, u_3) \\ u_5 &= u(u_1, u_3) \\ u_6 &= u(u_3, u_2) \\ u_7 &= u(u_3, u_2) \end{aligned}\right\}$$

skipped, since node 3 has already been used as a slave node

CONTACT SEARCH ORDER

- **Check Body 2:**



1.

Check Body 1:

$u_4 = u(u_1, u_3)$

$u_5 = u(u_1, u_3)$

$u_6 = u(u_3, u_2)$

$u_7 = u(u_3, u_2)$

2.

$u_3 = u(u_5, u_6)$

skipped, but okay!

CONTACT SEARCH ORDER

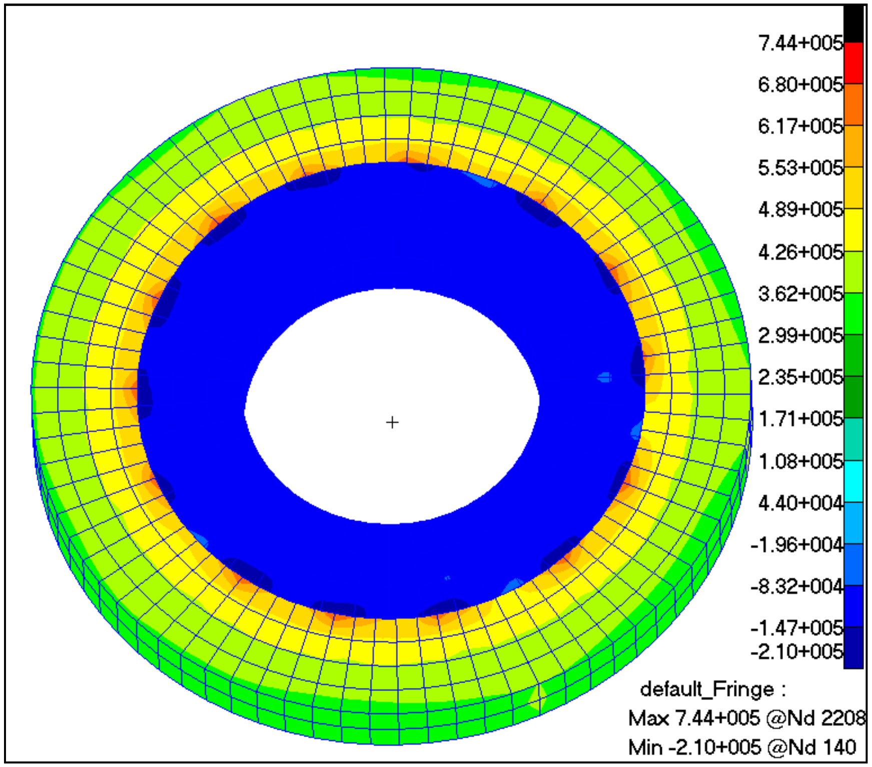
- **The default search order can be overruled by specifying ISEARCH on the BCONPRG entry:**
  - 0: double search (lower numbered bodies first, i.e. 1,1; 1,2; 1,3; 2,1; 2,2; 2,3; 3,1; 3,2; 3,3). Default.
  - 1: searching from slave to master (single sided)
  - 2: body with finer mesh automatically becomes slave
- **ISTYP in BCBDPRP can also be set to 2 (ISEARCH must be 0)**
  - double order search with automatic optimization of contact constraint equations ("optimized contact")
    - The decision is made for the areas in contact. The bodies which are softer and finer meshed in those areas come first. Soft has higher priority than mesh (based on HARDS in BCONPRG, the hard-soft ratio, default=2.0).

SEGMENT TO SEGMENT CONTACT ADVANTAGES

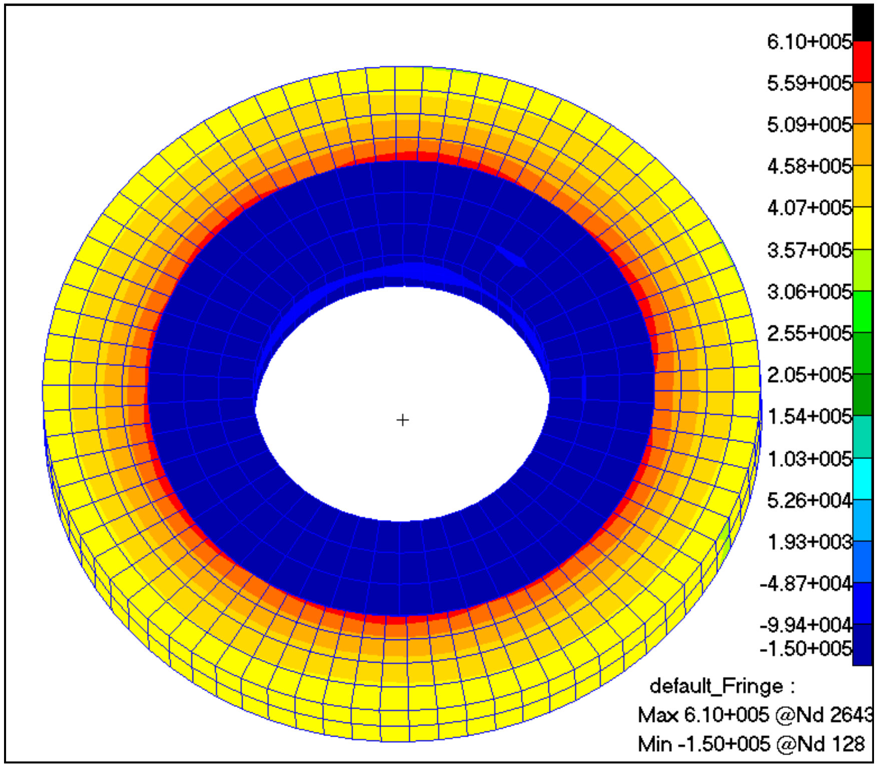
- **Node to Segment Contact (NTS) has some drawbacks which can be avoided using Segment to Segment Contact (STS)**

Improved accuracy

Node to Segment
(Stress)



Segment to Segment
(Stress)



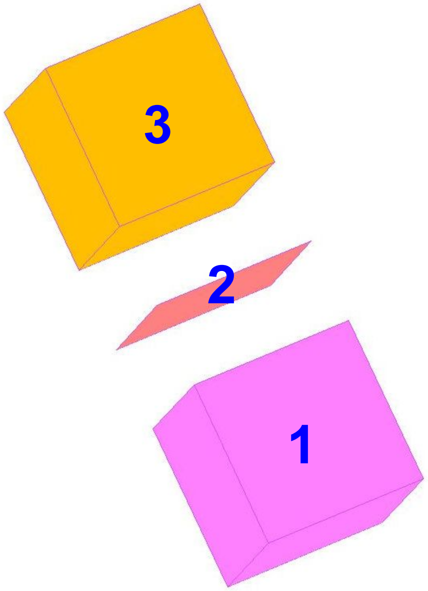
SEGMENT TO SEGMENT CONTACT ADVANTAGES

- **Advantages for Shell Contact**
  - For NTS, a shell cannot be “clamped” between two rigid bodies. This is possible with STS.
  - For NTS, a shell contact between two deformable bodies needs some care. STS does not.

NTS

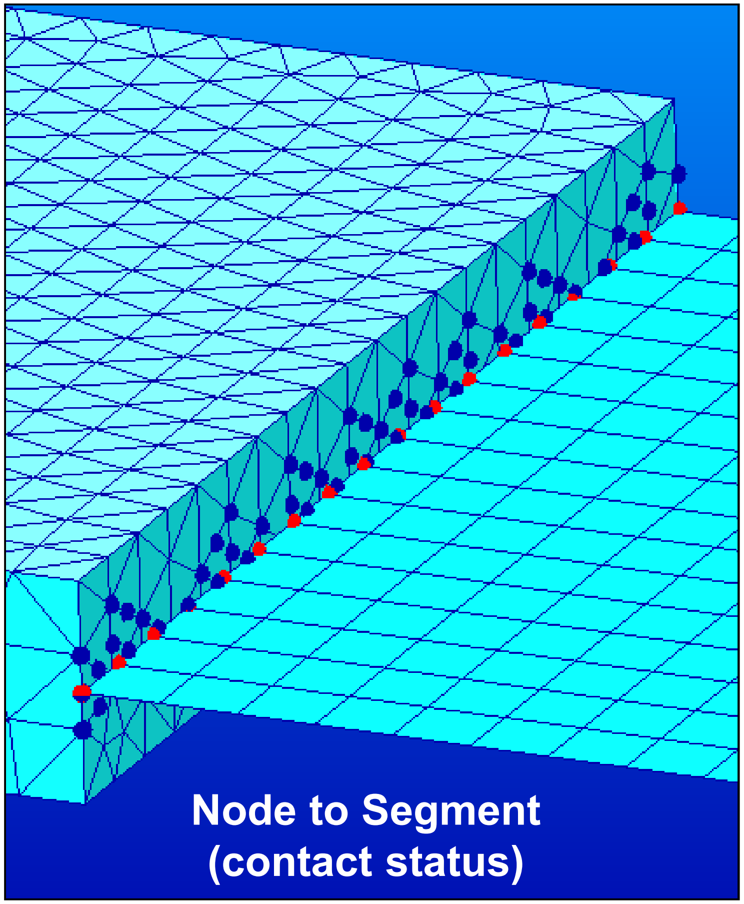
body 1 > body 2
body 2 > body 3
or
body 1 > body 2
body 3 > body 2

but not
Body 2 > body 1
Body 2 > body 3

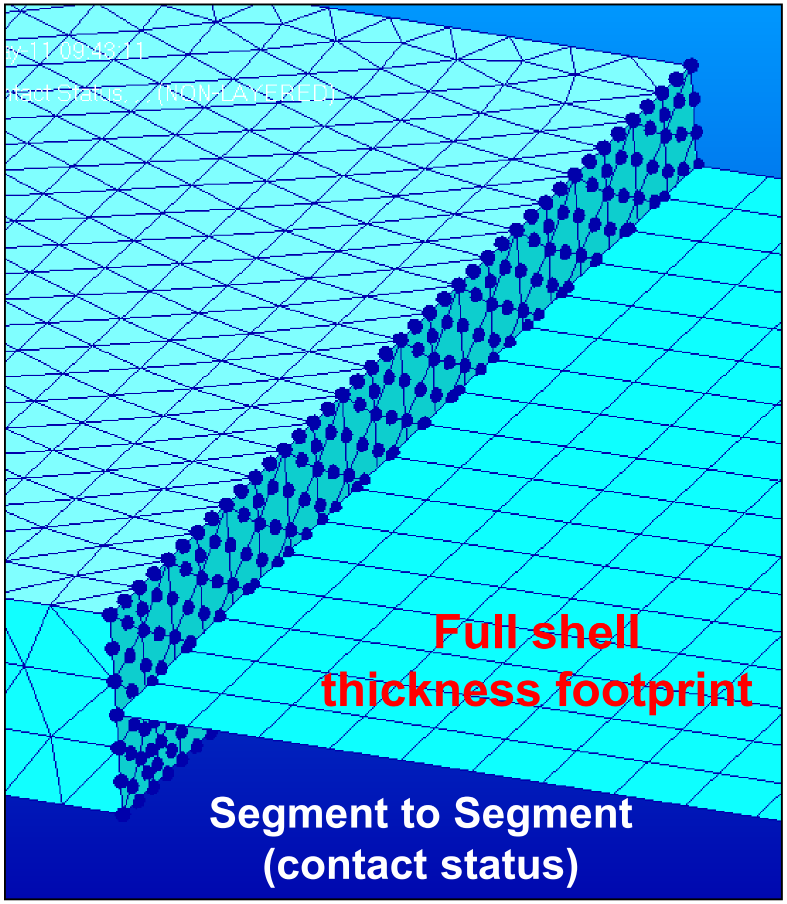


SEGMENT TO SEGMENT CONTACT ADVANTAGES

- **Advantages for Shell Contact (cont.)**
  - For NTS, shell edge contact is described as a line. STS describes the “footprint”.



Node to Segment (contact status)

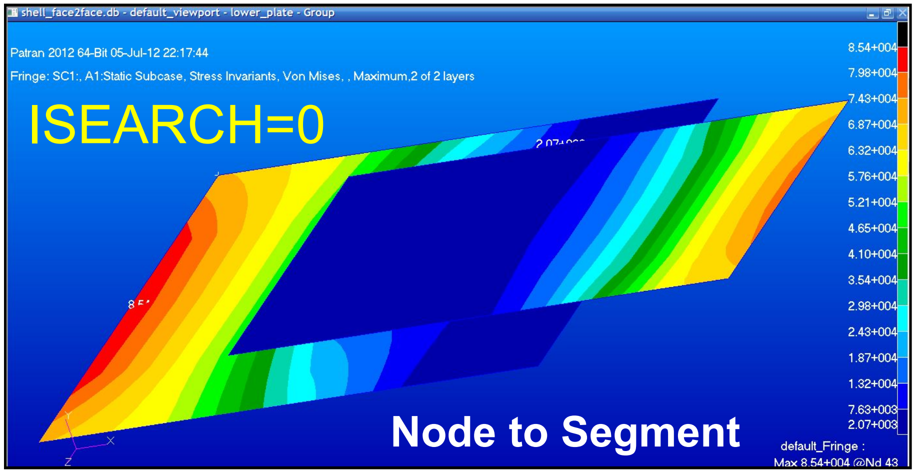


Full shell thickness footprint

Segment to Segment (contact status)

SEGMENT TO SEGMENT CONTACT ADVANTAGES

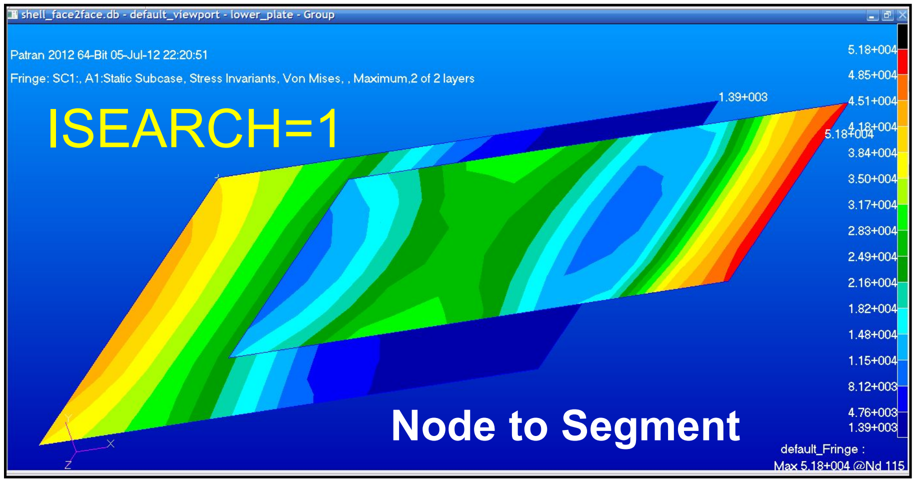
- No Master-Slave Concept (avoid slave master search order)



shell_face2face.db - default_viewport - lower_plate - Group

Patran 2012 64-Bit 05-Jul-12 22:17:44

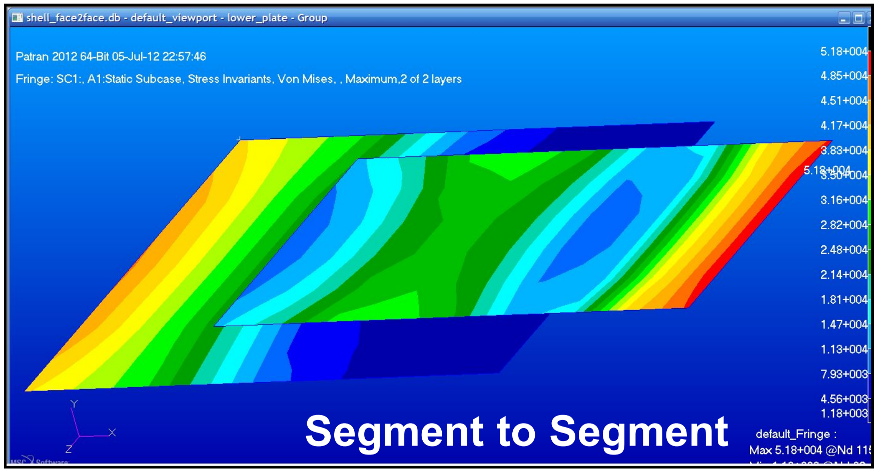
Fringe: SC1:, A1:Static Subcase, Stress Invariants, Von Mises, , Maximum,2 of 2 layers



shell_face2face.db - default_viewport - lower_plate - Group

Patran 2012 64-Bit 05-Jul-12 22:20:51

Fringe: SC1:, A1:Static Subcase, Stress Invariants, Von Mises, , Maximum,2 of 2 layers



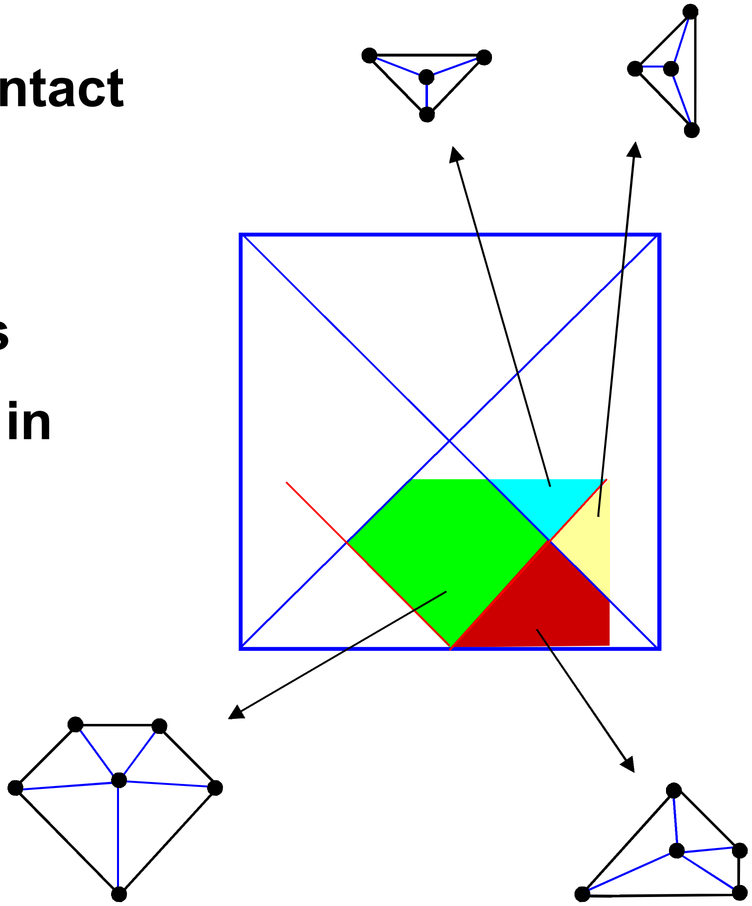
shell_face2face.db - default_viewport - lower_plate - Group

Patran 2012 64-Bit 05-Jul-12 22:57:46

Fringe: SC1:, A1:Static Subcase, Stress Invariants, Von Mises, , Maximum,2 of 2 layers

DEFINITION OF STS CONTACT

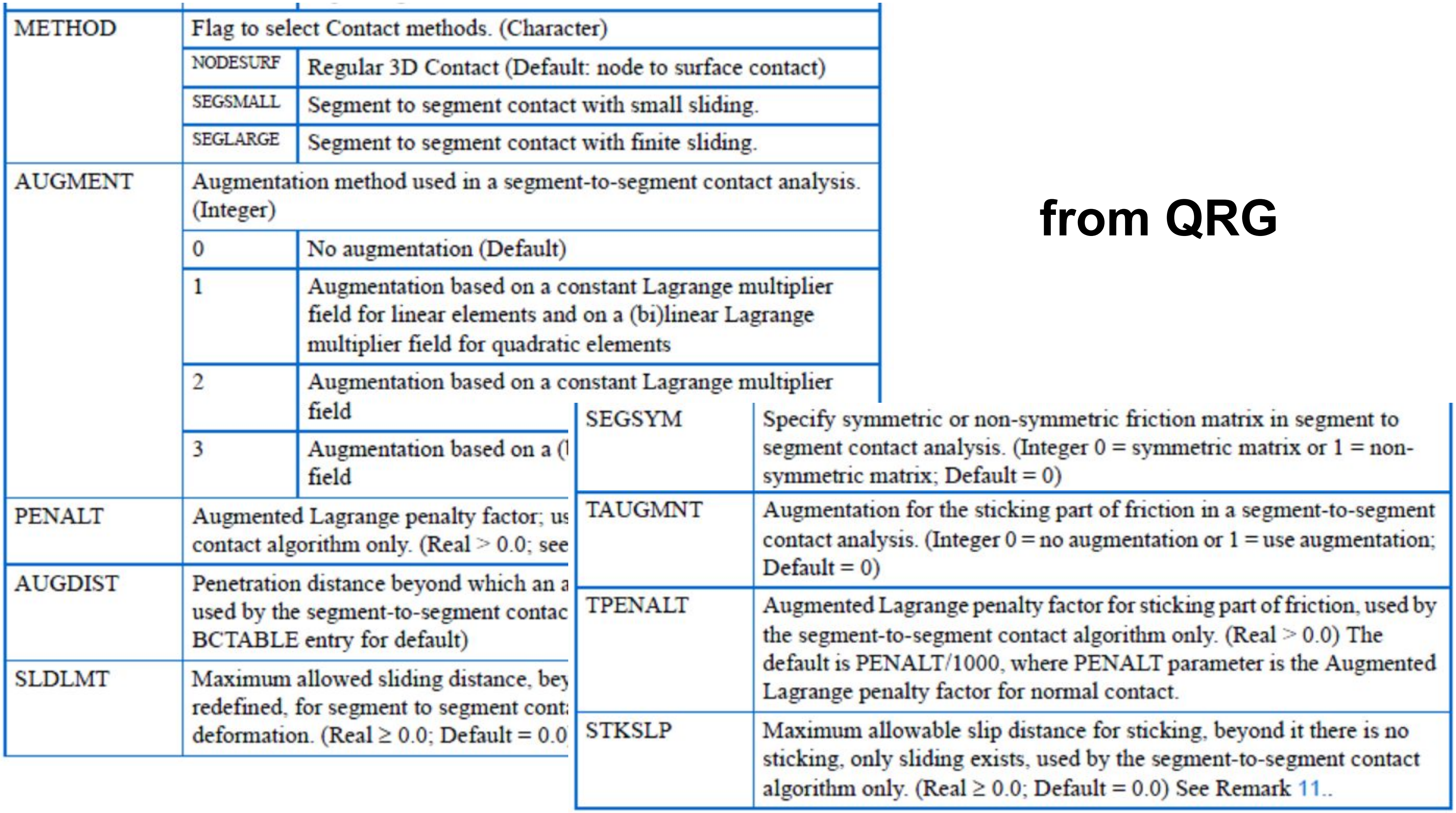
- Find the segments that are in contact
- Find the common areas of the segments that are in contact
- Divide these areas into polygons
- Find the polygon points that are in contact



- The polygon points define a local connection between two contact segments and the non-penetration constraints are enforced using augmented Lagrangian approach.

STS CONTACT INPUT

- **STS Contact can be controlled via BCPARA**



METHOD	Flag to select Contact methods. (Character)	
	NODESURF	Regular 3D Contact (Default: node to surface contact)
	SEGSMALL	Segment to segment contact with small sliding.
	SEGLARGE	Segment to segment contact with finite sliding.
AUGMENT	Augmentation method used in a segment-to-segment contact analysis. (Integer)	
	0	No augmentation (Default)
	1	Augmentation based on a constant Lagrange multiplier field for linear elements and on a (bi)linear Lagrange multiplier field for quadratic elements
	2	Augmentation based on a constant Lagrange multiplier field
	3	Augmentation based on a (field
PENALT	Augmented Lagrange penalty factor; us contact algorithm only. (Real > 0.0; see	
AUGDIST	Penetration distance beyond which an a used by the segment-to-segment contac BCTABLE entry for default)	
SLDLMT	Maximum allowed sliding distance, bey redefined, for segment to segment cont deformation. (Real ≥ 0.0; Default = 0.0	

SEGSYM	Specify symmetric or non-symmetric friction matrix in segment to segment contact analysis. (Integer 0 = symmetric matrix or 1 = non-symmetric matrix; Default = 0)
TAUGMNT	Augmentation for the sticking part of friction in a segment-to-segment contact analysis. (Integer 0 = no augmentation or 1 = use augmentation; Default = 0)
TPENALT	Augmented Lagrange penalty factor for sticking part of friction, used by the segment-to-segment contact algorithm only. (Real > 0.0) The default is PENALT/1000, where PENALT parameter is the Augmented Lagrange penalty factor for normal contact.
STKSLP	Maximum allowable slip distance for sticking, beyond it there is no sticking, only sliding exists, used by the segment-to-segment contact algorithm only. (Real ≥ 0.0; Default = 0.0) See Remark 11..

from QRG

STS CONTACT INPUT

- **STS Contact can be controlled for contact pairs via Geometric Contact Parameters in BCONPRG**
  - AUGDIST: Penetration distance beyond which an augmentation will be applied
  - PENALT: Augmented Lagrange penalty factor
  - STKSLP: Maximum allowable slip distance for sticking, beyond it there is no sticking, only sliding exists
  - TPENALT: Augmented Lagrange penalty factor for sticking part of friction

STS CONTACT INPUT

- **METHOD to select small or large sliding STS**
  - NODESURF: NTS (default)
  - SEGSMALL: STS, small sliding
  - SEGLARGE: STS, large sliding
- **AUGMENT to select the Augmentation Method**
  - 0: no augmentation (default)
  - 1: constant Lagrange Multiplier field for linear elements, (bi)linear field for quadratic elements
  - 2: constant Lagrange Multiplier field
  - 3: (bi)linear Lagrange Multiplier field
  - For rigid-deformable contact Augmentation is always on

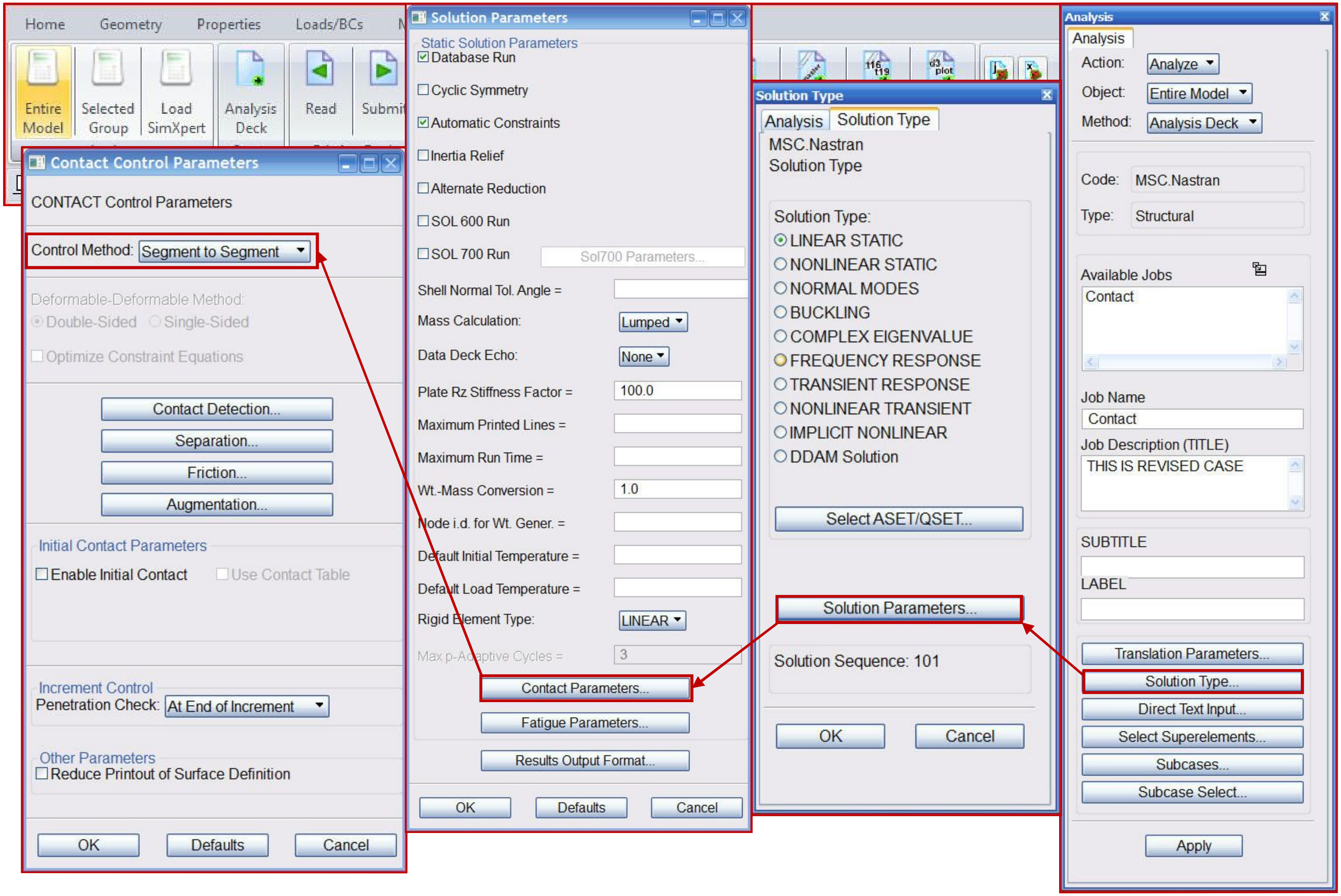
STS CONTACT INPUT

- **PENALT, Augmented Lagrange penalty factor**
  - Default: depends on the contacting body stiffnesses and a characteristic length (unit=force/length^3)
- **AUGDIST, Penetration distance beyond which an augmentation will be applied**
  - Default: 1e-3 of the characteristic length
- **SLDLMT, for seglarge. Sliding distance beyond which contact segments are to be redefined**
  - Default: 5 times the default contact tolerance ERROR
- **SEGSYM, symmetric or non-symmetric friction, 0: symmetric (default), 1: non-symmetric**

STS CONTACT INPUT

- **TAUGMNT, Augmentation for the sticking part of friction**
  - 0: no Augmentation (Default), 1: Augmentation
- **TPENALT, Augmented Lagrange penalty factor for sticking part of friction**
  - default is PENALT/1000
- **STKSLP, Maximum allowable slip distance for sticking**
  - 0.0: maximum sticking displacement (default)

DEFINE STS IN PATRAN



Contact Control Parameters

CONTACT Control Parameters

Control Method: Segment to Segment

Deformable-Deformable Method: ● Double-Sided ○ Single-Sided

☐ Optimize Constraint Equations

Initial Contact Parameters

☐ Enable Initial Contact ☐ Use Contact Table

Increment Control

Penetration Check: At End of Increment

Other Parameters

☐ Reduce Printout of Surface Definition

Solution Parameters

Static Solution Parameters

☑ Database Run

☐ Cyclic Symmetry

☑ Automatic Constraints

☐ Inertia Relief

☐ Alternate Reduction

☐ SOL 600 Run

☐ SOL 700 Run

Shell Normal Tol. Angle =

Mass Calculation: Lumped

Data Deck Echo: None

Plate Rz Stiffness Factor = 100.0

Maximum Printed Lines =

Maximum Run Time =

Wt.-Mass Conversion = 1.0

Node i.d. for Wt. Gener. =

Default Initial Temperature =

Default Load Temperature =

Rigid Element Type: LINEAR

Max p-Adaptive Cycles = 3

Solution Type

MSC.Nastran
Solution Type

Solution Type:

● LINEAR STATIC
○ NONLINEAR STATIC
○ NORMAL MODES
○ BUCKLING
○ COMPLEX EIGENVALUE
○ FREQUENCY RESPONSE
○ TRANSIENT RESPONSE
○ NONLINEAR TRANSIENT
○ IMPLICIT NONLINEAR
○ DDAM Solution

Solution Sequence: 101

Analysis

Action: Analyze

Object: Entire Model

Method: Analysis Deck

Code: MSC.Nastran

Type: Structural

Available Jobs
Contact

Job Name
Contact

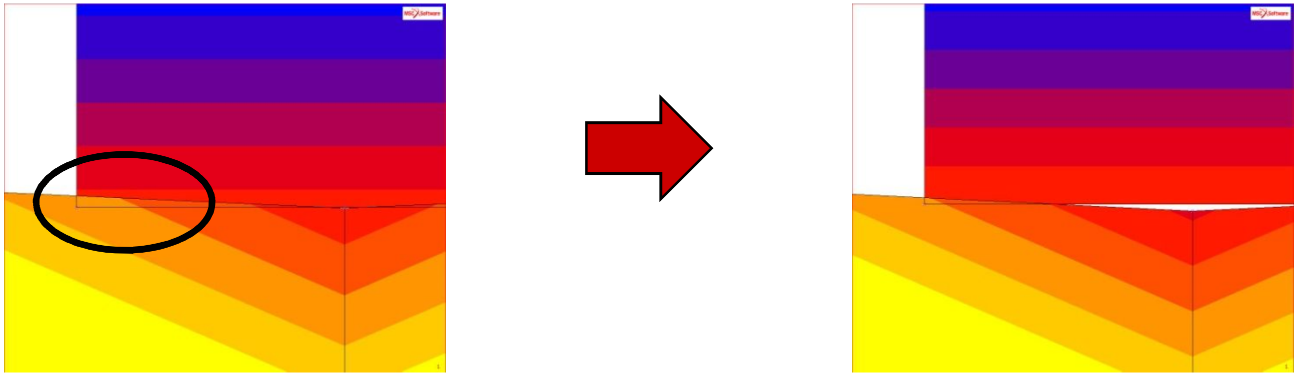
Job Description (TITLE)
THIS IS REVISED CASE

SUBTITLE

LABEL

INFLUENCE OF AUGMENTATION

- **Augmentation enables minimization of penetration as the following example demonstrates**



- **The augmentation method will influence the number of iterations and the results; however, to get a quick insight in a problem you could choose not to augment. This is therefore the default.**

STS CONTACT DETECTION PARAMETERS

- **Contact Distance Tolerance, same as NTS, but**
  - SLDLMT is defaulted to 5 times ERROR. For small values of ERROR, this may unnecessarily result in the message “Recalculating Segments”, thus increasing the computational costs.
- **Separation Control**
  - Only stress based separation based upon absolute stresses is available. Be careful when an input deck comes from NTS and an FNTOL refers to force.
- **Search order is immaterial**
  - ISEARCH, ISTYP, HARDS are ignored

STS CONTACT LIMITATIONS

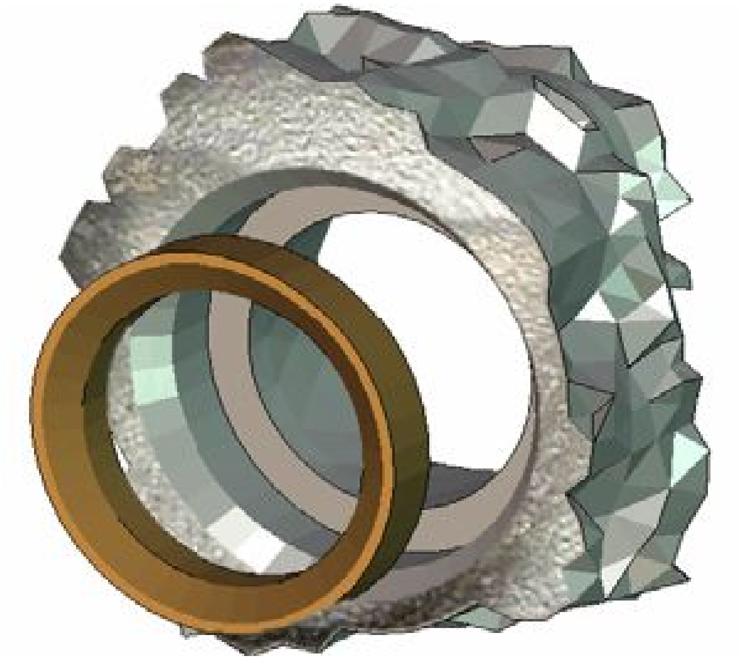
- **Not available at this time:**
  - Thermal analysis
  - Coupled analysis with multiple physics
  - Brake squeal analysis
  - Adaptive meshing
  - Beam to beam contact
  - Breaking glue
  - Output of normal contact stress and friction contact stress for glued or frictional contact
  - The following parameters are not supported:
    - MAXSEP, ICSEP, IBSEP, RVCNST, BEAMB and NLGLUE

OVERVIEW

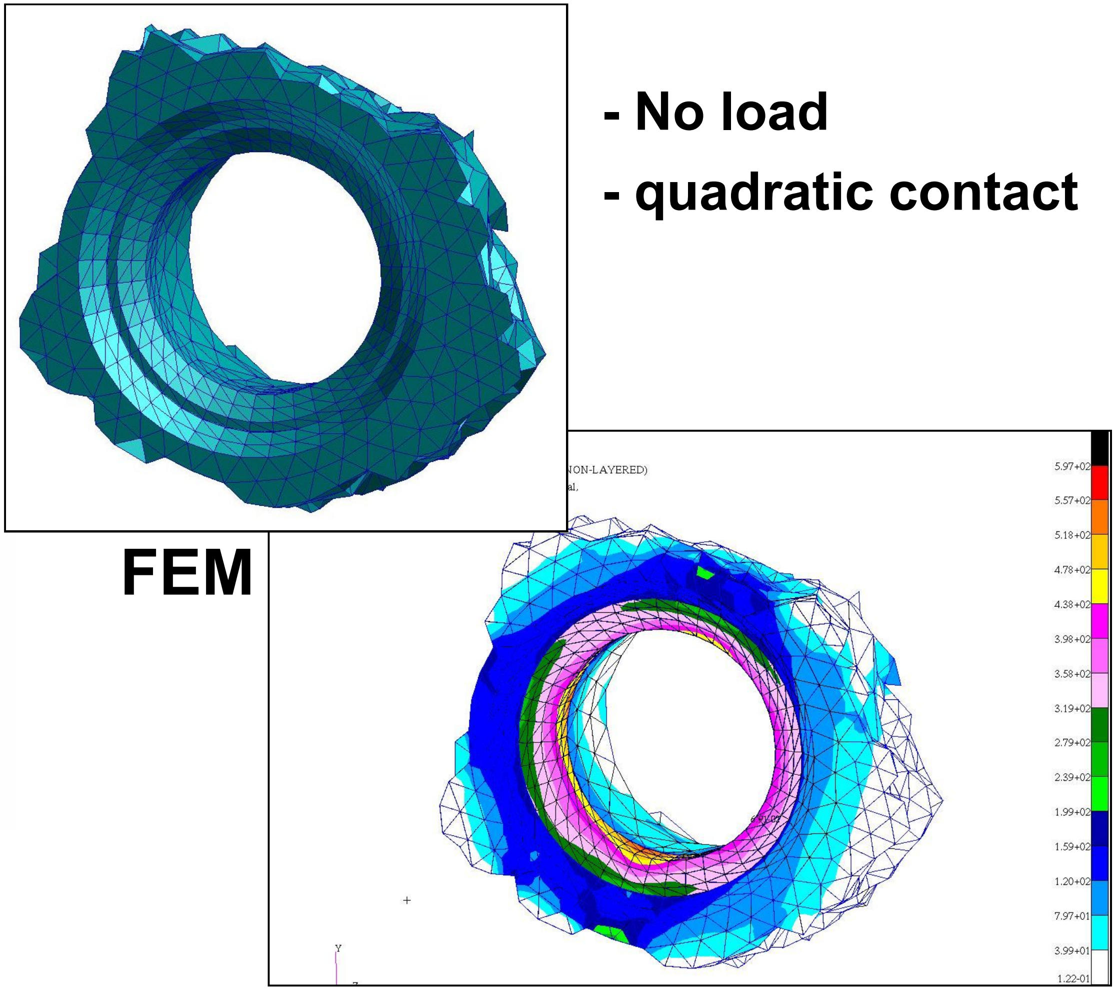
- What is and when to consider contact
- Contact Bodies (Workshop – Rubber Door Seal)
- Contact Pairs/Tables (Workshop – Deformable to Rigid Contact)
- Contact Detection
  - Node to segment
  - Segment to segment
- **Special Features**
  - Interference (Workshop – Interference Fit)
  - Stress free initial contact
  - Glued contact (Workshop – Contact Pairs)
  - Contact with Shells
  - Friction
- Convergence and Controls with contact

CONTACT INTERFERENCE – EXAMPLE

- **Valve Insert Fitted into Cylinder Head**



Real Model



FEM

- No load
- quadratic contact

von Mises

CONTACT INTERFERENCE

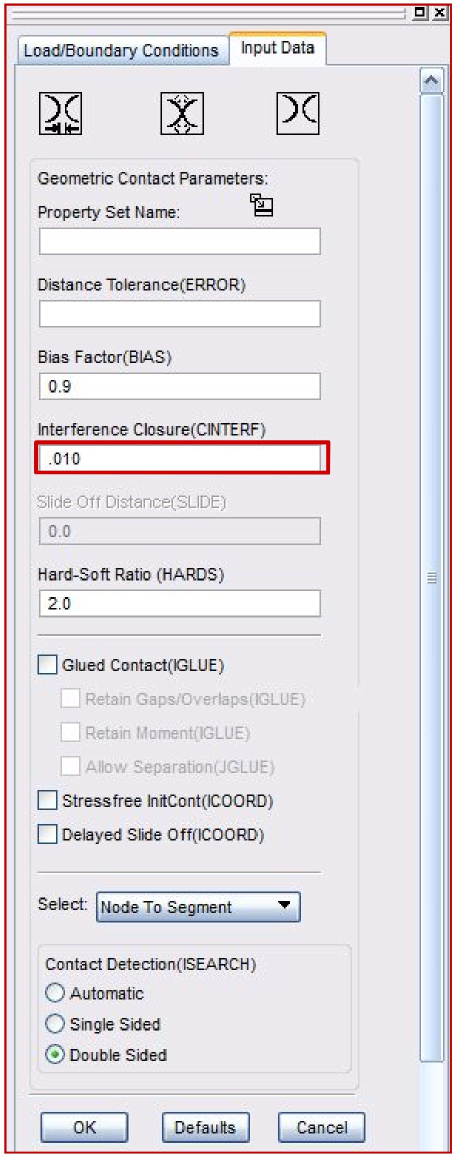
- **The interference closure distance is defined via CINTERF in BCONPRG entry**
  - It only affects nodes which are already in contact
  - The regular contact-related incremental displacement is modified by:

$$\Delta\bar{v}_A := \Delta\bar{v}_A + d_i\bar{y}$$

  - di is CINTERF, the user-defined interference closure distance normal to the contact surface
- **When contact bodies are moving**
  - CINTERF>0: contact starts earlier by di (initial interference)
  - CINTERF<0: contact occurs after penetration of di (initial clearance)

CONTACT INTERFERENCE

- **In Patran:**



Geometric Contact Parameters:

Property Set Name:

Distance Tolerance(ERROR)

Bias Factor(BIAS) 0.9

Interference Closure(CINTERF) .010

Slide Off Distance(SLIDE) 0.0

Hard-Soft Ratio (HARDS) 2.0

☐ Glued Contact(IGLUE)
  ☐ Retain Gaps/Overlaps(IGLUE)
  ☐ Retain Moment(IGLUE)
  ☐ Allow Separation(JGLUE)
☐ Stressfree InitCont(ICOORD)
☐ Delayed Slide Off(ICOORD)

Select: Node To Segment

Contact Detection(ISEARCH)
○ Automatic
○ Single Sided
● Double Sided

Specification of interference

CONTACT INTERFERENCE

- **BCONPRG – used to define CINTERF**

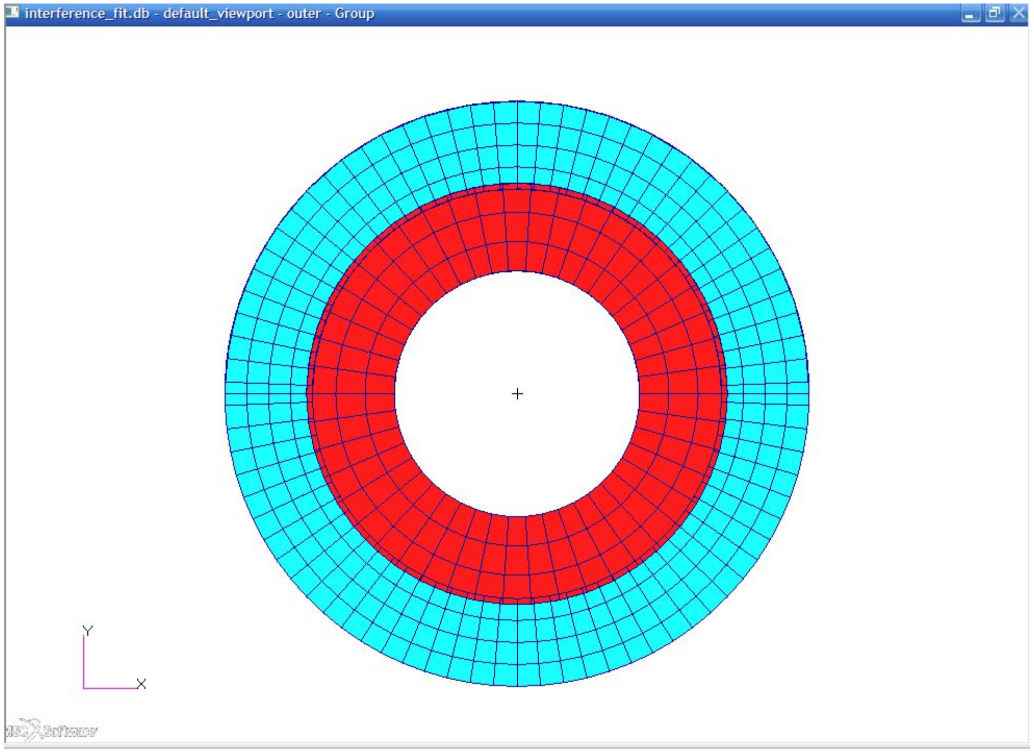
BCONPRG	BCGPID		PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	
	PARAM4	VAL4	PARAM5	VAL5	-etc.-				

- **Example**

BCONPRG	90		**CINTERF**	**.01**					

CASE STUDY: INTERFERENCE FIT

- **Define contact bodies**



interference_fit.db - default_viewport - outer - Group

Load/Boundary Conditions

Action: Create

Object: Contact

Type: Element Uniform

Option: Body Pair

Current Load Case: Default...

Type: Static

Existing Sets

New Set Name: contact_pair

Load/Boundary Conditions

Action: Create

Object: Contact

Type: Element Uniform

Option: Deformable Body

Current Load Case: Default...

Type: Static

Existing Sets:
inner
outer

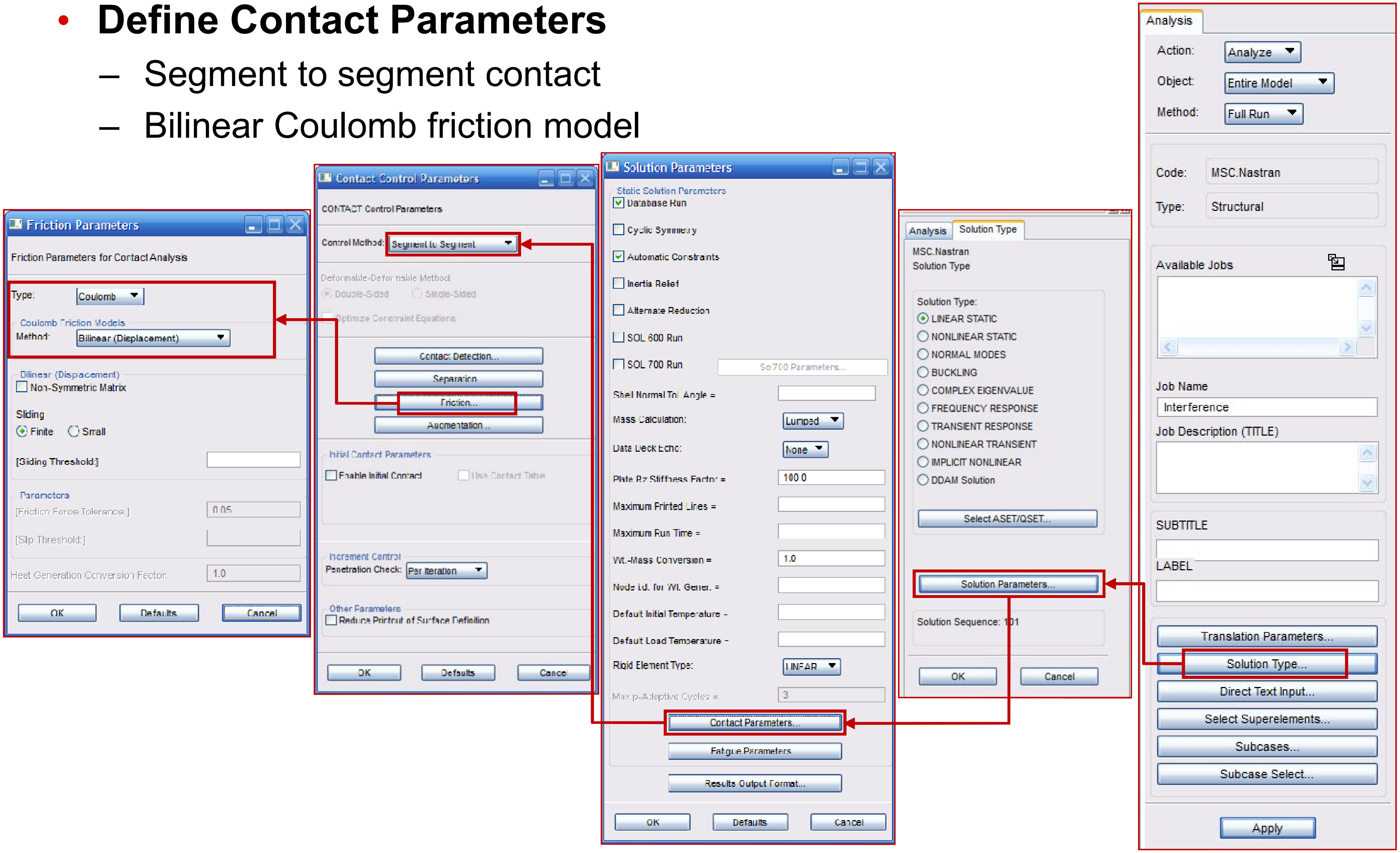
New Set Name

Target Element Type: 3D

Play Case study video:
NAS133_S4_cs_interference_fit

CASE STUDY: INTERFERENCE FIT

- **Define Contact Parameters**
  - Segment to segment contact
  - Bilinear Coulomb friction model



Friction Parameters

Friction Parameters for Contact Analysis

Type: Coulomb

Coulomb Friction Models

Method: Bilinear (Displacement)

Bilinear (Displacement)

☐ Non-Symmetric Matrix

Sliding

● Finite ○ Small

[Sliding Threshold]

Parameters

[Friction Force Tolerance:] 0.05

[Slip Threshold:]

Heat Generation Conversion Factor: 1.0

Contact Control Parameters

CONTACT Control Parameters

Control Method: Segment to Segment

Deformable-Deformable Method

○ Double-Sided ○ Single-Sided

☐ Optimize Constraint Equations

Initial Contact Parameters

☐ Enable Initial Contact ☐ Use Contact Table

Increment Control

Penetration Check: Per Iteration

Other Parameters

☐ Reduce Printout of Surface Definition

Solution Parameters

Static Solution Parameters

☑ Database Run

☐ Cyclic Symmetry

☑ Automatic Constraints

☐ Inertia Relief

☐ Alternate Reduction

☐ SOL 600 Run

☐ SOL 700 Run

Shell Normal Tol. Angle =

Mass Calculation: Lumped

Data Deck Echo: None

Plate Rz Stiffness Factor = 100.0

Maximum Printed Lines =

Maximum Run Time =

Wt.-Mass Conversion = 1.0

Node i.d. for Wt. Gener. =

Default Initial Temperature =

Default Load Temperature =

Rigid Element Type: LINEAR

Max p-Adaptive Cycles = 3

Analysis / Solution Type

MSC.Nastran Solution Type

Solution Type:

● LINEAR STATIC

○ NONLINEAR STATIC

○ NORMAL MODES

○ BUCKLING

○ COMPLEX EIGENVALUE

○ FREQUENCY RESPONSE

○ TRANSIENT RESPONSE

○ NONLINEAR TRANSIENT

○ IMPLICIT NONLINEAR

○ DDAM Solution

Solution Sequence: 101

Analysis

Action: Analyze

Object: Entire Model

Method: Full Run

Code: MSC.Nastran

Type: Structural

Available Jobs

Job Name: Interference

Job Description (TITLE)

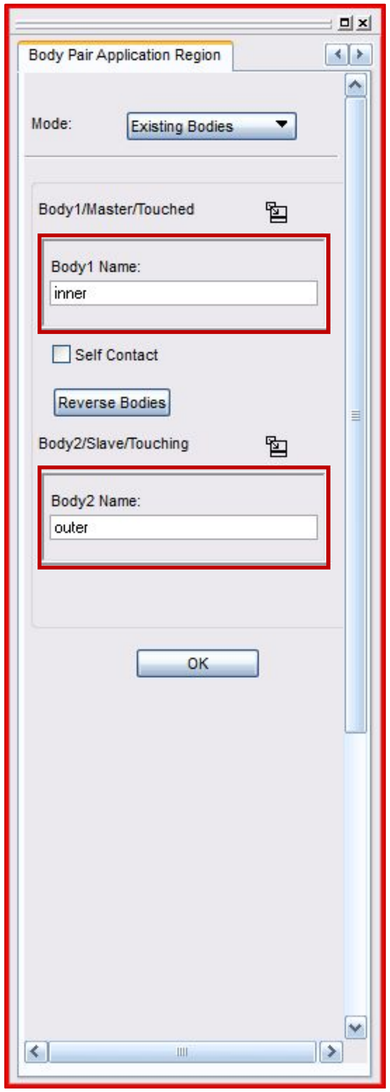
SUBTITLE

LABEL

CASE STUDY: INTERFERENCE FIT

- **Define Contact Pair Parameters**
  - Set contact distance tolerance larger than interference fit amount
  - Enter friction coefficient
- **Define Application Region**
  - Assign master and slave bodies

Select Application Region



Body Pair Application Region

Mode: Existing Bodies

Body1/Master/Touched

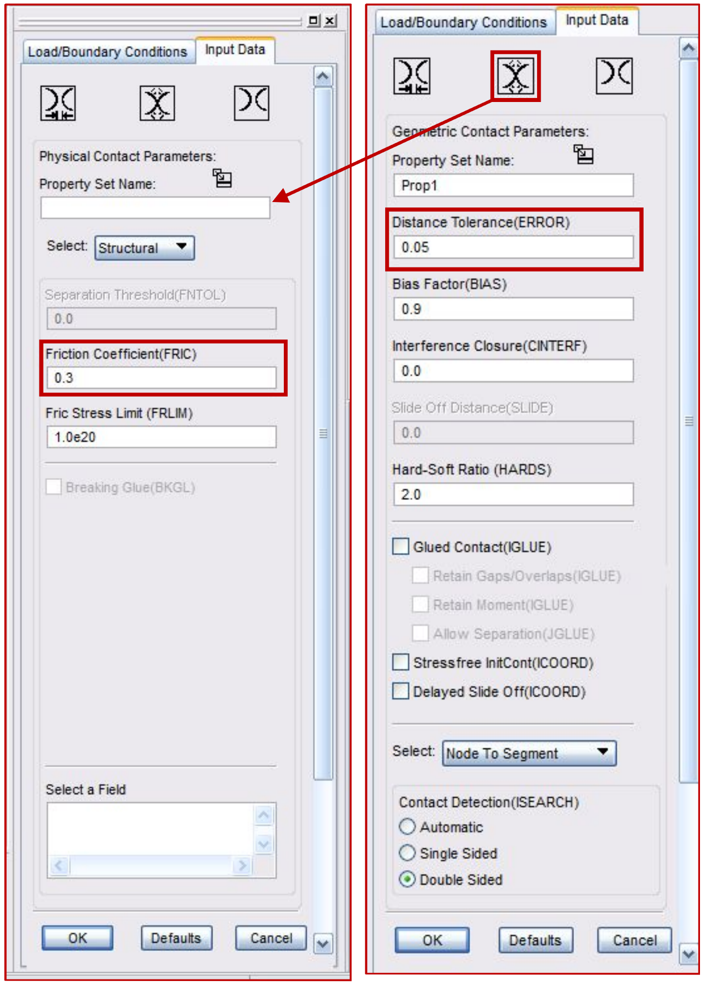
Body1 Name: inner

☐ Self Contact

Body2/Slave/Touching

Body2 Name: outer

Input Data



Physical Contact Parameters:

Property Set Name:

Select: Structural

Separation Threshold(FNTOL): 0.0

Friction Coefficient(FRIC): 0.3

Fric Stress Limit (FRLIM): 1.0e20

☐ Breaking Glue(BKGL)

Select a Field

Geometric Contact Parameters:

Property Set Name: Prop1

Distance Tolerance(ERROR): 0.05

Bias Factor(BIAS): 0.9

Interference Closure(CINTERF): 0.0

Slide Off Distance(SLIDE): 0.0

Hard-Soft Ratio (HARDS): 2.0

☐ Glued Contact(IGLUE)

☐ Retain Gaps/Overlaps(IGLUE)

☐ Retain Moment(IGLUE)

☐ Allow Separation(JGLUE)

☐ Stressfree InitCont(ICOORD)

☐ Delayed Slide Off(ICOORD)

Select: Node To Segment

Contact Detection(ISEARCH)

○ Automatic

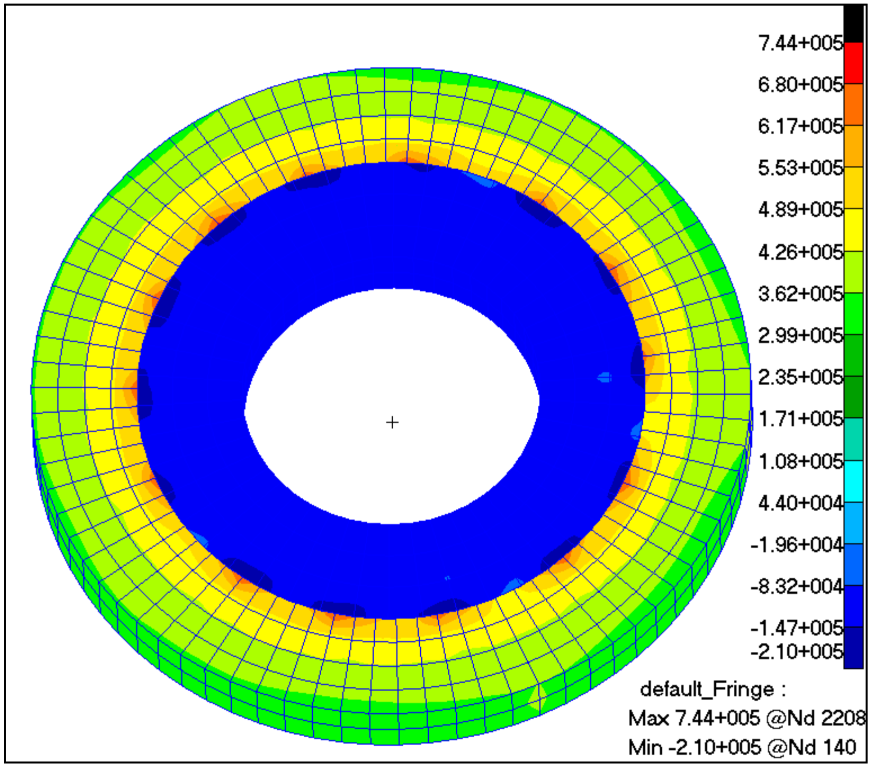
○ Single Sided

● Double Sided

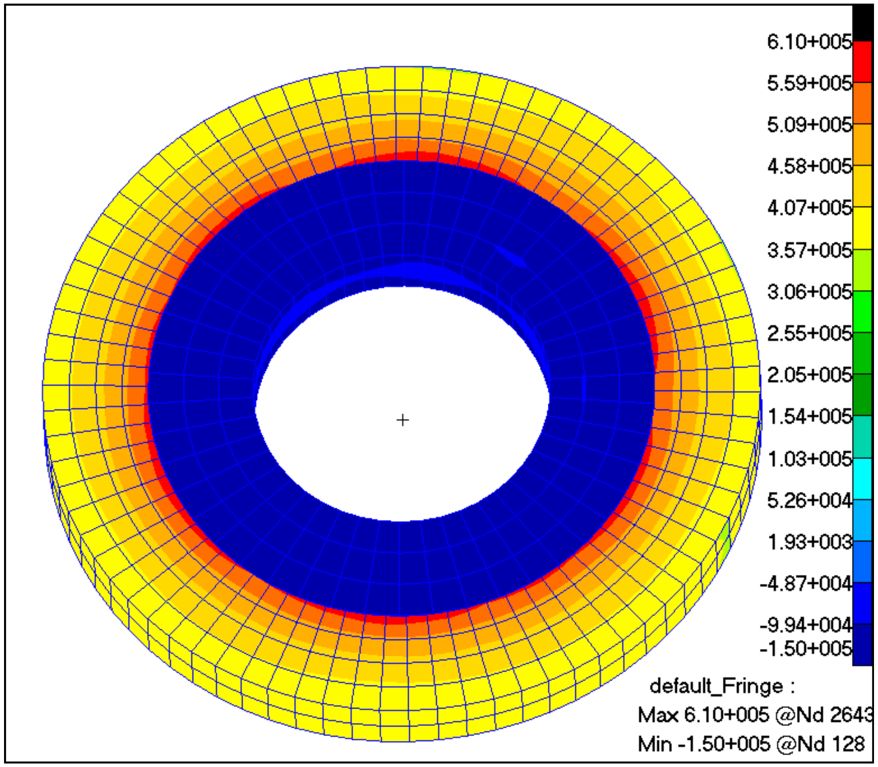
CASE STUDY: INTERFERENCE FIT

- **Plot circumferencial stress**

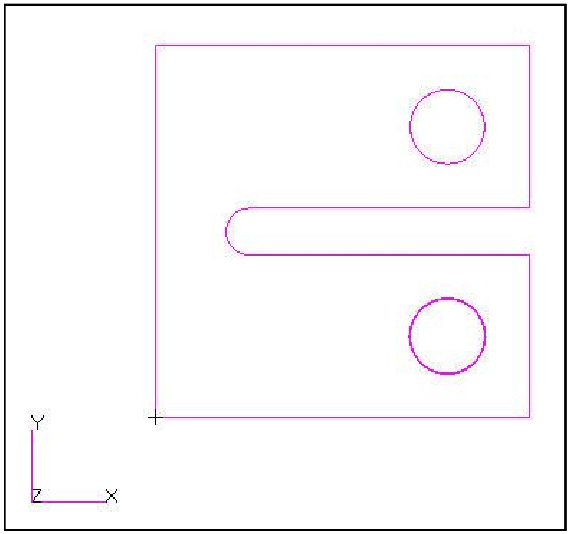
Node to Segment

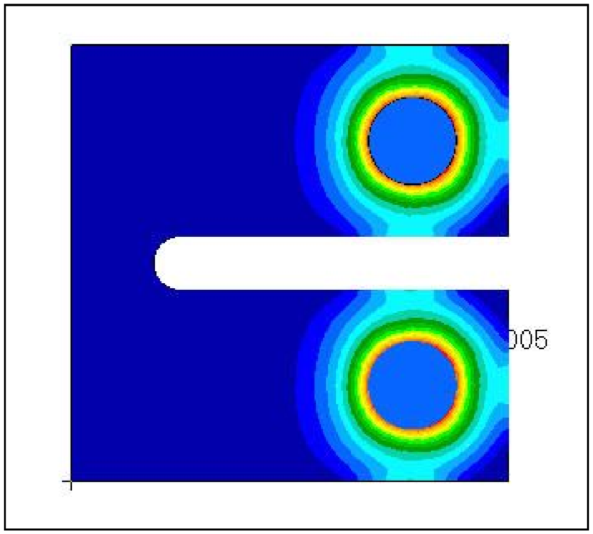


Segment to Segment



WORKSHOP 4 – INTERFERENCE FIT





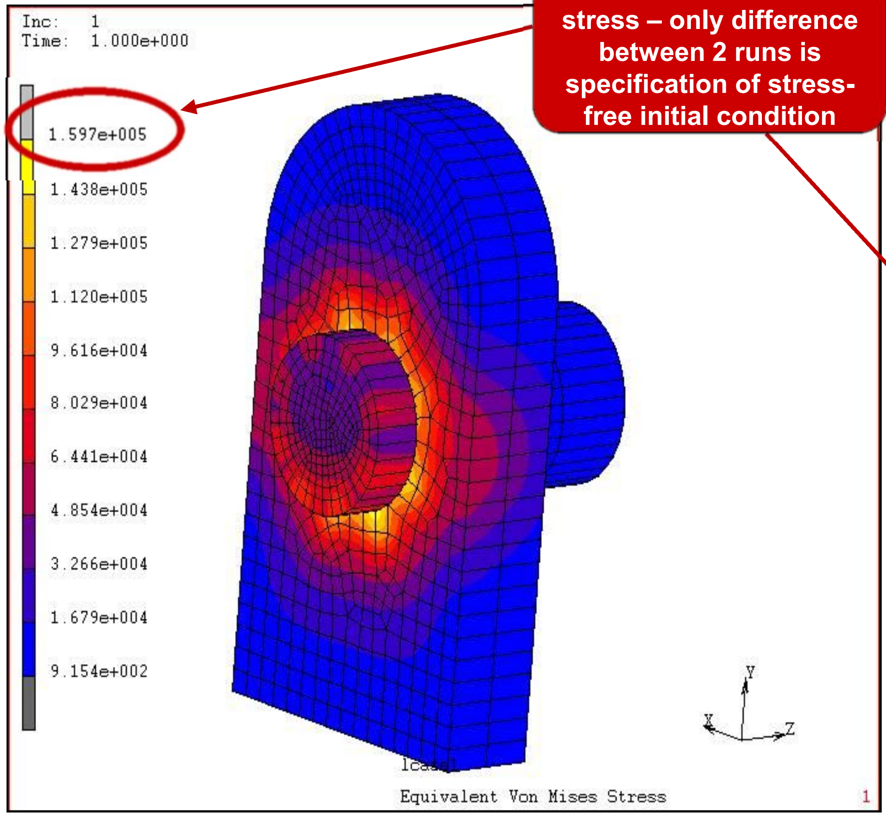
- **Workshop Objectives:**
  - Perform interference fit analysis using deformable-deformable contact
  - Modify contact table parameters
  - Understand the two methods of modeling interference fit
  - Switch between Node to Segment and Segment to Segment contact.

OVERVIEW

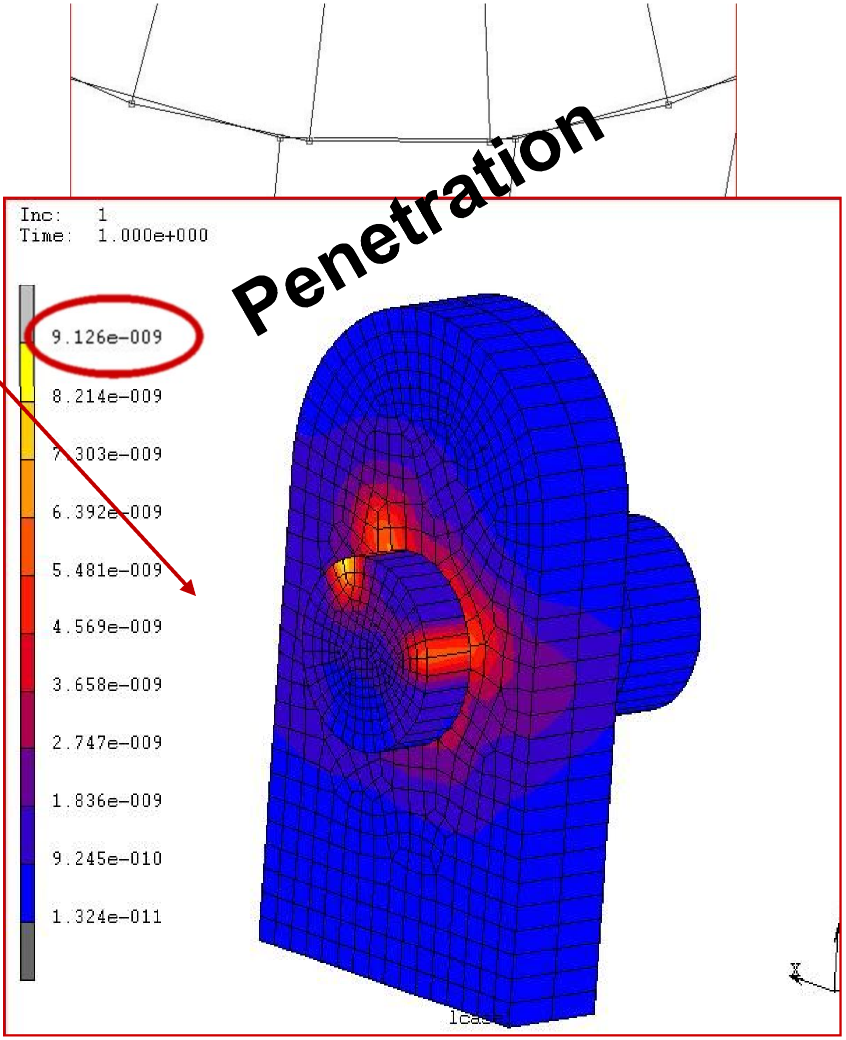
- **What is and when to consider contact**
- **Contact Bodies** (Workshop – Rubber Door Seal)
- **Contact Pairs/Tables** (Workshop – Deformable to Rigid Contact)
- **Contact Detection**
  - Node to segment
  - Segment to segment
- **Special Features**
  - Interference (Workshop – Interference Fit)
  - Stress free initial contact
  - Glued contact (Workshop – Contact Pairs)
  - Contact with Shells
  - Friction
- **Convergence and Controls with contact**

INITIAL STRESS FREE - EXAMPLE

Note difference in max stress – only difference between 2 runs is specification of stress-free initial condition



No Initial Stress-Free

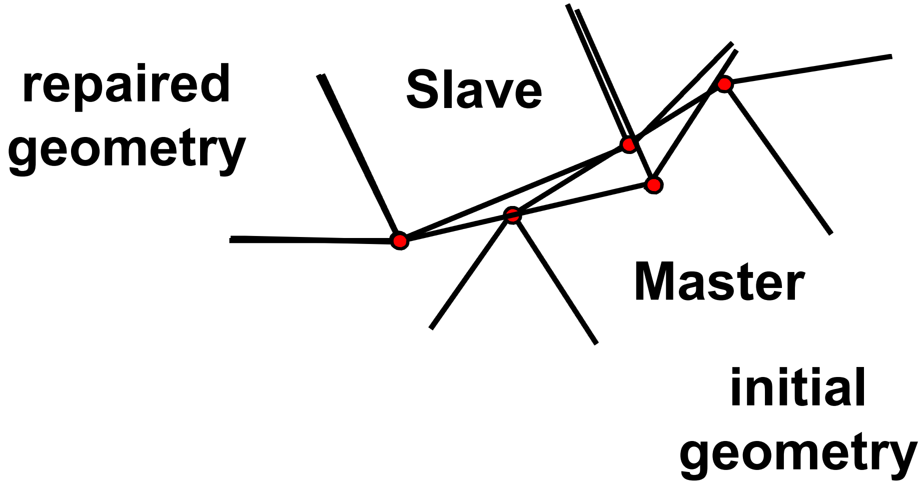


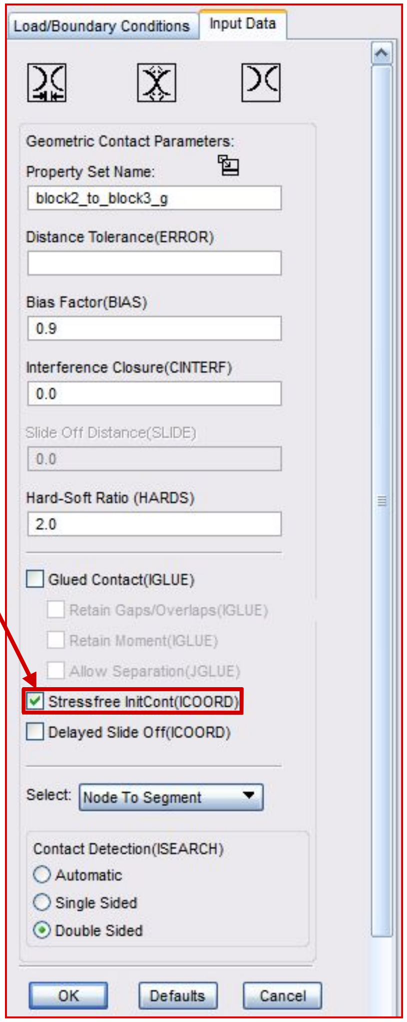
Penetration

Initial Stress-Free

INITIAL STRESS FREE CONTACT

- **Before starting with the first increment a check is performed on initial contact (BCTABL1, 0)**
  - Upon penetration slave is constrained to master
  - As a result stresses may develop
  - For stress free initial contact is set by defining ICOORD to 1
    - BCONPRG in MSC Nastran input file or in Patran





Geometric Contact Parameters:

Property Set Name: block2_to_block3_g

Distance Tolerance(ERROR):

Bias Factor(BIAS): 0.9

Interference Closure(CINTERF): 0.0

Slide Off Distance(SLIDE): 0.0

Hard-Soft Ratio (HARDS): 2.0

☐ Glued Contact(IGLUE)

☐ Retain Gaps/Overlaps(IGLUE)

☐ Retain Moment(IGLUE)

☐ Allow Separation(JGLUE)

☑ Stressfree InitCont(ICOORD)

☐ Delayed Slide Off(ICOORD)

Select: Node To Segment

Contact Detection(ISEARCH)

○ Automatic

○ Single Sided

● Double Sided

CONTACT STRESS FREE CONTACT

- **BCONPRG – used to define ICOORD**

BCONPRG	BCGPID		PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	
	PARAM4	VAL4	PARAM5	VAL5	-etc.-				

- **Example**

BCONPRG	90		**ICOORD**	**1**					

OVERVIEW

- **What is and when to consider contact**
- **Contact Bodies** (Workshop – Rubber Door Seal)
- **Contact Pairs/Tables** (Workshop – Deformable to Rigid Contact)
- **Contact Detection**
  - Node to segment
  - Segment to segment
- **Special Features**
  - Interference (Workshop – Interference Fit)
  - Stress free initial contact
  - Glued contact (Workshop – Contact Pairs)
  - Contact with Shells
  - Friction
- **Convergence and Controls with contact**

GLUED CONTACT

- **SOL 101 and 400 support the general AND permanent glued contact capability**
- **General glued contact is a nonlinear solution**
  - 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

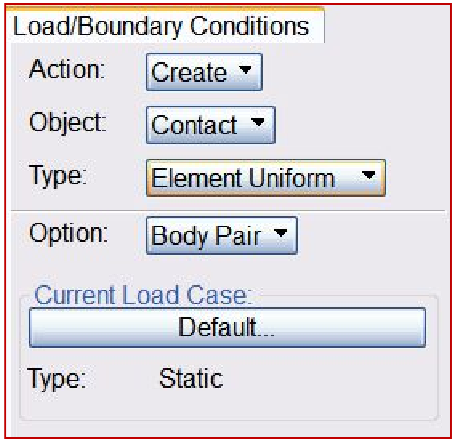
- **Permanent Glued Contact is a special case of glued contact**
  - Designed to help users quickly assemble components with dissimilar meshes
  - Bodies should be in contact initially since contact detection is performed only in the beginning of the analysis
  - Also available in SOL 103, 105, 107, 108, 109, 110, 111, 112, and 200
  - A linear solution. Permanent contact constraint MPC equations are used. No nonlinear increments or iterations involved

SETTING UP GLUED CONTACT - IGLUE

- **Glued contact is activated by IGLUE parameter in contact pair geometric properties BCONPRG**
  - **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. This option is recommended when there is no gap or overlap between contact surfaces or initial stress free contact is specified.
  - **2** - Activates a special glue option to ensure that there is no relative tangential or 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** - Ensures **full moment carrying glue** when shells contact. This option is recommended when there is no gap or overlap between contact surfaces or initial stress free contact is specified.
  - **4** - Ensures **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 IN PATRAN

- **Check Glued Contact to activate glued contact**
- **Select Retain Gaps/Overlaps to set IGLUE=2**
- **Select Retain Moment to set IGLUE=3 or 4**



Load/Boundary Conditions

Action: Create

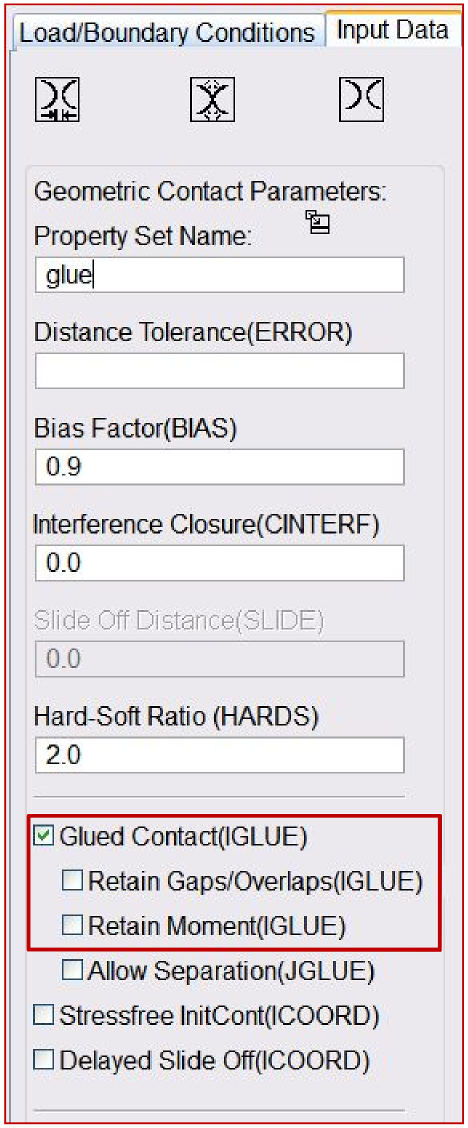
Object: Contact

Type: Element Uniform

Option: Body Pair

Current Load Case: Default...

Type: Static



Load/Boundary Conditions | Input Data

Geometric Contact Parameters:

Property Set Name: glue

Distance Tolerance(ERROR)

Bias Factor(BIAS) 0.9

Interference Closure(CINTERF) 0.0

Slide Off Distance(SLIDE) 0.0

Hard-Soft Ratio (HARDS) 2.0

☑ Glued Contact(IGLUE)

☐ Retain Gaps/Overlaps(IGLUE)

☐ Retain Moment(IGLUE)

☐ Allow Separation(JGLUE)

☐ Stressfree InitCont(ICOORD)

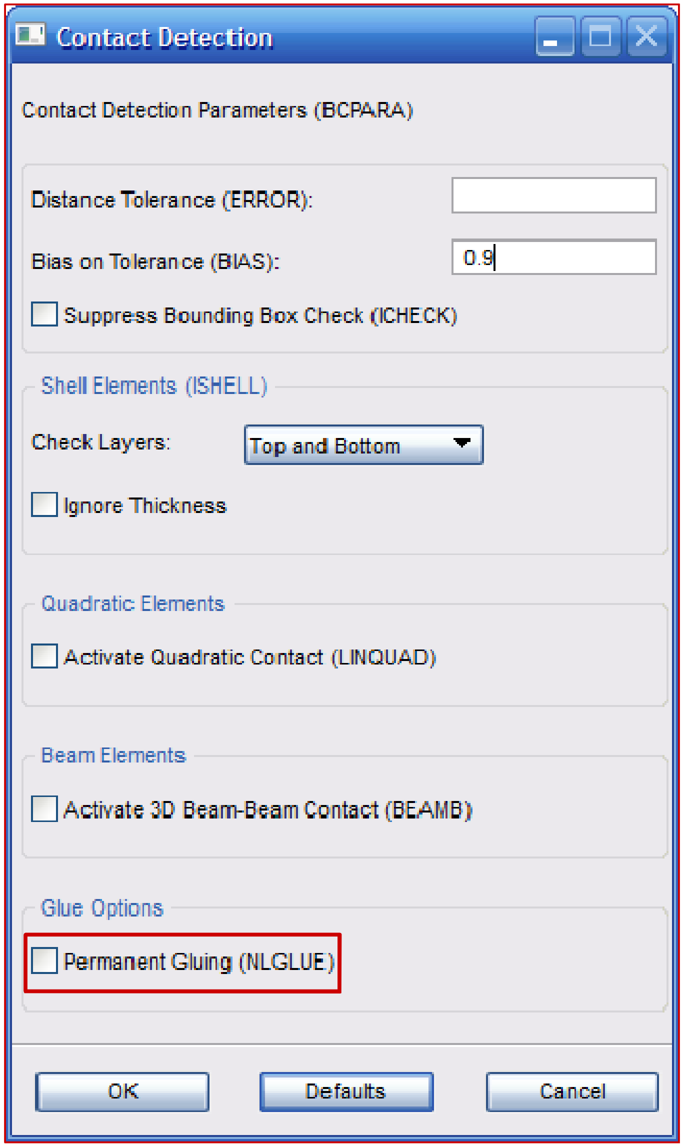
☐ Delayed Slide Off(ICOORD)

SETTING UP GLUED CONTACT - SEARCH

- **When glued contact is used, MSC Nastran will automatically create a network of MPCs connecting the bodies in contact**
- **The quality of this network does depend on the contact search order**
- **A good strategy is to use the Automatic search order when initially setting up glued contacts, and then double-check the MPCs created (discussion to follow)**

PERMANENT GLUED CONTACT

- **If all contact pairs corresponding to the first load step contain IGLUE > 0, permanent glued contact will be used in the analysis**
- **For permanent glued contact, contact status never changes, any contact pair that is not within contact tolerance initially will not be glued**
- **To allow bodies come in contact and become glued during the analysis, general glued contact must be used**
- **Set NLGLUE to 1 in BCPARA entry for general glued contact**



Contact Detection Parameters (BCPARA)

Distance Tolerance (ERROR):

Bias on Tolerance (BIAS): 0.9

☐ Suppress Bounding Box Check (ICHECK)

Shell Elements (ISHELL)

Check Layers: Top and Bottom

☐ Ignore Thickness

Quadratic Elements

☐ Activate Quadratic Contact (LINQUAD)

Beam Elements

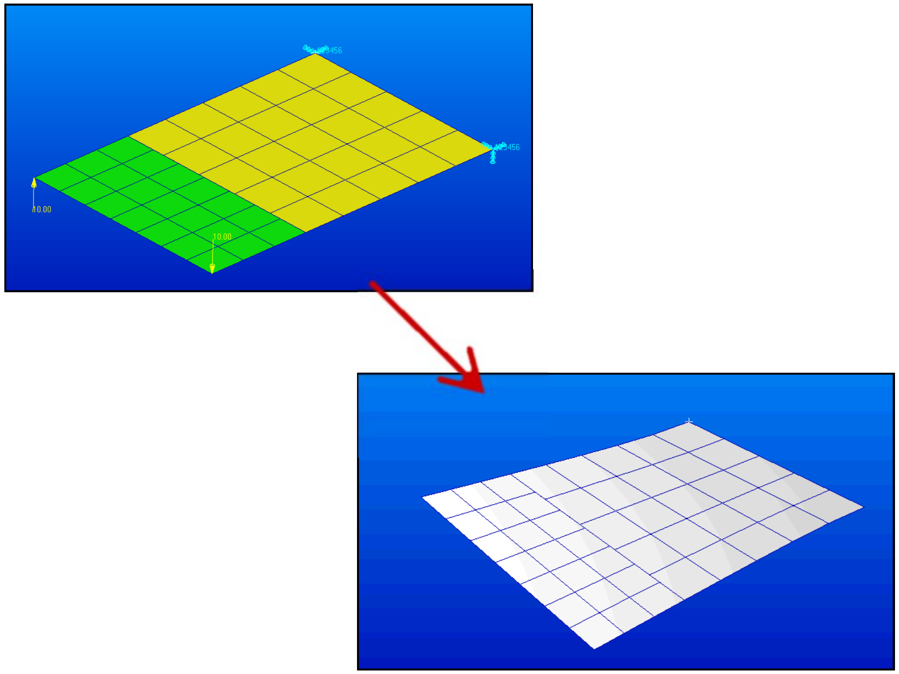
☐ Activate 3D Beam-Beam Contact (BEAMB)

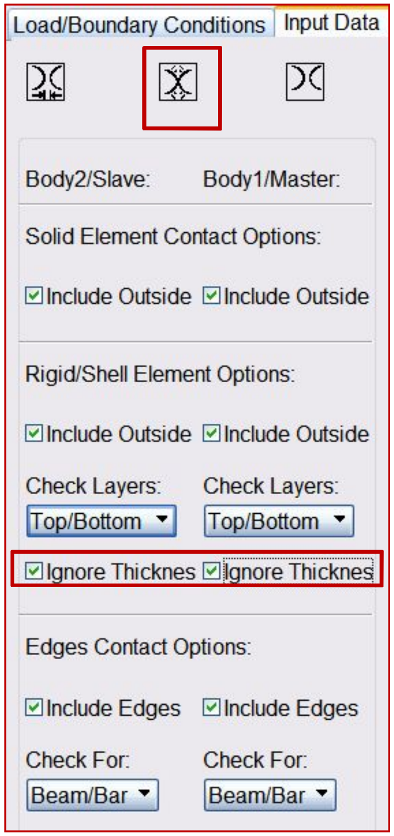
Glue Options

☐ Permanent Gluing (NLGLUE)

MOMENT CARRYING GLUED CONTACT

- For shell Edge-to-Edge or shell Edge-to-Face glued contact, it is important to turn on the Retain Moment option to enable the glued joint to transfer moments
- Set contact option to ignore shell thickness to simply glue the mid-planes of the shells





Body2/Slave: Body1/Master:

Solid Element Contact Options:

☑Include Outside ☑Include Outside

Rigid/Shell Element Options:

☑Include Outside ☑Include Outside

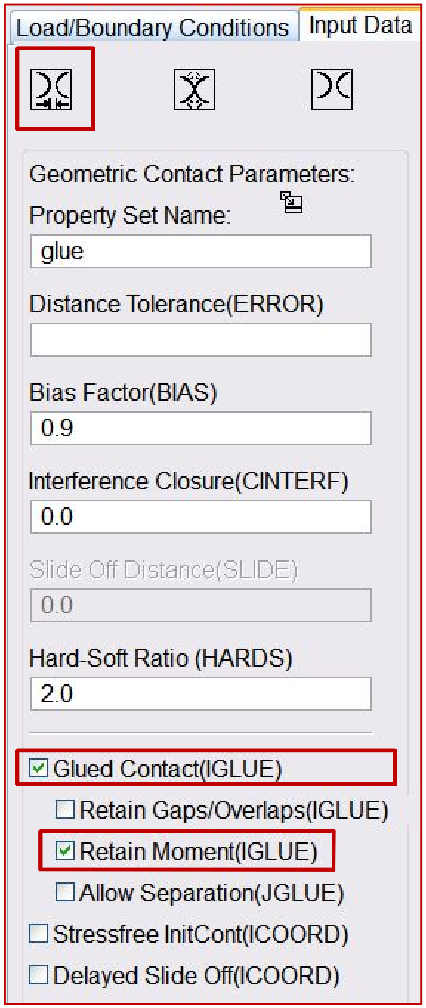
Check Layers: Top/Bottom Check Layers: Top/Bottom

☑Ignore Thicknes ☑Ignore Thicknes

Edges Contact Options:

☑Include Edges ☑Include Edges

Check For: Beam/Bar Check For: Beam/Bar



Geometric Contact Parameters:

Property Set Name: glue

Distance Tolerance(ERROR)

Bias Factor(BIAS) 0.9

Interference Closure(CINTERF) 0.0

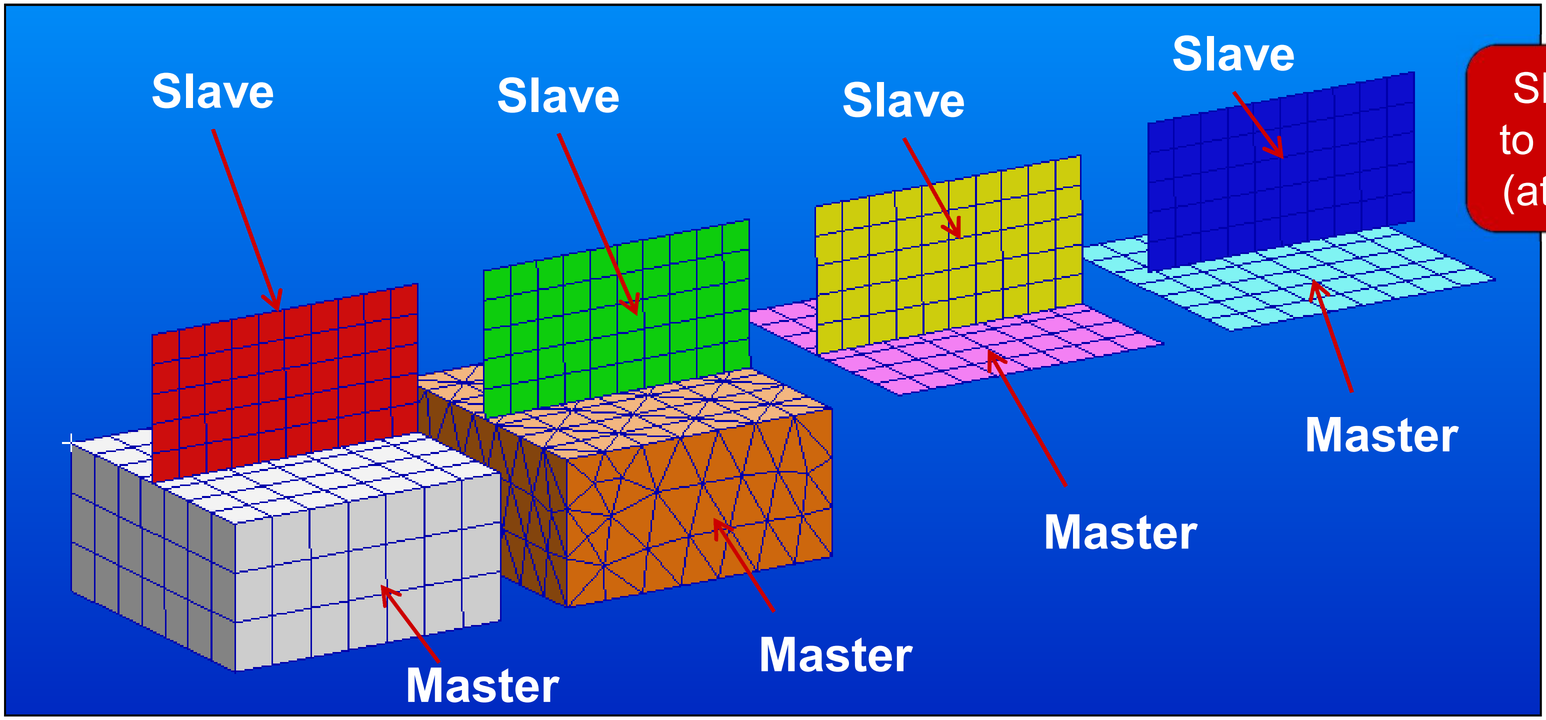
Slide Off Distance(SLIDE) 0.0

Hard-Soft Ratio (HARDS) 2.0

☑Glued Contact(IGLUE)
☐Retain Gaps/Overlaps(IGLUE)
☑Retain Moment(IGLUE)
☐Allow Separation(JGLUE)
☐Stressfree InitCont(ICOORD)
☐Delayed Slide Off(ICOORD)

MOMENT CARRYING GLUED CONTACT

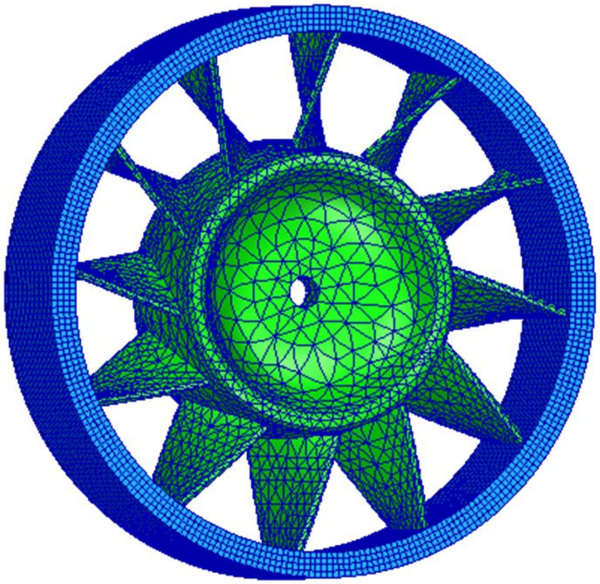
- **For shell Edge-to-Face glued contact, slave and master choices are important**
  - The shell edge body needs to be the slave.
  - The face body needs to be the master.



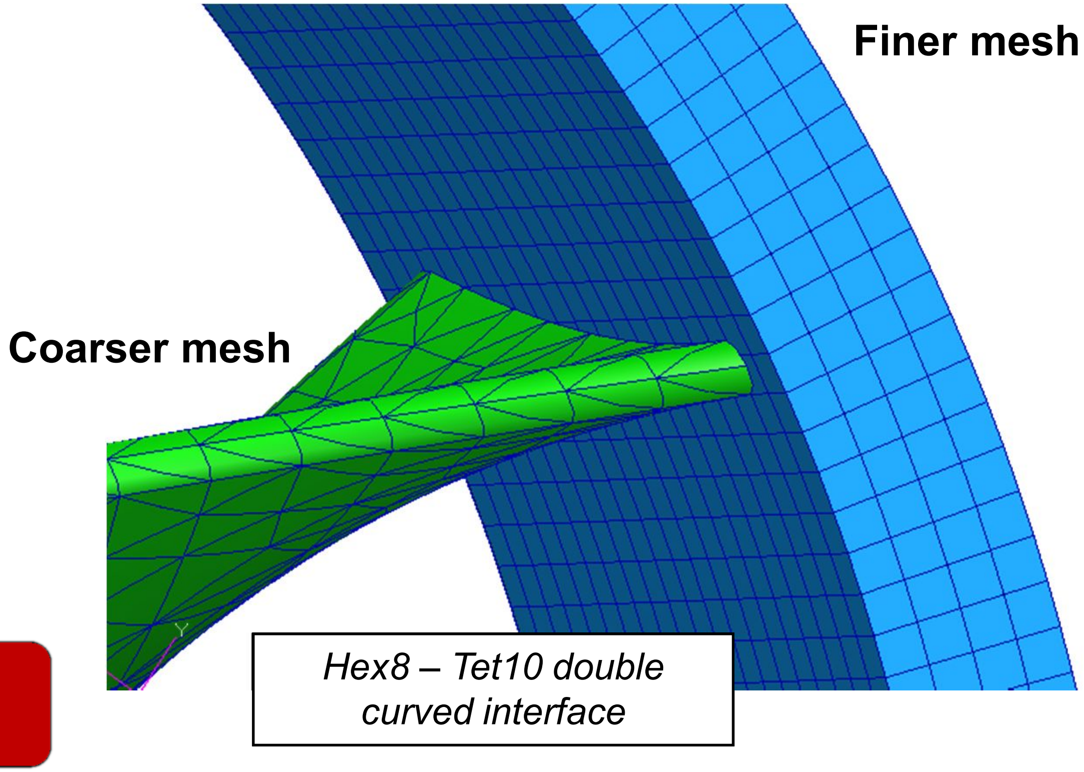
Shell edge to shell face (at $^t/_2$ away)

CASE STUDY: MODAL ANALYSIS WITH GLUED CONTACT

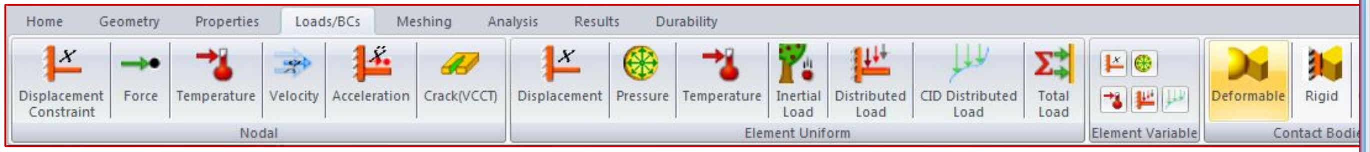
- **Analyze the shrouded vanes shown below**
  - This is the MSC Nastran Demonstration Problems Manual Example 25
  - Hub and vanes are meshed with Tet10 elements
  - Shroud is meshed with Hex8 elements
  - Glue the two bodies together and compute the first 10 free-free modes



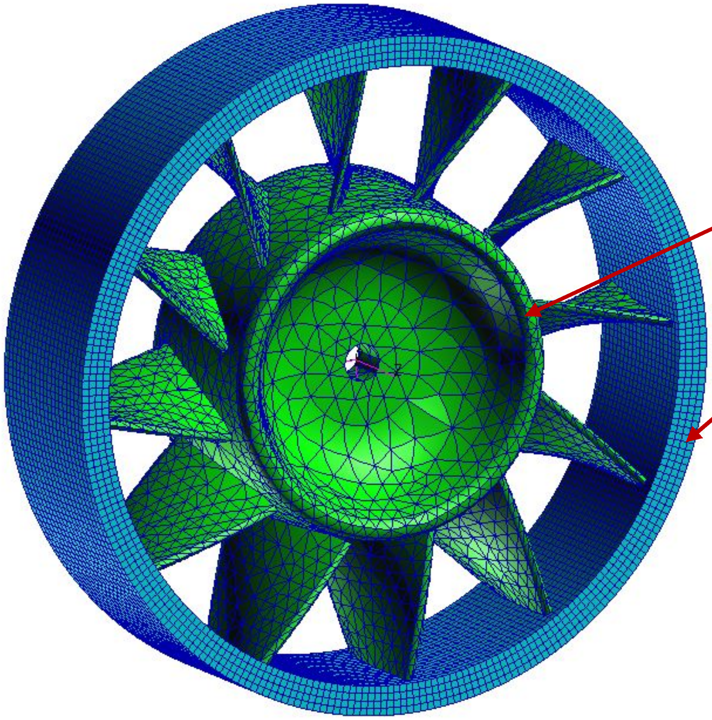
Play Case study video:
NAS133_S5_cs_glued_contact

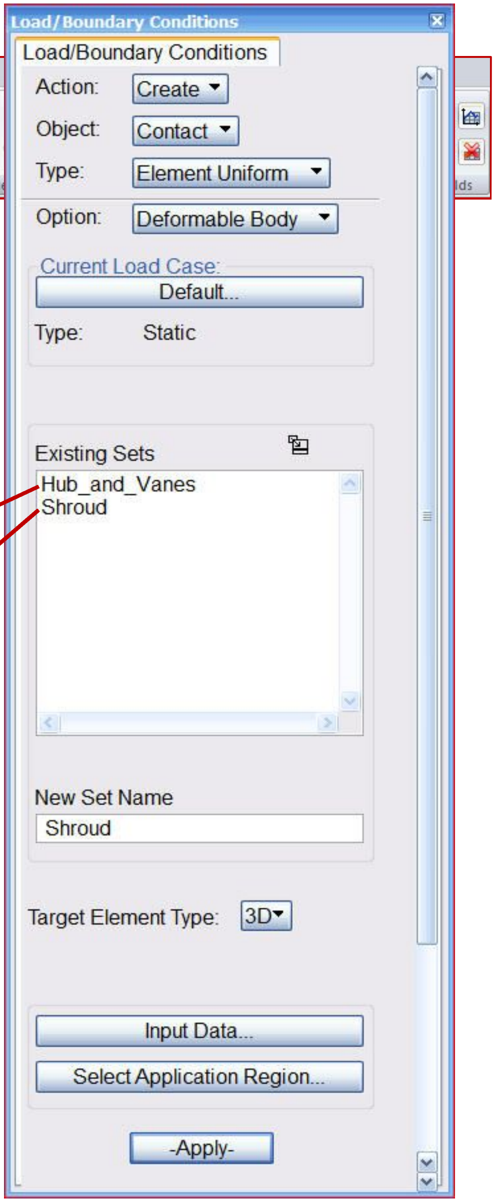


CASE STUDY: MODAL ANALYSIS WITH GLUED CONTACT



- **Import the MSC Nastran input file Vanes_and_Shroud.dat**
- **Create two contact bodies**





Load/Boundary Conditions

Action: Create

Object: Contact

Type: Element Uniform

Option: Deformable Body

Current Load Case: Default...

Type: Static

Existing Sets

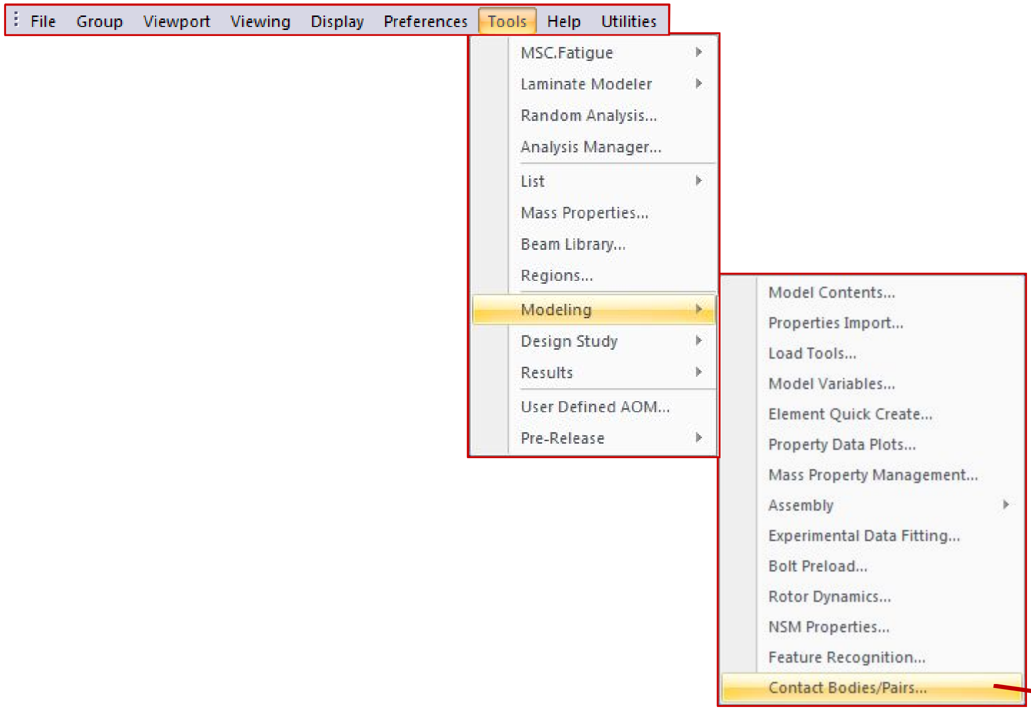
Hub_and_Vanes
Shroud

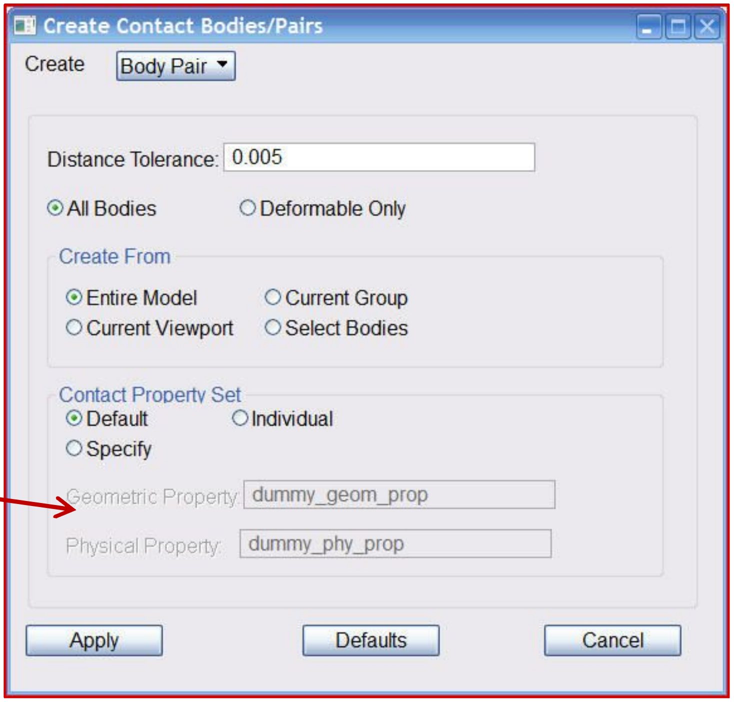
New Set Name: Shroud

Target Element Type: 3D

CASE STUDY: MODAL ANALYSIS WITH GLUED CONTACT

- **Define Contact Pair using Automatic feature in Patran.**





Create: Body Pair

Distance Tolerance: 0.005

● All Bodies ○ Deformable Only

Create From

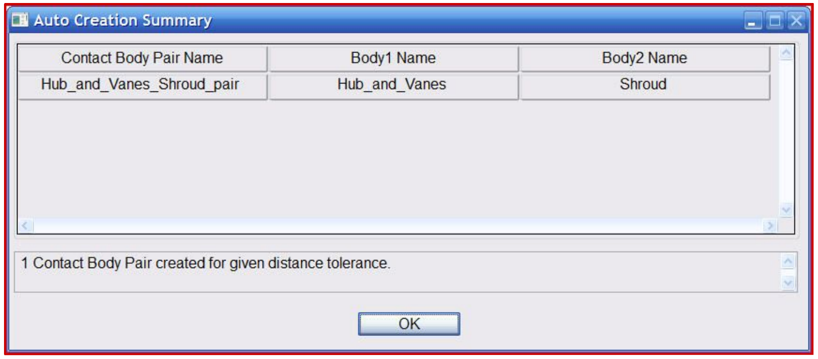
● Entire Model ○ Current Group
○ Current Viewport ○ Select Bodies

Contact Property Set

● Default ○ Individual
○ Specify

Geometric Property: dummy_geom_prop

Physical Property: dummy_phy_prop

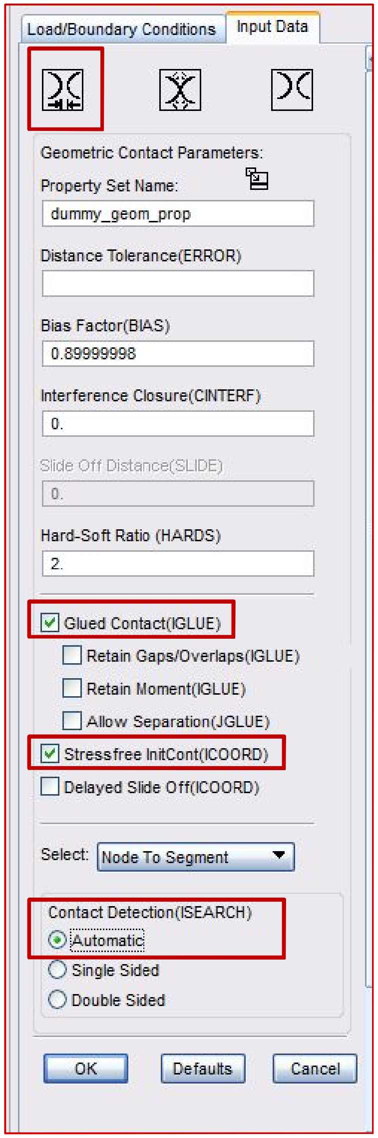


Contact Body Pair Name	Body1 Name	Body2 Name
Hub_and_Vanes_Shroud_pair	Hub_and_Vanes	Shroud

1 Contact Body Pair created for given distance tolerance.

CASE STUDY: MODAL ANALYSIS WITH GLUED CONTACT

- **Check IGLUE**
- **Turn on Stress-Free Initial Contact (ICOORD = 1). This specifies stress-free initial contact which modifies the coordinates of the nodes in contact to close gaps and penetrations between the two bodies.**
- **Turn on Automatic Contact Detection (ISEARCH = 2).**
  - This is an important step to improve clean rigid-body modes and help the model pass the grounding check (discussed later). It also ensures no artificial stresses are induced.



Geometric Contact Parameters:

Property Set Name: dummy_geom_prop

Distance Tolerance(ERROR)

Bias Factor(BIAS) 0.89999998

Interference Closure(CINTERF) 0.

Slide Off Distance(SLIDE) 0.

Hard-Soft Ratio (HARDS) 2.

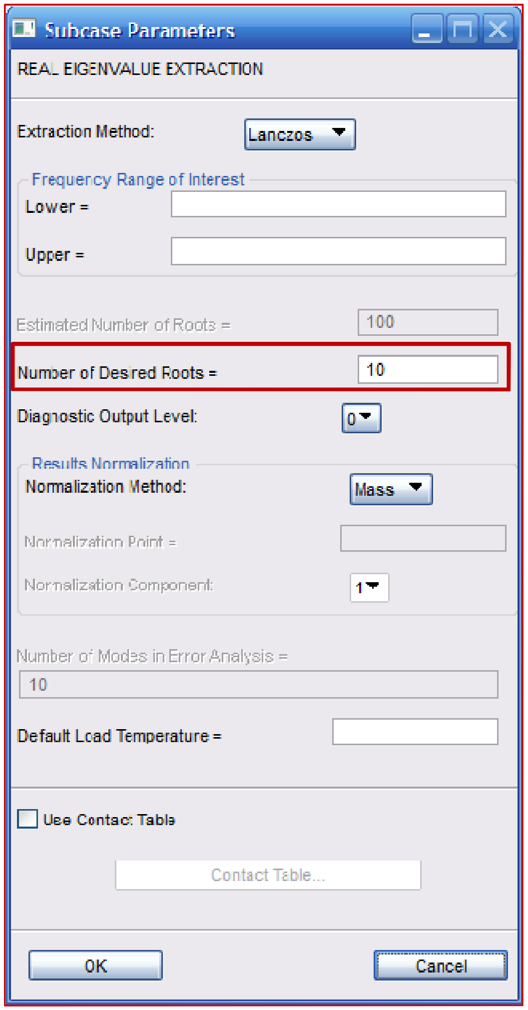
☑ Glued Contact(IGLUE)
☐ Retain Gaps/Overlaps(IGLUE)
☐ Retain Moment(IGLUE)
☐ Allow Separation(JGLUE)
☑ Stressfree InitCont(ICOORD)
☐ Delayed Slide Off(ICOORD)

Select: Node To Segment

Contact Detection(ISEARCH)
● Automatic
○ Single Sided
○ Double Sided

CASE STUDY: MODAL ANALYSIS WITH GLUED CONTACT

- **Set up a SOL 103 normal modes analysis and request the first 10 modes.**



REAL EIGENVALUE EXTRACTION

Extraction Method: Lanczos

Frequency Range of Interest

Lower =

Upper =

Estimated Number of Roots = 100

Number of Desired Roots = 10

Diagnostic Output Level: 0

Results Normalization

Normalization Method: Mass

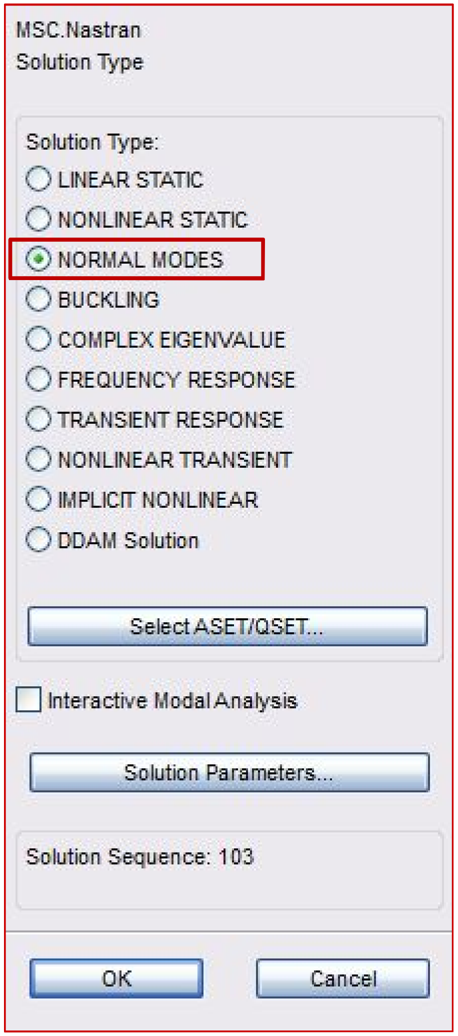
Normalization Point =

Normalization Component: 1

Number of Modes in Error Analysis = 10

Default Load Temperature =

☐ Use Contact Table


MSC.Nastran
Solution Type

Solution Type:
○ LINEAR STATIC
○ NONLINEAR STATIC
● NORMAL MODES
○ BUCKLING
○ COMPLEX EIGENVALUE
○ FREQUENCY RESPONSE
○ TRANSIENT RESPONSE
○ NONLINEAR TRANSIENT
○ IMPLICIT NONLINEAR
○ DDAM Solution

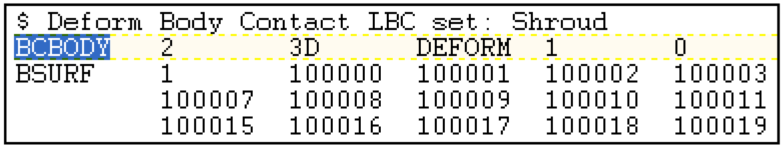
☐ Interactive Modal Analysis

Solution Sequence: 103

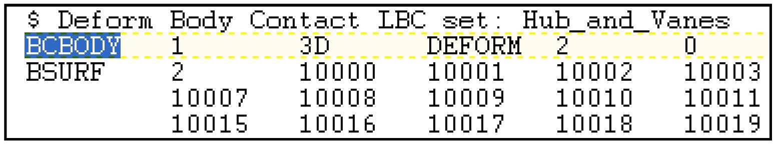
CASE STUDY: MODAL ANALYSIS WITH GLUED CONTACT

- **Review the BCONPRG entry.**
  - IGLUE=1
  - ICOORD=1
  - ISEARCH=2

```
BEGIN BULK
PARAM    POST    0
PARAM   PRTMAXIM YES
EIGRL    1                       10      0
BCTABL1  0       8004
BCONECT  8004    3003            2       1
BCONPRG  3003            ICOORD  1       IGLUE   1       ISEARCH 2
BCTABL1  1       8004
```



```
$ Deform Body Contact LBC set: Shroud
BCBODY   2       3D      DEFORM  1       0
BSURF    1       100000  100001  100002  100003
         100007  100008  100009  100010  100011
         100015  100016  100017  100018  100019
```

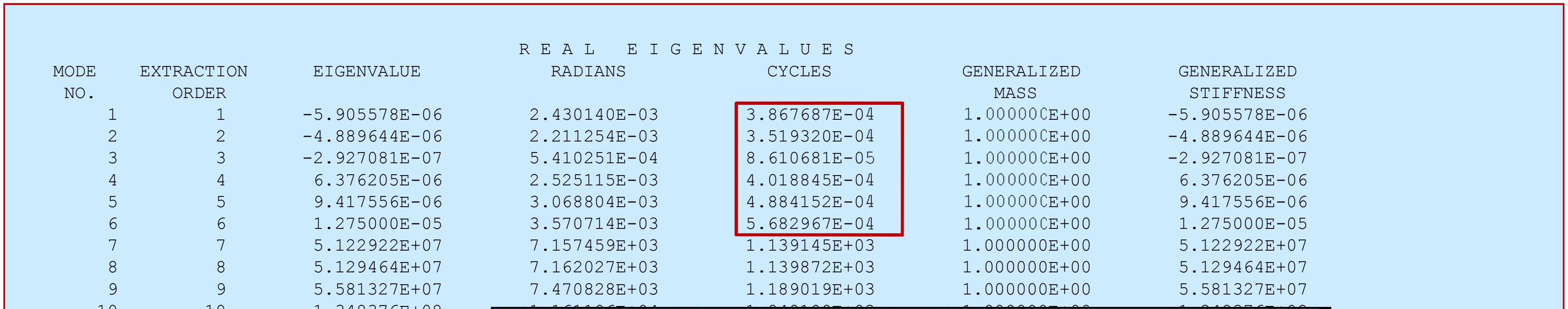


```
$ Deform Body Contact LBC set: Hub_and_Vanes
BCBODY   1       3D      DEFORM  2       0
BSURF    2       10000   10001   10002   10003
         10007   10008   10009   10010   10011
         10015   10016   10017   10018   10019
```

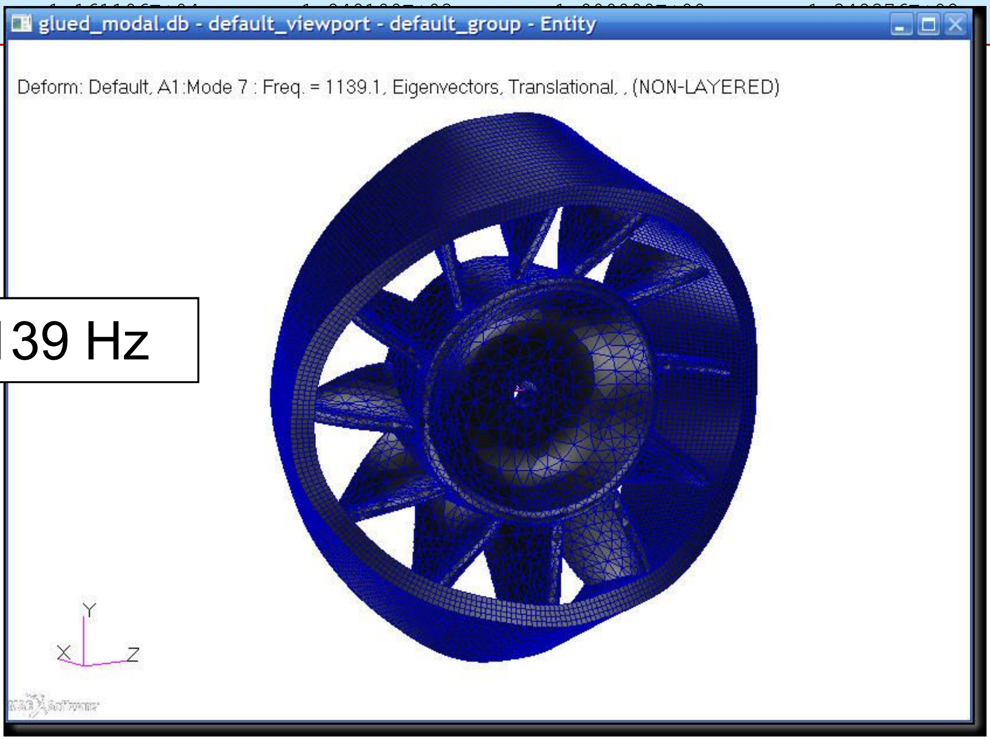
- **Important: BIAS is not specified, which defaults to 0.0 for glued contact. A Patran entry for BIAS will be ignored for glued contact.**

CASE STUDY: MODAL ANALYSIS WITH GLUED CONTACT

- **Review normal modes results:**
  - 6 clean rigid-body modes



```
                                              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        -5.905578E-06        2.430140E-03        3.867687E-04        1.000000E+00       -5.905578E-06
        2         2        -4.889644E-06        2.211254E-03        3.519320E-04        1.000000E+00       -4.889644E-06
        3         3        -2.927081E-07        5.410251E-04        8.610681E-05        1.000000E+00       -2.927081E-07
        4         4         6.376205E-06        2.525115E-03        4.018845E-04        1.000000E+00        6.376205E-06
        5         5         9.417556E-06        3.068804E-03        4.884152E-04        1.000000E+00        9.417556E-06
        6         6         1.275000E-05        3.570714E-03        5.682967E-04        1.000000E+00        1.275000E-05
        7         7         5.122922E+07        7.157459E+03        1.139145E+03        1.000000E+00        5.122922E+07
        8         8         5.129464E+07        7.162027E+03        1.139872E+03        1.000000E+00        5.129464E+07
        9         9         5.581327E+07        7.470828E+03        1.189019E+03        1.000000E+00        5.581327E+07
       10        10         1.348376E+08
```

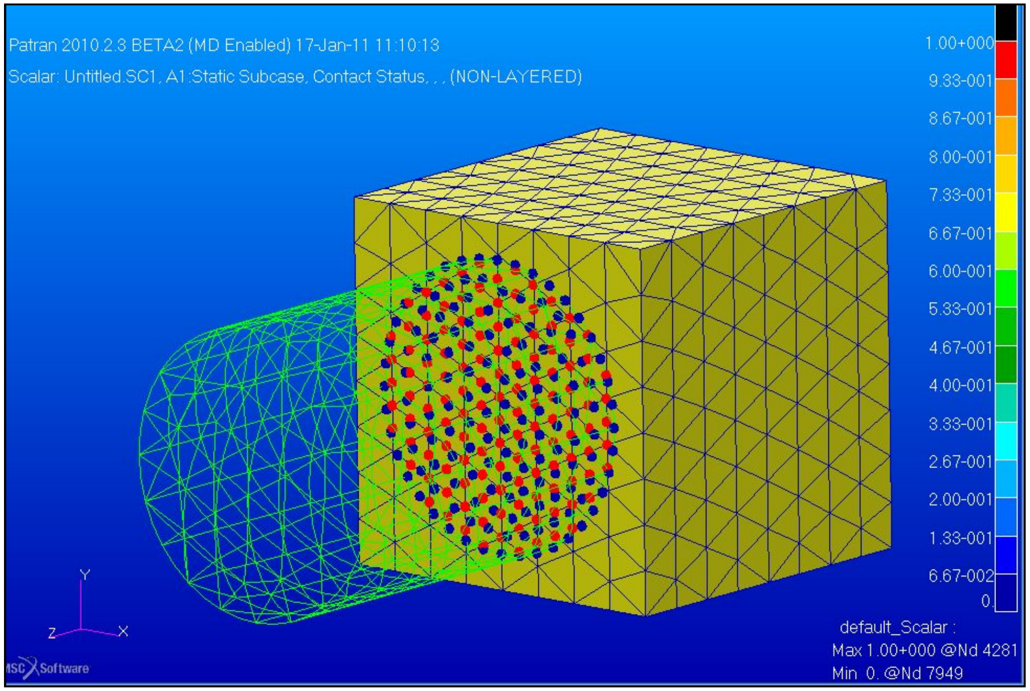


Deform: Default, A1:Mode 7 : Freq. = 1139.1, Eigenvectors, Translational, , (NON-LAYERED)

1st flexible mode 1,139 Hz

GLUED CONTACT – STATUS

- **Verifying the glued contact status is important. It is possible to glue only a few nodes on a large surface. This leads to incorrect deflections and stresses**
- **Glued contact can be evaluated by examining the deflected shapes and stresses of the model**
  - Glued contact can be visualized by making contact status marker plots



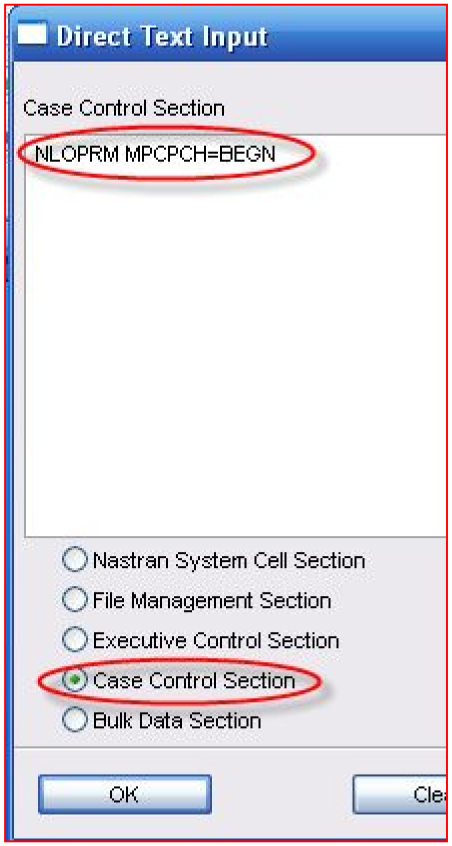
Patran 2010.2.3 BETA2 (MD Enabled) 17-Jan-11 11:10:13

Scalar: Untitled.SC1, A1:Static Subcase, Contact Status, , , (NON-LAYERED)

default_Scalar :
Max 1.00+000 @Nd 4281
Min 0. @Nd 7949

GLUED CONTACT – MPC EQUATIONS

- **MSC Nastran internally generates MPC equations to represent the glued contact**
- **The MPCs generated to model glued contact can be written during the analysis in a punch (*.pch) file**
- **These MPCs can then be directly imported into the Patran database to provide a visual reference to the glued contact**
- **The MPC punch file is created with the following Case Control Command:**
  - NLOPRM MPCPCH = BEGN



Case Control Section

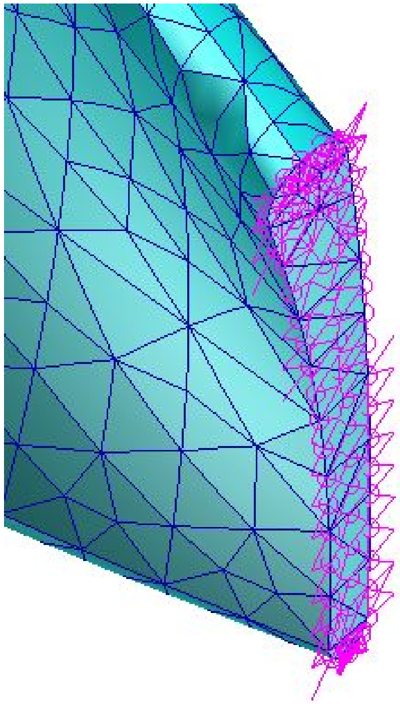
NLOPRM MPCPCH=BEGN

○ Nastran System Cell Section
○ File Management Section
○ Executive Control Section
● Case Control Section
○ Bulk Data Section

GLUED CONTACT – MPC DISPLAY

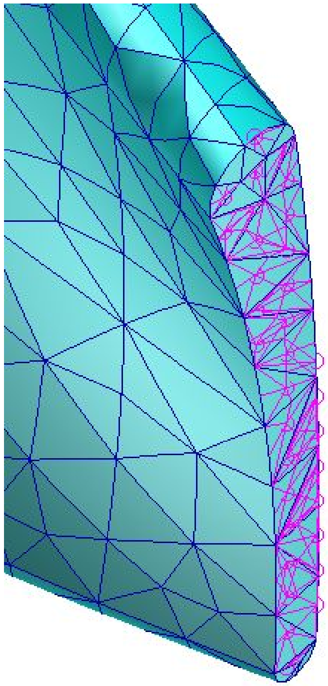
- **Confirm network of MPCs connecting contact bodies**
- **Use UNDO to remove MPCs from model, or delete them manually**

Vane from CASE STUDY 1



MPCs with Automatic Search

Details are discussed in the Contact Results Output section



MPCs with Default Search

GLUED CONTACT GROUNDING ISSUES

- **What is Grounding?**
  - If an MSC Nastran model is properly created, and it is completely unconstrained, movement at one point of the model should cause the entire model to move as a rigid body, with no internal forces. If this is not the case, the model has Grounding errors.
  - Grounding most often occurs with the misuse of CELAS and CBUSH elements, RBEs, and MPCs
  - The GROUNDCHECK command can be used in MSC Nastran to help determine in any such problems exist.
  - Glued contacts are covered by the GROUNDCHECK N-Set Output.

GLUED CONTACT GROUNDING ISSUES

- **While glued contacts may work perfectly in a model, their use can still cause failures to be flagged via GROUNDCHECK**
- **IGLUE=1 or 3 will cause grounding if:**
  - There are gaps or overlaps between the contact surfaces (they are not perfectly aligned)
  - And permanent glued contact is used
  - And stress-free initial contact (ICOORD = 1) is not specified
- **The above scenario is a common occurrence in real world models**

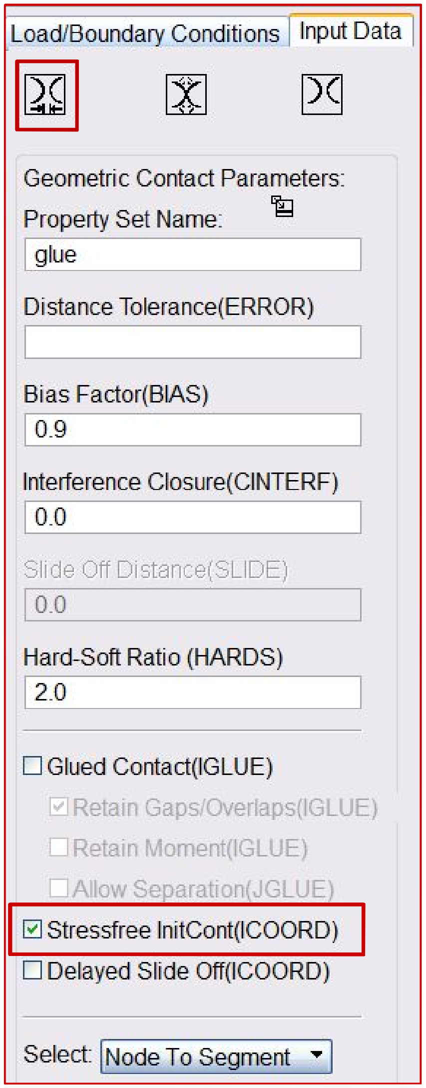
WHY DOES IGLUE CAUSE GROUNDING?

- **Glued contact creates a network of MPCs rigidly linking the contact bodies together**
  - IGLUE = 1 will create a network with 3 MPCs per slave node, linking the 3 Translational DOF.
  - IGLUE = 3 will create a network with 6 MPCs per slave node, linking the 3 Translational and 3 Rotational DOFs.
  - The MPC coefficients for these options will not take the geometry change caused by a gap or overlap properly into account, leading to “unbalanced” MPCs and a grounding problem.

HOW TO PREVENT GROUNDING?

- **To prevent a grounding problem you can:**
  - Retain initial gaps or overlaps between the contact bodies
    - Check **Retain Gaps/Overlaps**
      - To use IGLUE = 2
    - Check **Retain Gaps/Overlaps** and **Retain Moment**
      - To use IGLUE = 4
  - Check **Stress-free Initial Contact**
    - To use ICOORD = 1 which maintains stress-free initial contact.

Use only one of these two options. Don't use both at the same time.



Property Set Name: glue

Distance Tolerance(ERROR):

Bias Factor(BIAS): 0.9

Interference Closure(CINTERF): 0.0

Slide Off Distance(SLIDE): 0.0

Hard-Soft Ratio (HARDS): 2.0

☐ Glued Contact(IGLUE)

☑ Retain Gaps/Overlaps(IGLUE)

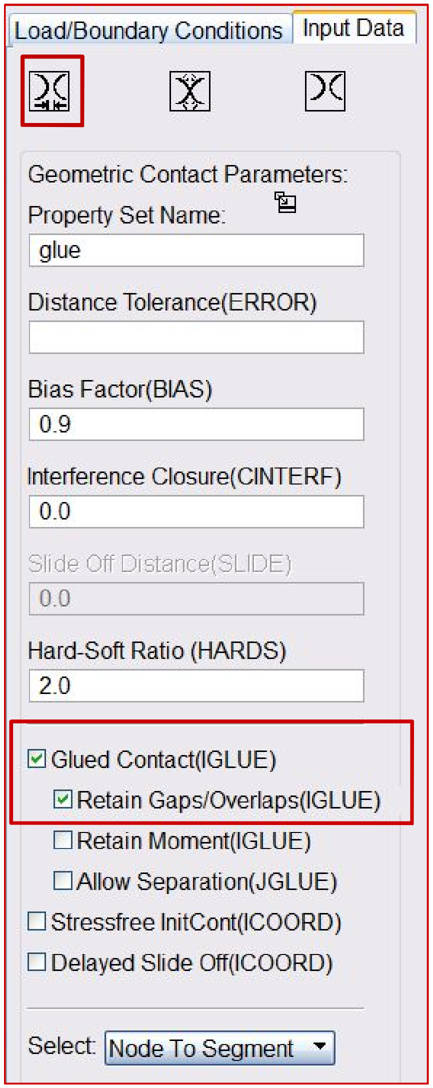
☐ Retain Moment(IGLUE)

☐ Allow Separation(JGLUE)

☑ Stressfree InitCont(ICOORD)

☐ Delayed Slide Off(ICOORD)

Select: Node To Segment



Property Set Name: glue

Distance Tolerance(ERROR):

Bias Factor(BIAS): 0.9

Interference Closure(CINTERF): 0.0

Slide Off Distance(SLIDE): 0.0

Hard-Soft Ratio (HARDS): 2.0

☑ Glued Contact(IGLUE)

☑ Retain Gaps/Overlaps(IGLUE)

☐ Retain Moment(IGLUE)

☐ Allow Separation(JGLUE)

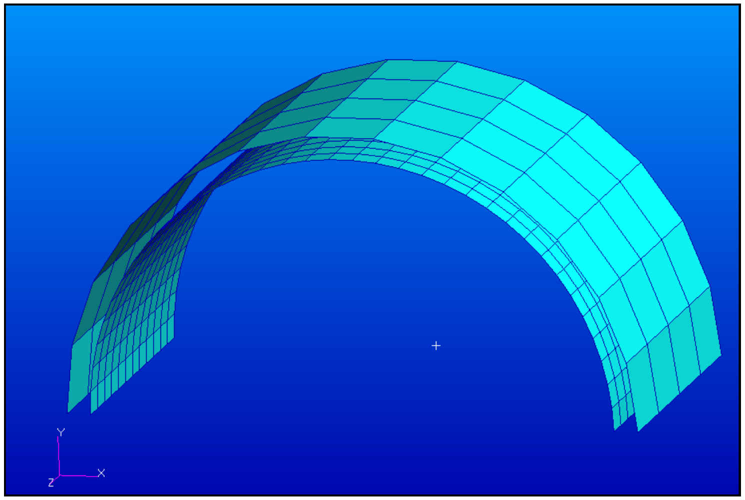
☐ Stressfree InitCont(ICOORD)

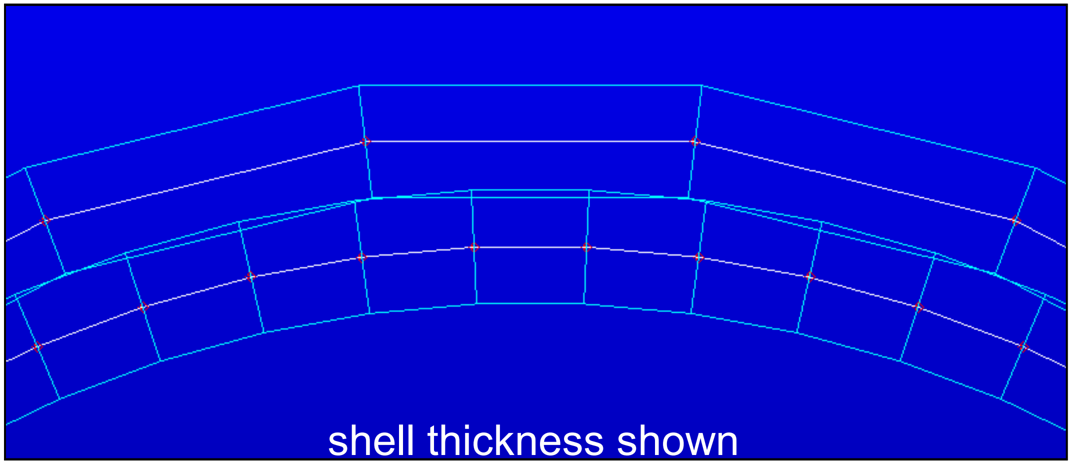
☐ Delayed Slide Off(ICOORD)

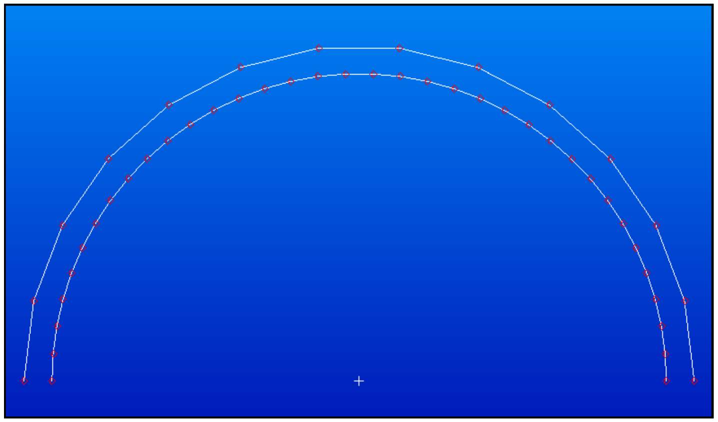
Select: Node To Segment

GROUNDING EXAMPLE 1

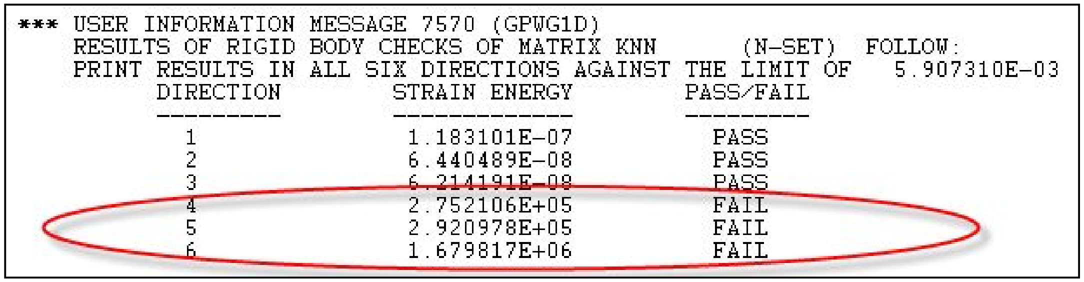
- **Faceted face glued contact**
- **Overlaps exist between the two contacting shell bodies**




shell thickness shown



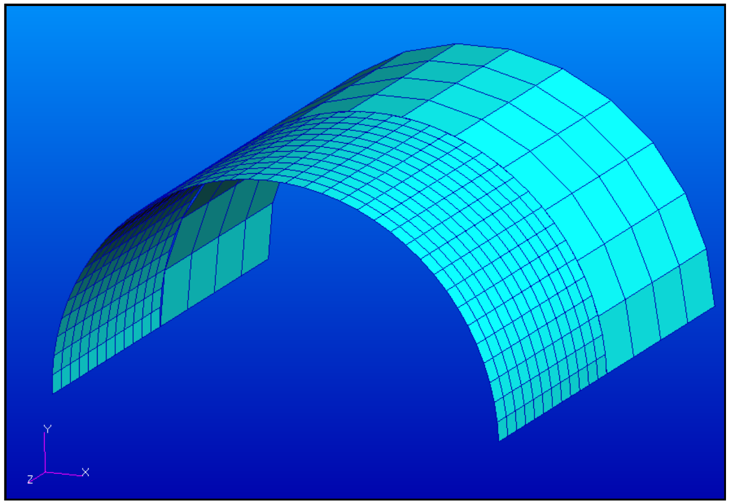
GROUNDCHECK results with IGLUE=1

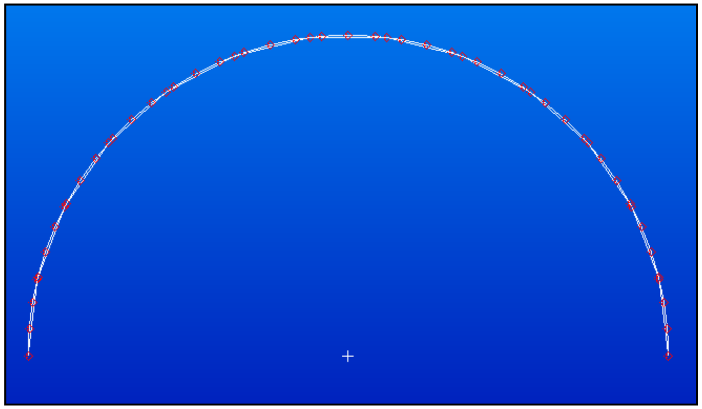


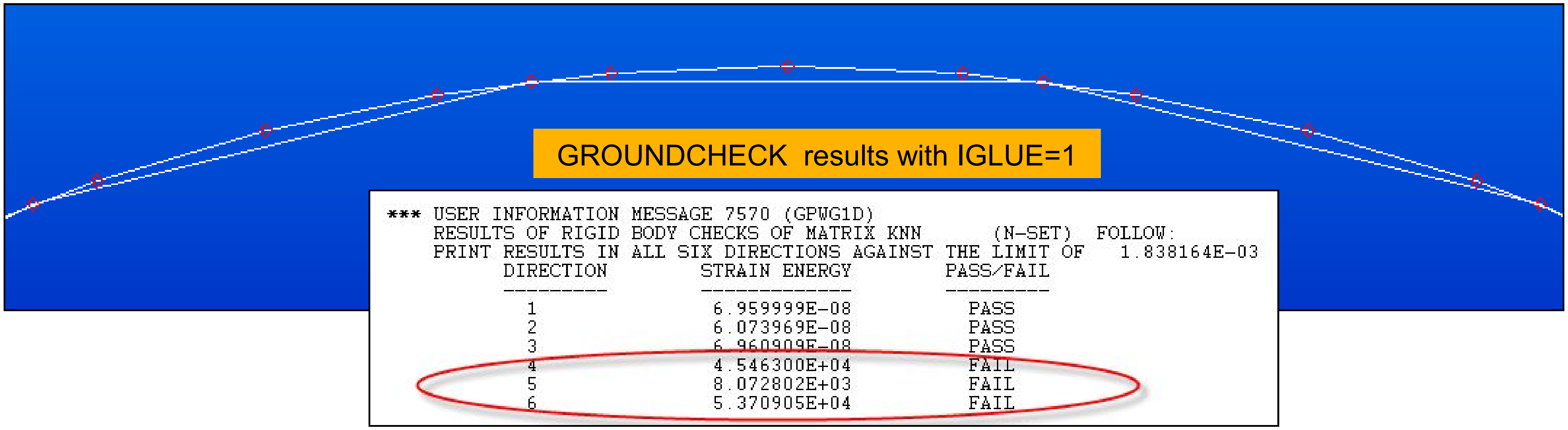
```
*** USER INFORMATION MESSAGE 7570 (GPWG1D)
    RESULTS OF RIGID BODY CHECKS OF MATRIX KNN       (N-SET)  FOLLOW:
    PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF    5.907310E-03
          DIRECTION          STRAIN ENERGY          PASS/FAIL
          ---------          -------------          ---------
              1               1.183101E-07            PASS
              2               6.440489E-08            PASS
              3               6.214191E-08            PASS
              4               2.752106E+05            FAIL
              5               2.920978E+05            FAIL
              6               1.679817E+06            FAIL
```

GROUNDING EXAMPLE 2

- Faceted edge glued contact
- Gaps exist between the two contacting shell bodies



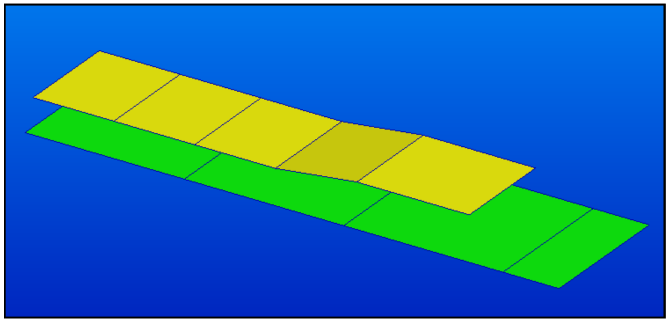


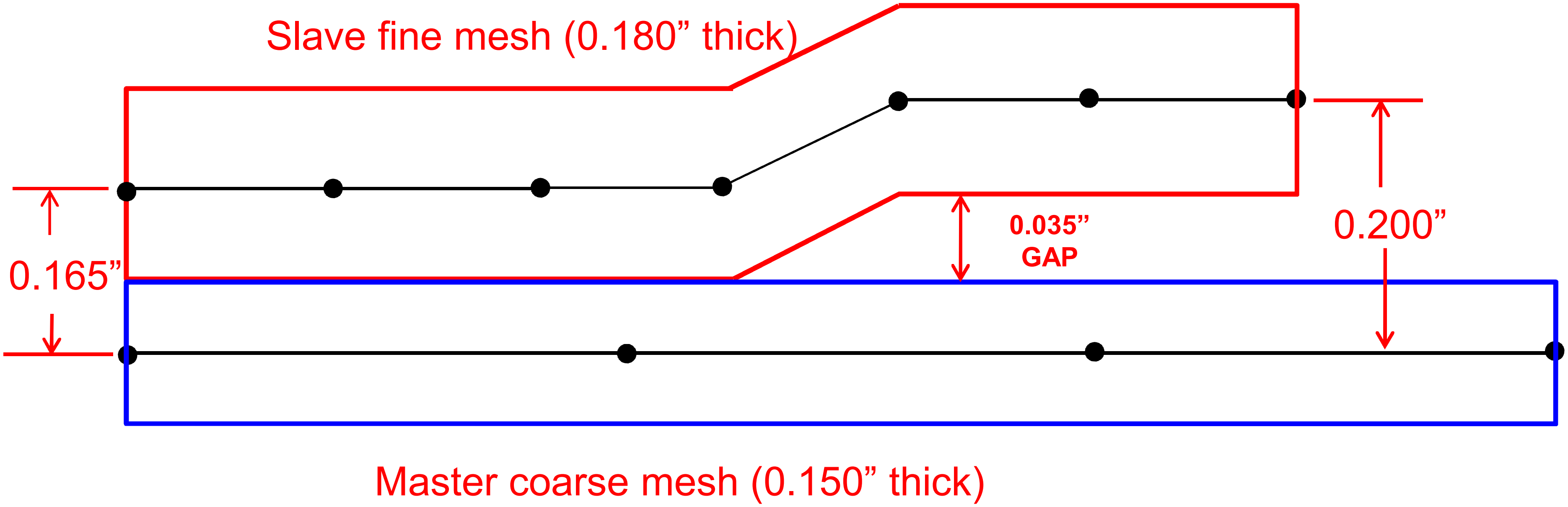


```
*** USER INFORMATION MESSAGE 7570 (GPWG1D)
    RESULTS OF RIGID BODY CHECKS OF MATRIX KNN      (N-SET)  FOLLOW:
    PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF   1.838164E-03
          DIRECTION         STRAIN ENERGY         PASS/FAIL
          ---------         -------------         ---------
              1              6.959999E-08           PASS
              2              6.073969E-08           PASS
              3              6.960909E-08           PASS
              4              4.546300E+04           FAIL
              5              8.072802E+03           FAIL
              6              5.370905E+04           FAIL
```

GROUNDING TEST MODEL – IGLUE = 1

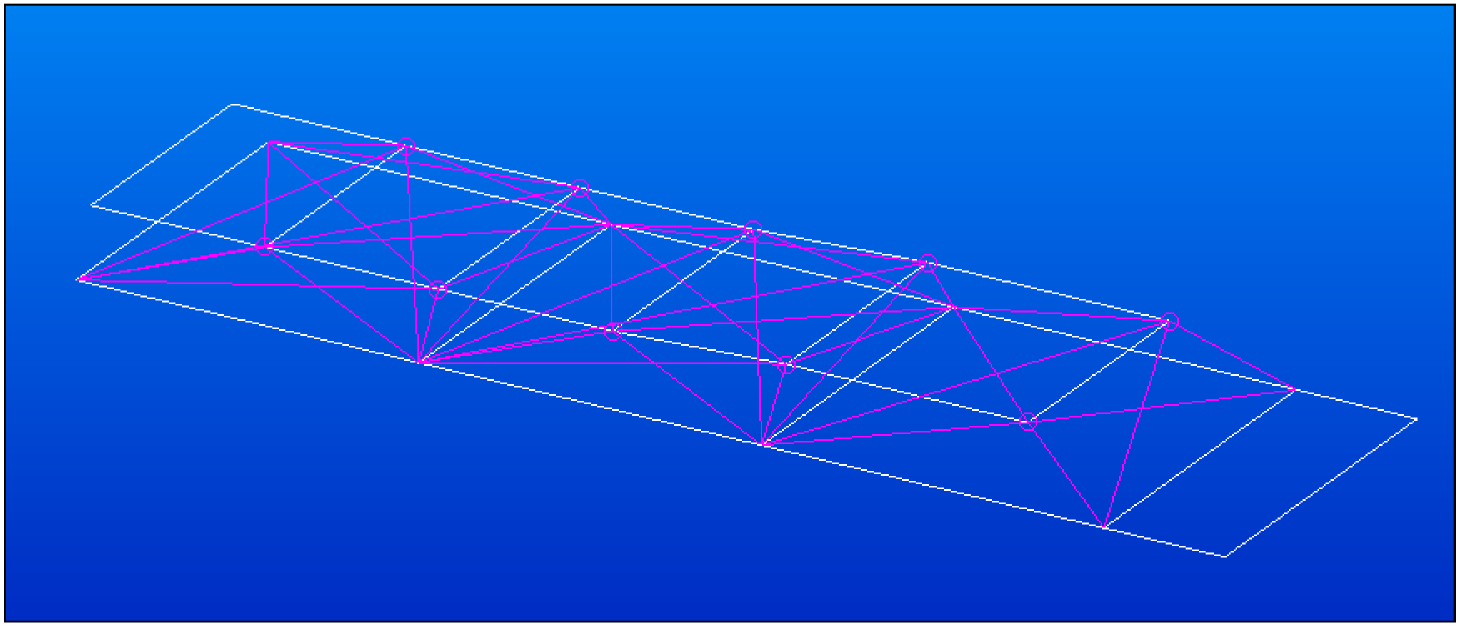
- **This model has a gap between the two shell bodies**
  - 0.180”/2 + 0.150”/2 = 0.165”
  - 0.200” – 0.165” = 0.035”
  - Contact tolerance is 0.0359”
  - As contact tolerance is greater than the gap, nodes will find contact and be glued

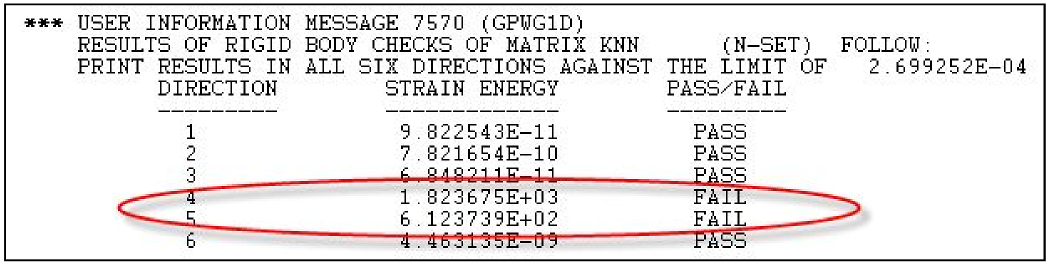




TEST MODEL – IGLUE = 1

- **Glued contact MPCs shown in Patran**
- **Grounding check shows the model is grounded**

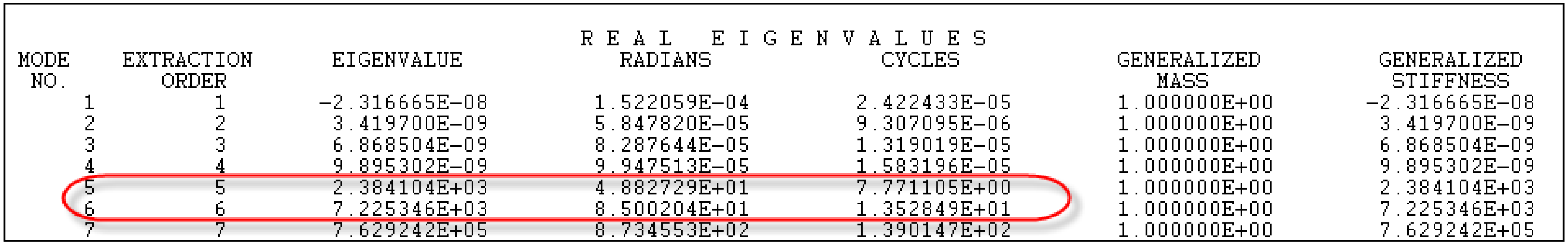




```
*** USER INFORMATION MESSAGE 7570 (GPWG1D)
    RESULTS OF RIGID BODY CHECKS OF MATRIX KNN      (N-SET)  FOLLOW:
    PRINT RESULTS IN ALL SIX DIRECTIONS AGAINST THE LIMIT OF   2.699252E-04
          DIRECTION          STRAIN ENERGY         PASS/FAIL
          ---------          -------------         ---------
            1                 9.822543E-11           PASS
            2                 7.821654E-10           PASS
            3                 6.848211E-11           PASS
            4                 1.823675E+03           FAIL
            5                 6.123739E+02           FAIL
            6                 4.463135E-09           PASS
```

TEST MODEL – IGLUE = 1

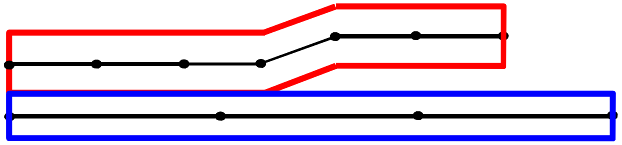
- **Perform a free-free modal check**
  - Run a Normal Modes analysis with all boundary conditions removed
  - The model does not have 6 clean rigid body modes

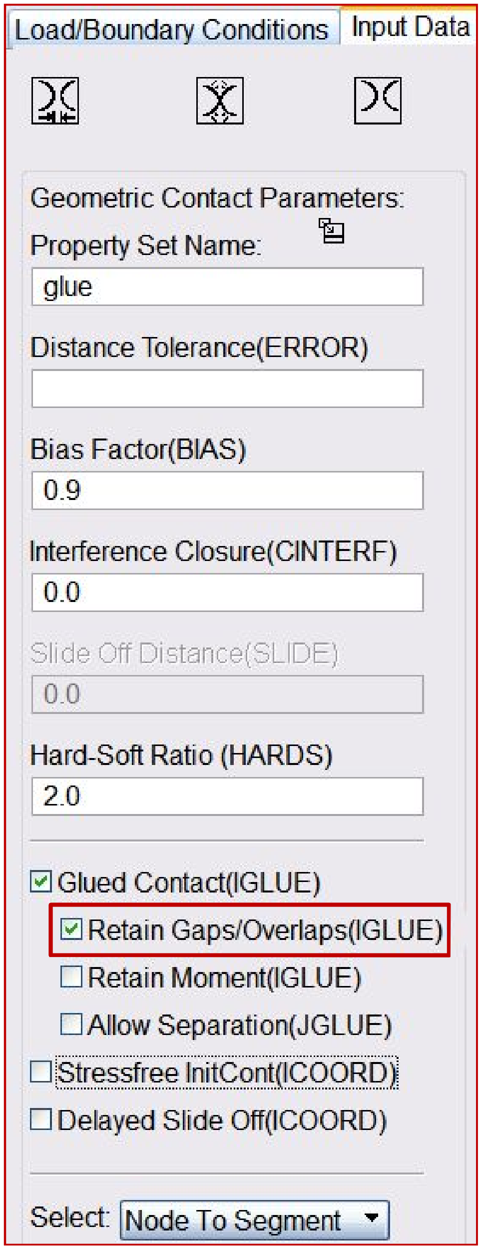


```
                                              R E A L   E I G E N V A L U E S
MODE     EXTRACTION       EIGENVALUE            RADIANS             CYCLES            GENERALIZED         GENERALIZED
 NO.        ORDER                                                                         MASS              STIFFNESS
    1          1        -2.316665E-08        1.522059E-04        2.422433E-05        1.000000E+00       -2.316665E-08
    2          2         3.419700E-09        5.847820E-05        9.307095E-06        1.000000E+00        3.419700E-09
    3          3         6.868504E-09        8.287644E-05        1.319019E-05        1.000000E+00        6.868504E-09
    4          4         9.895302E-09        9.947513E-05        1.583196E-05        1.000000E+00        9.895302E-09
    5          5         2.384104E+03        4.882729E+01        7.771105E+00        1.000000E+00        2.384104E+03
    6          6         7.225346E+03        8.500204E+01        1.352849E+01        1.000000E+00        7.225346E+03
    7          7         7.629242E+05        8.734553E+02        1.390147E+02        1.000000E+00        7.629242E+05
```

TEST MODEL – IGLUE = 2

- **Turn on Retain Gaps/Overlaps option**
- **Slave nodes will not be projected onto master surface**
- **No grounding**





Geometric Contact Parameters:

Property Set Name: glue

Distance Tolerance(ERROR)

Bias Factor(BIAS) 0.9

Interference Closure(CINTERF) 0.0

Slide Off Distance(SLIDE) 0.0

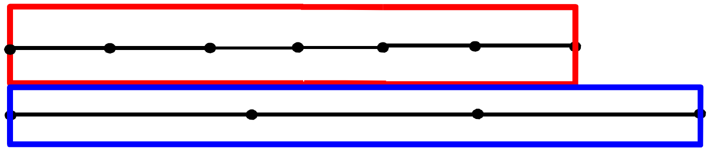
Hard-Soft Ratio (HARDS) 2.0

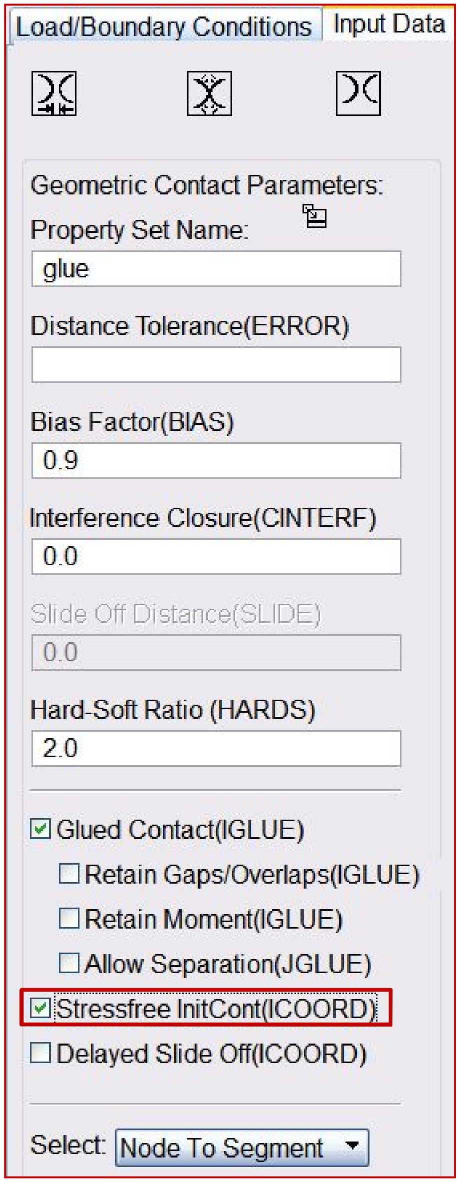
☑ Glued Contact(IGLUE)
  ☑ Retain Gaps/Overlaps(IGLUE)
  ☐ Retain Moment(IGLUE)
  ☐ Allow Separation(JGLUE)
☐ Stressfree InitCont(ICOORD)
☐ Delayed Slide Off(ICOORD)

Select: Node To Segment

USING STRESS-FREE INITIAL CONTACT

- **If stress-free initial contact (ICOORD=1) is used, no grounding**
- **The user needs to be aware that the model geometry has changed during initial contact detection**





Geometric Contact Parameters:

Property Set Name: glue

Distance Tolerance(ERROR)

Bias Factor(BIAS) 0.9

Interference Closure(CINTERF) 0.0

Slide Off Distance(SLIDE) 0.0

Hard-Soft Ratio (HARDS) 2.0

☑ Glued Contact(IGLUE)

  ☐ Retain Gaps/Overlaps(IGLUE)

  ☐ Retain Moment(IGLUE)

  ☐ Allow Separation(JGLUE)

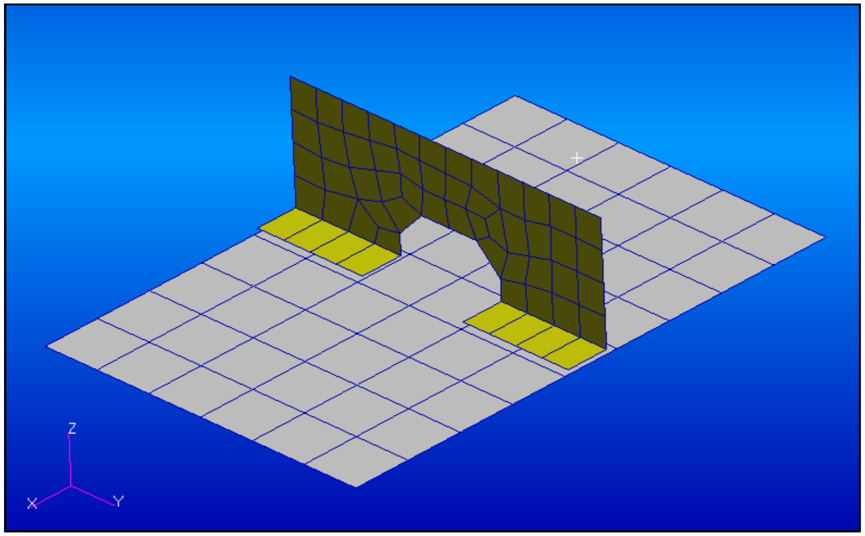
☑ Stressfree InitCont(ICOORD)

☐ Delayed Slide Off(ICOORD)

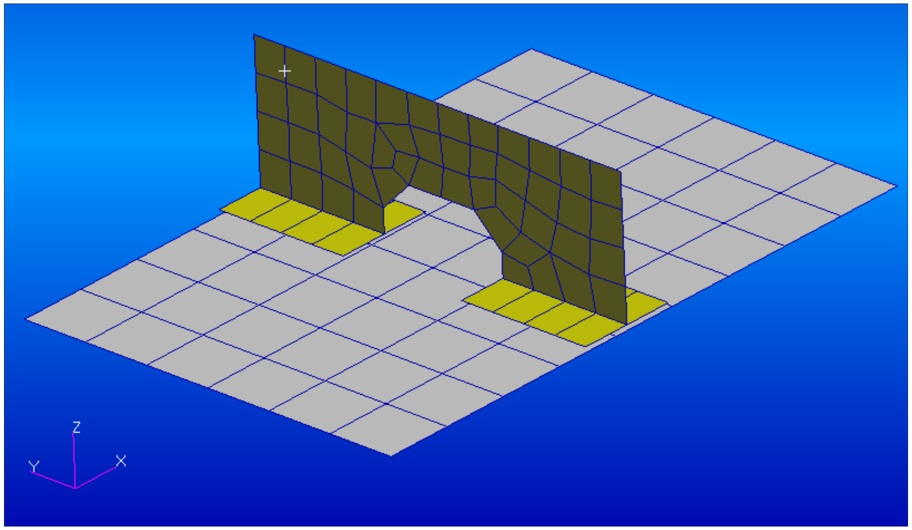
Select: Node To Segment

GROUNDING – ANGLE AND TEE CLIPS

- **A special case of grounding involves Angle or Tee shear clips, with or without mouse holes**
- **When the faces of the flanges are glued to the skin beneath, grounding issues and missing contact detection can occur at the junction of the clip flange and web**
- **Solution is to not include the web in the contact body created, only use the flange**



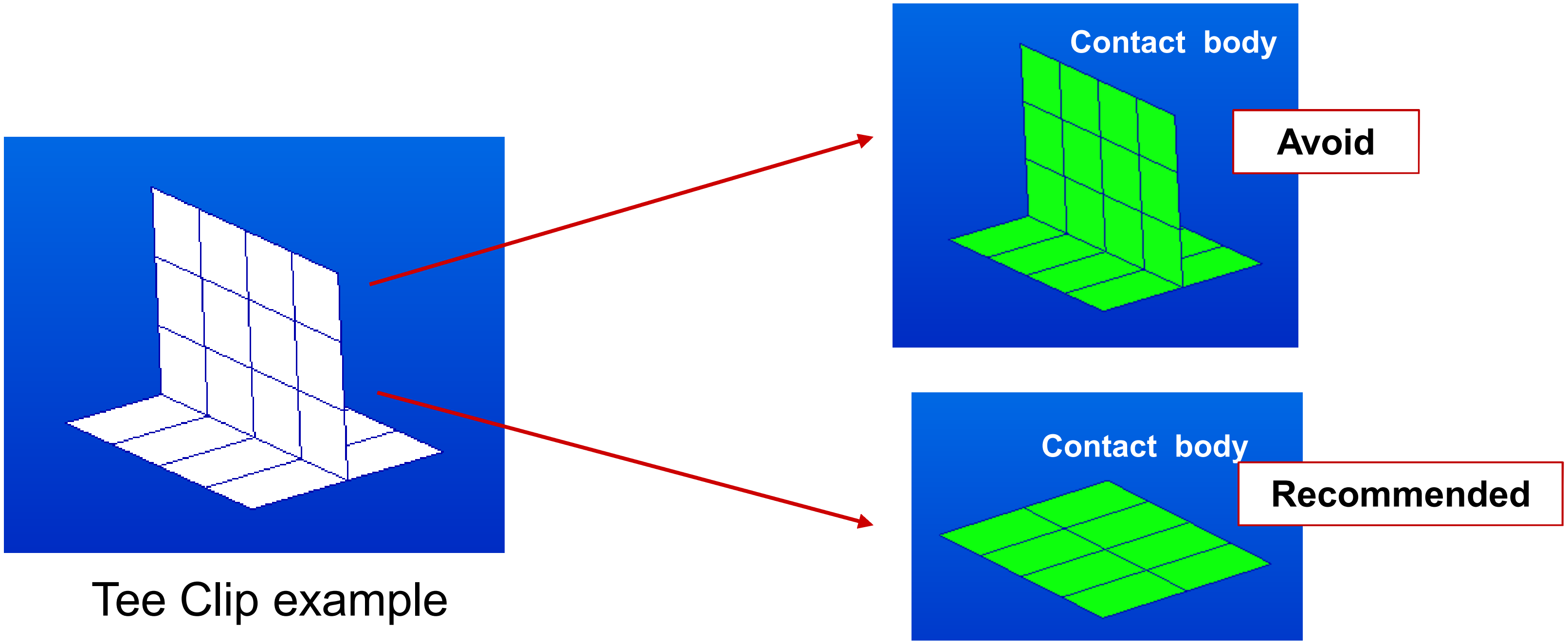
Angle clip



Tee clip

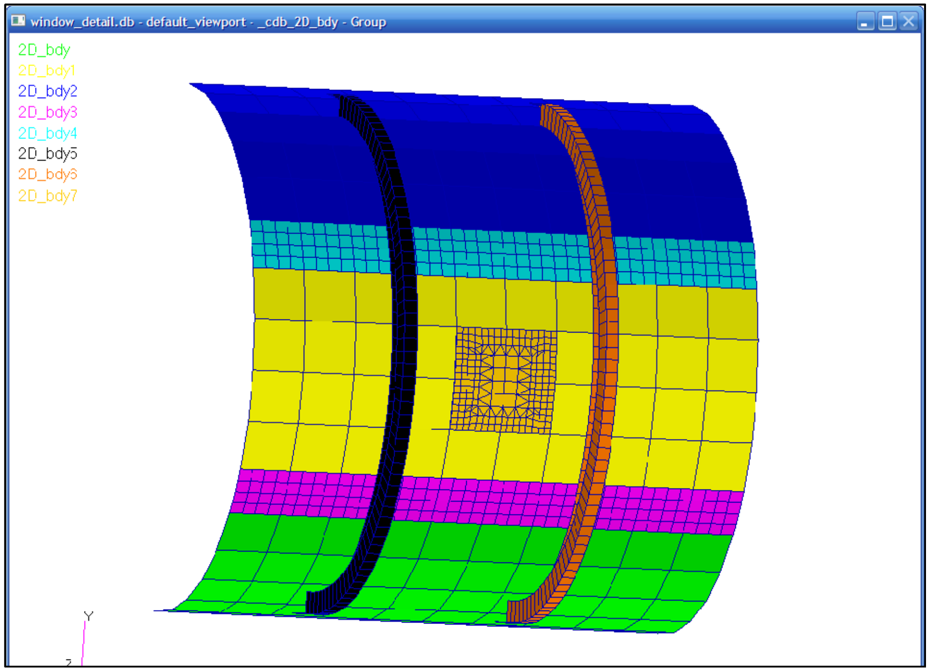
GROUNDING – ANGLE AND TEE CLIPS

- **Recommended practice**
  - Do not create a contact body which contains both the web and flanges, i.e., avoid corners and junctions in a contact body
  - When corners or junctions are included in a contact body, the contact detection behavior can be unexpected



Tee Clip example

WORKSHOP 5 – CONTACT PAIRS



- **Workshop Objectives**
  - Include a fine mesh locally in a coarser-meshed model.
  - Use Patran’s Automatic Contact Body and Body Pair detection to detect and create multiple contact bodies.
  - Use the Distance Tolerance (ERROR) to achieve glued contact between bodies with gaps.

OVERVIEW

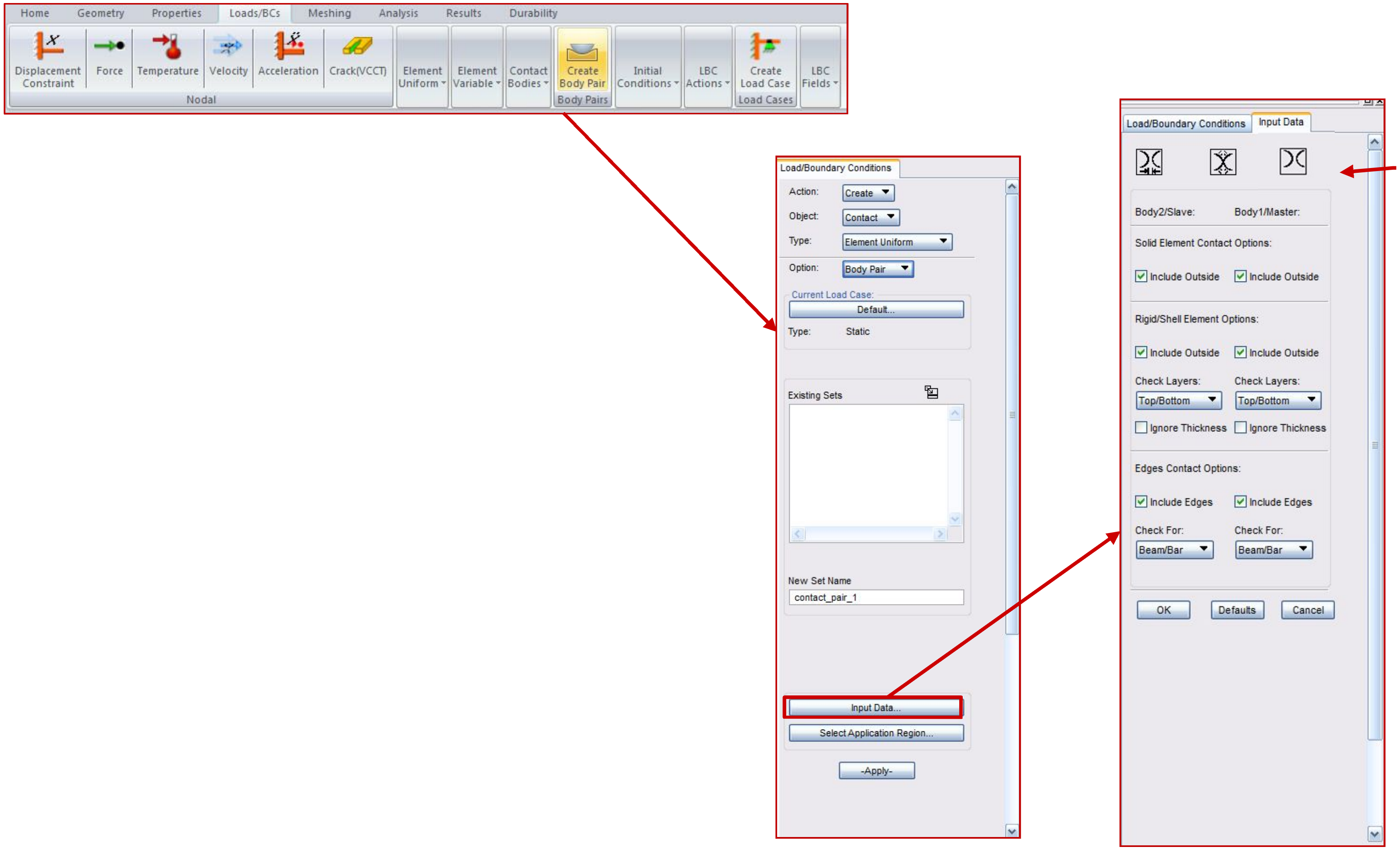
- What is and when to consider contact
- Contact Bodies (Workshop – Rubber Door Seal)
- Contact Pairs/Tables (Workshop – Deformable to Rigid Contact)
- Contact Detection
  - Node to segment
  - Segment to segment
- **Special Features**
  - Interference (Workshop – Interference Fit)
  - Stress free initial contact
  - Glued contact (Workshop – Contact Pairs)
  - Contact with Shells
  - Friction
- Convergence and Controls with contact

CONTACT DETECTION FOR SHELLS

- **Shells can have contact at their bottom, top, and mid-plane**
  - By default, contact is checked for top and bottom
  - Patran allows user specification, see next page
- **In order to define other scenarios the parameter, COPT is introduced**
- **This parameter has additional functions (see following pages). In the future it will define which different element classes contribute to one single BCBODY**
  - At this time a BCBODY may consist only of one element class, for example quadratic shells

CONTACT DETECTION FOR SHELLS

- **Patran allows specification of what shell surface should be considered for contact**



Action: Create
Object: Contact
Type: Element Uniform
Option: Body Pair
Current Load Case: Default...
Type: Static
Existing Sets
New Set Name contact_pair_1

Body2/Slave: | Body1/Master:

Solid Element Contact Options:
☑ Include Outside ☑ Include Outside

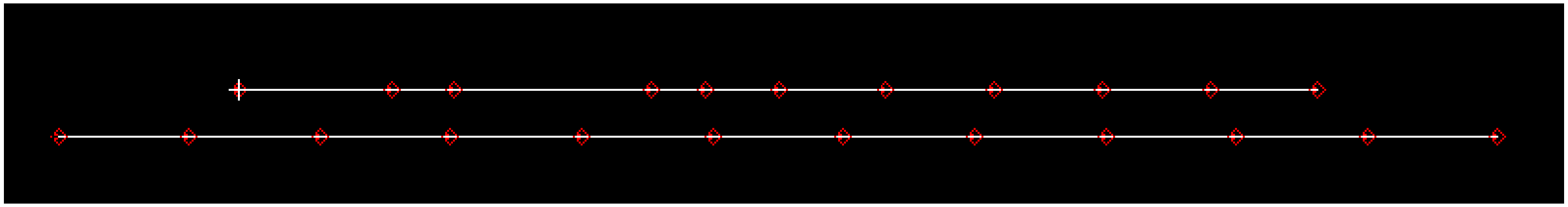
Rigid/Shell Element Options:
☑ Include Outside ☑ Include Outside
Check Layers: Top/Bottom | Check Layers: Top/Bottom
☐ Ignore Thickness ☐ Ignore Thickness

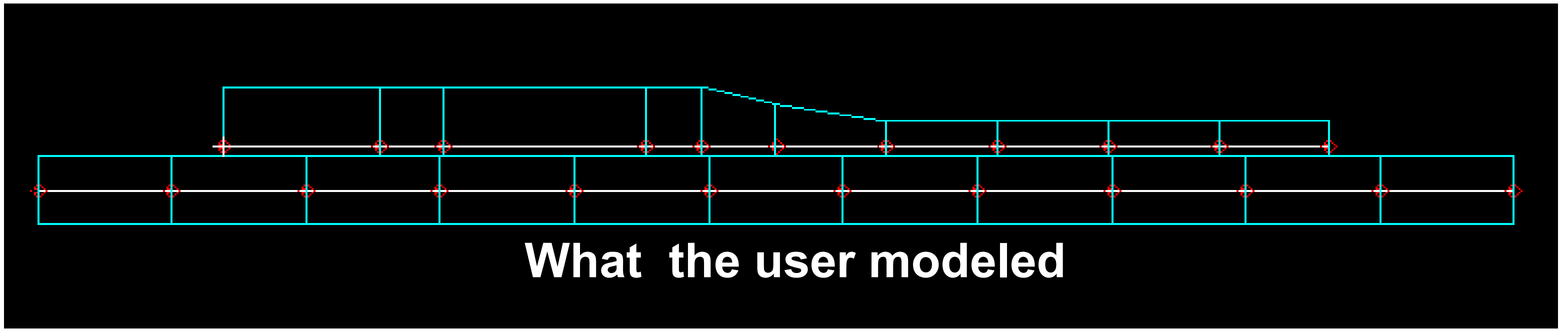
Edges Contact Options:
☑ Include Edges ☑ Include Edges
Check For: Beam/Bar | Check For: Beam/Bar

Contact Options

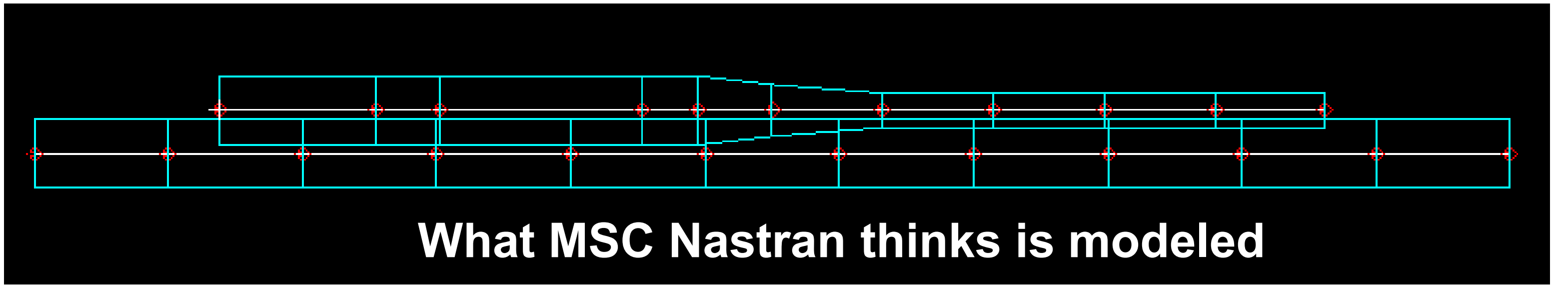
CURRENT LIMITATIONS

- **Composite Layup Offset**
  - MSC Nastran currently does not look at the Z0 definition in PCOMP and PCOMPG when doing contact search
  - It assumes the layup is centered about the grid plane
  - ZOFFS on the CQUAD4/8 entry is supported





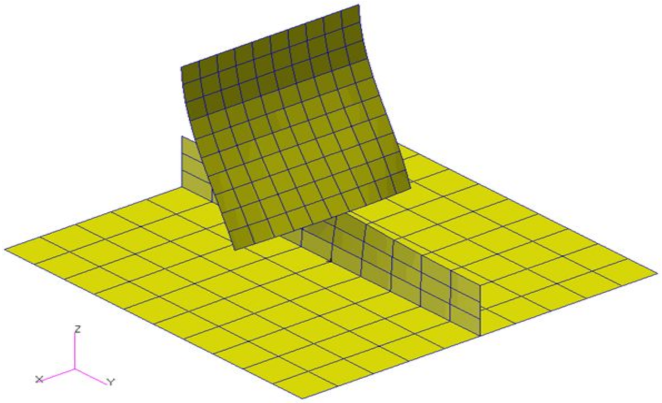
What the user modeled



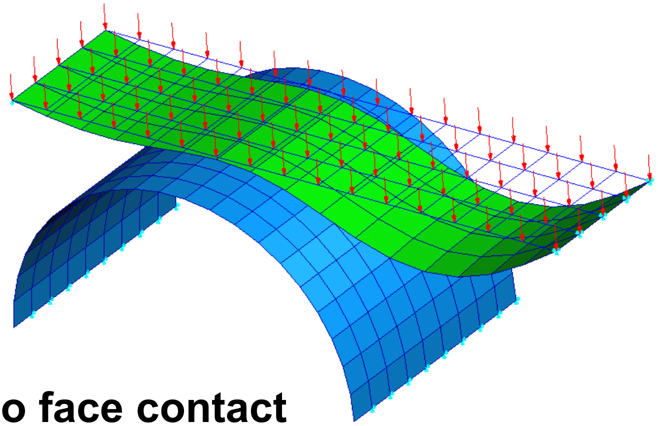
What MSC Nastran thinks is modeled

SHELL CONTACT ANALYSIS EXAMPLES

- **Shell and beam contact examples**

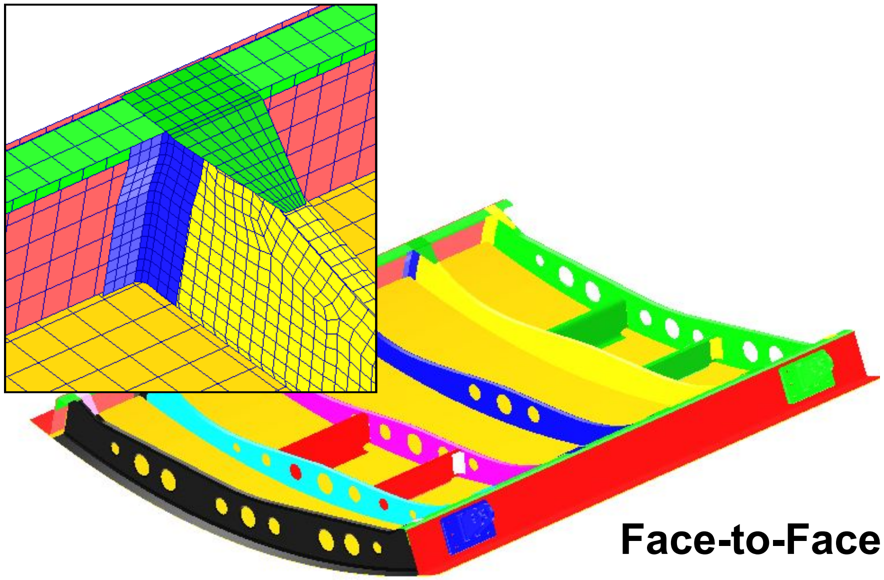


Edge-to-edge contact

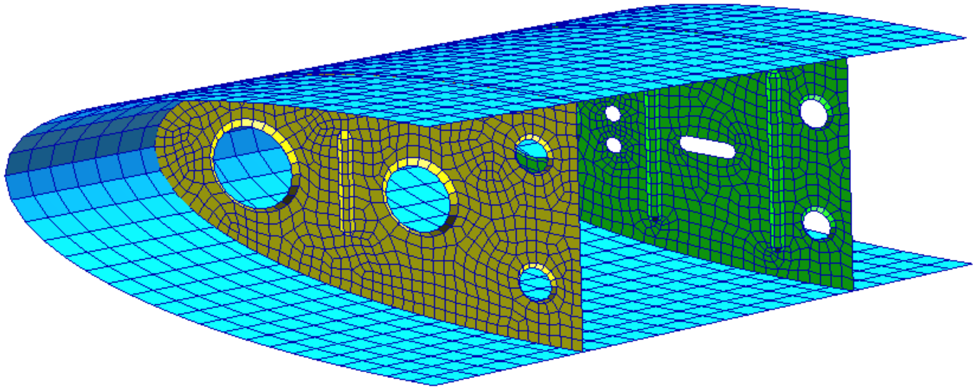


Face to face contact

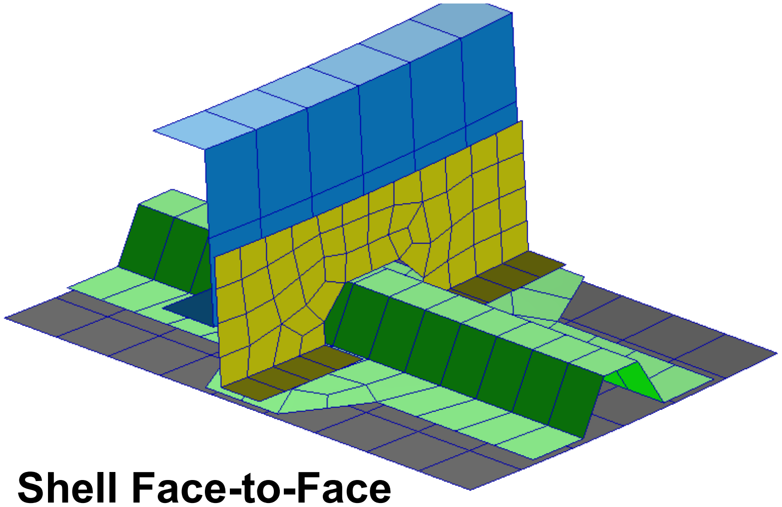
SHELL CONTACT ANALYSIS EXAMPLES



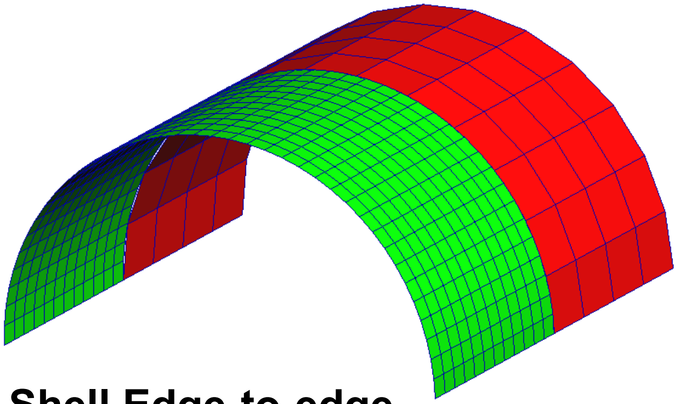
Face-to-Face



Shell Edge-to-Face



Shell Face-to-Face



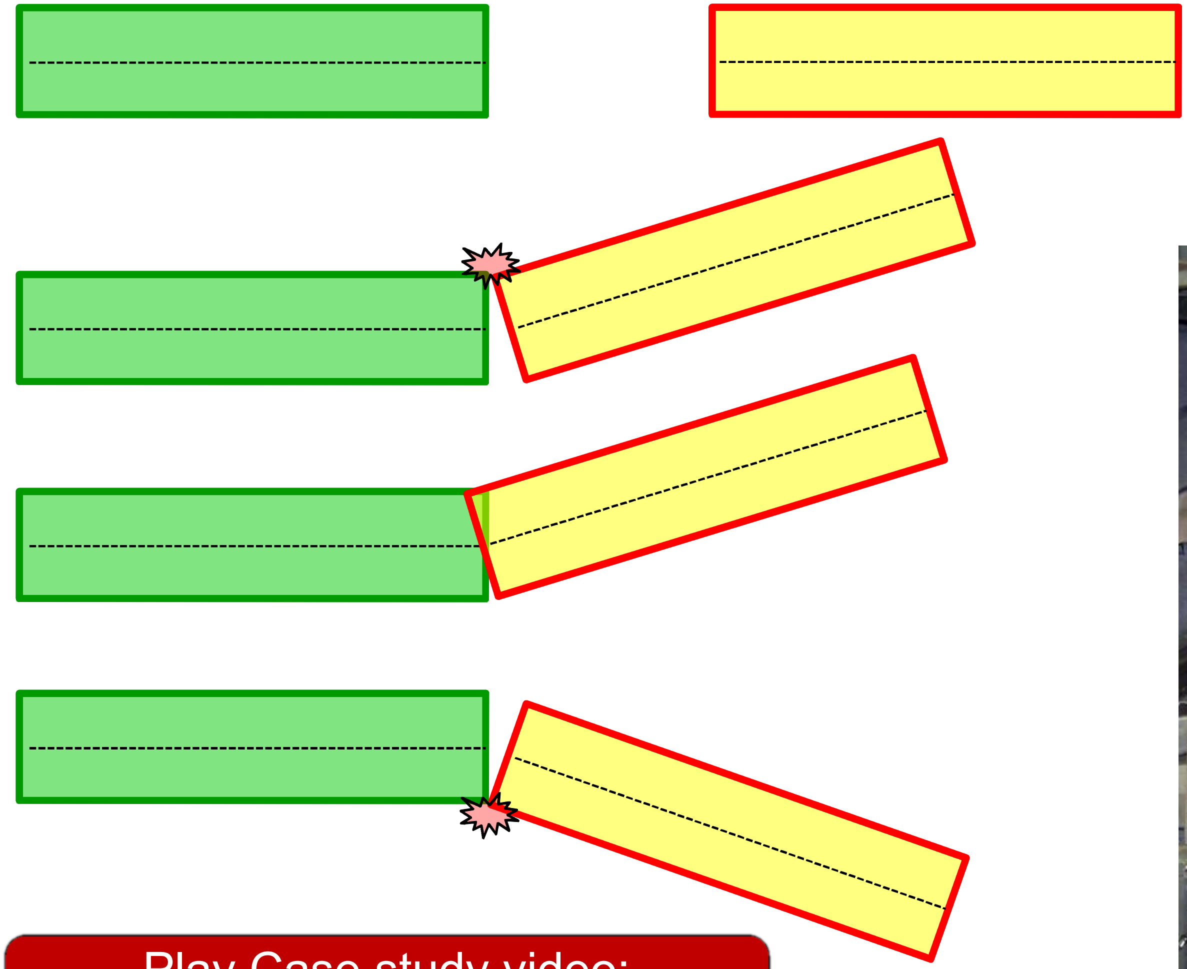
Shell Edge-to-edge

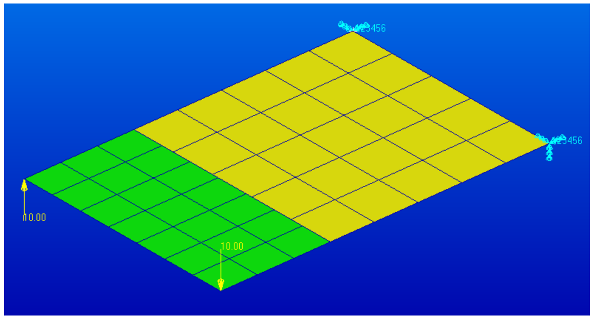
SHELL CONTACT ANALYSIS EXAMPLES

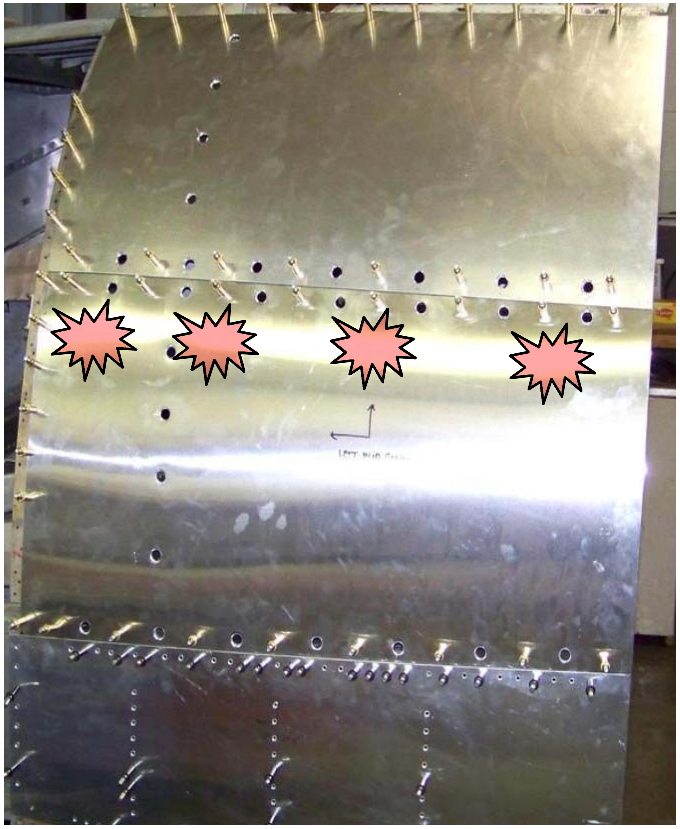
- **Shell Contact Analysis can be explained well by reviewing a series of Case studies**
- **The following page will introduce you to a live demo of Edge to Edge contact**
- **More examples of Shell contact can be found in Appendix D**
  - Shell Face-to-Face Touching Contact
  - Curved Shell Face-to-Face Touching Contact
  - Shell Edge-to-Face Glued Contact
  - Defining Contact With Tees & Clips

CASE STUDY: SHELL EDGE-TO-EDGE GLUED CONTACT

- **Meeting in the middle**



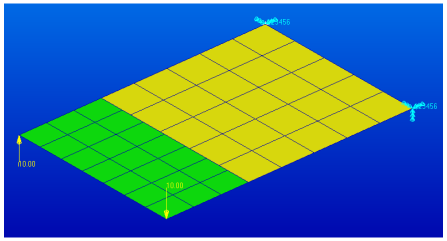


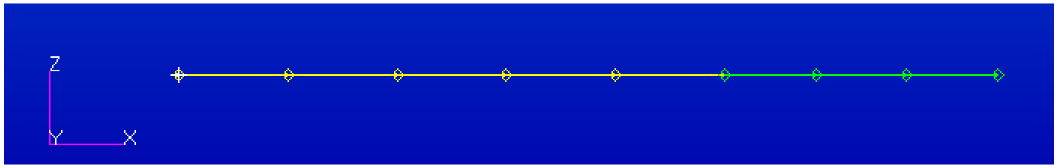


Play Case study video:
NAS133_S7_cs_edge_to_edge

CASE STUDY: SHELL EDGE-TO-EDGE GLUED CONTACT

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

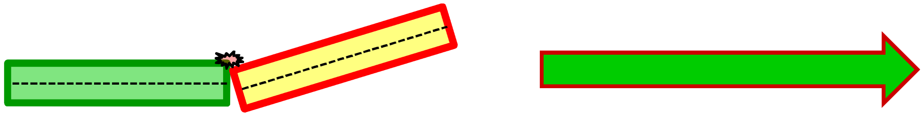


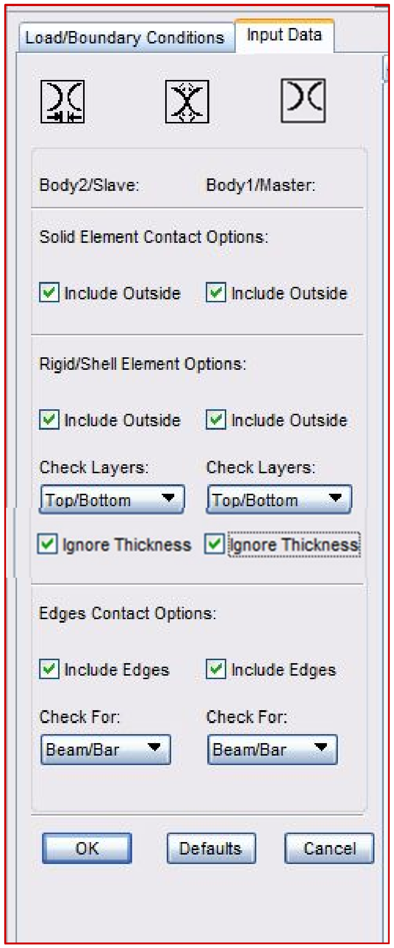


Shell thickness display

CASE STUDY 4: SHELL EDGE-TO-EDGE GLUED CONTACT

- **Edge-to-Edge Contact**
  - Ignore shell thickness for both bodies - simply glue the mid-planes of the shell bodies together (don’t try to detect corner-to-corner contact)





Body2/Slave:
Body1/Master:

Solid Element Contact Options:

☑ Include Outside ☑ Include Outside

Rigid/Shell Element Options:

☑ Include Outside ☑ Include Outside

Check Layers: Top/Bottom
Check Layers: Top/Bottom

☑ Ignore Thickness ☑ Ignore Thickness

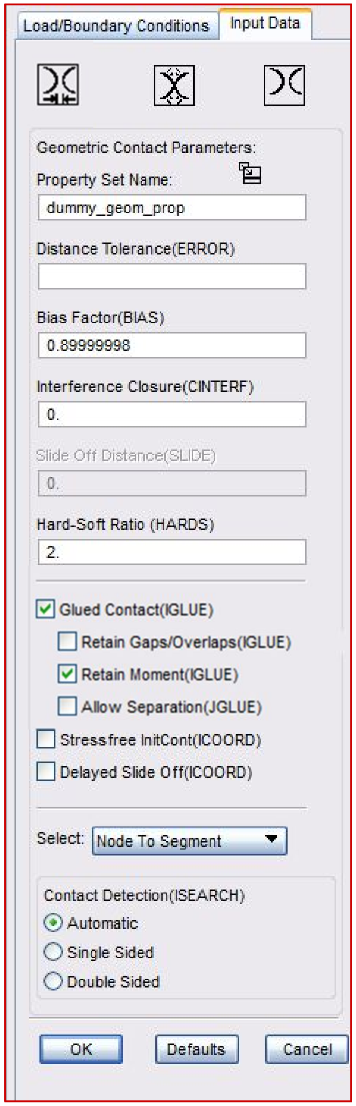
Edges Contact Options:

☑ Include Edges ☑ Include Edges

Check For: Beam/Bar
Check For: Beam/Bar

CASE STUDY: SHELL EDGE-TO-EDGE GLUED CONTACT

- **Wait just a *Moment*!**
  - Turn on *moment carrying capability* to enable the glued joint to transfer moments



Geometric Contact Parameters:

Property Set Name: dummy_geom_prop

Distance Tolerance(ERROR):

Bias Factor(BIAS): 0.89999998

Interference Closure(CINTERF): 0.

Slide Off Distance(SLIDE): 0.

Hard-Soft Ratio (HARDS): 2.

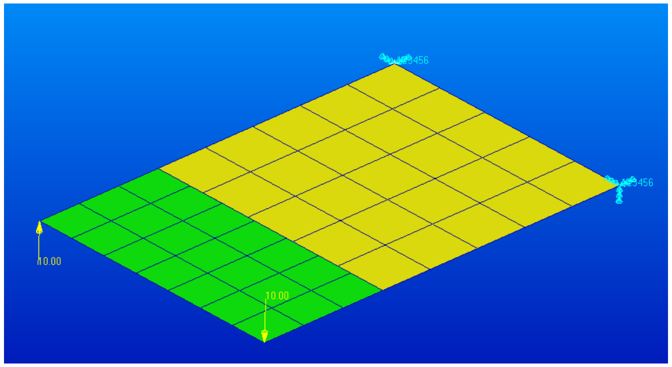
☑ Glued Contact(IGLUE)
  ☐ Retain Gaps/Overlaps(IGLUE)
  ☑ Retain Moment(IGLUE)
  ☐ Allow Separation(JGLUE)
☐ Stressfree InitCont(ICOORD)
☐ Delayed Slide Off(ICOORD)

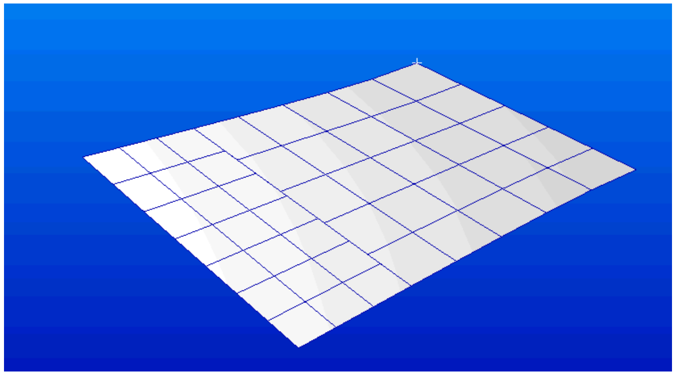
Select: Node To Segment

Contact Detection(ISEARCH)
● Automatic
○ Single Sided
○ Double Sided

CASE STUDY: SHELL EDGE-TO-EDGE GLUED CONTACT

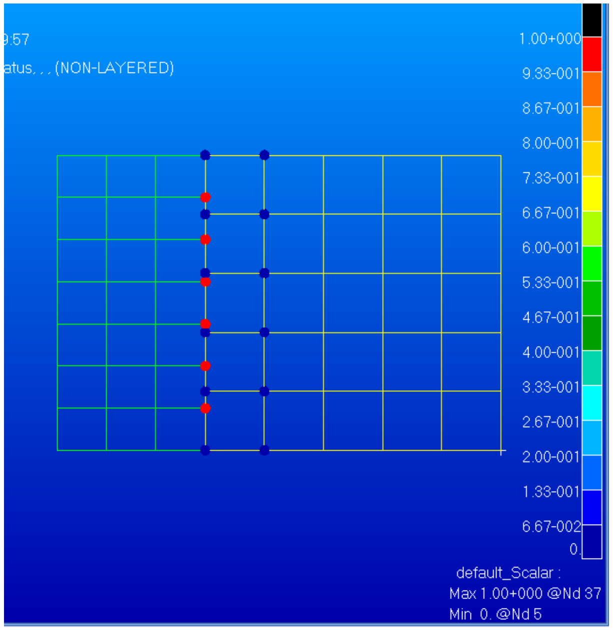
- **Plot the deformed shape**
  - The two plates appear to deform as one assembly

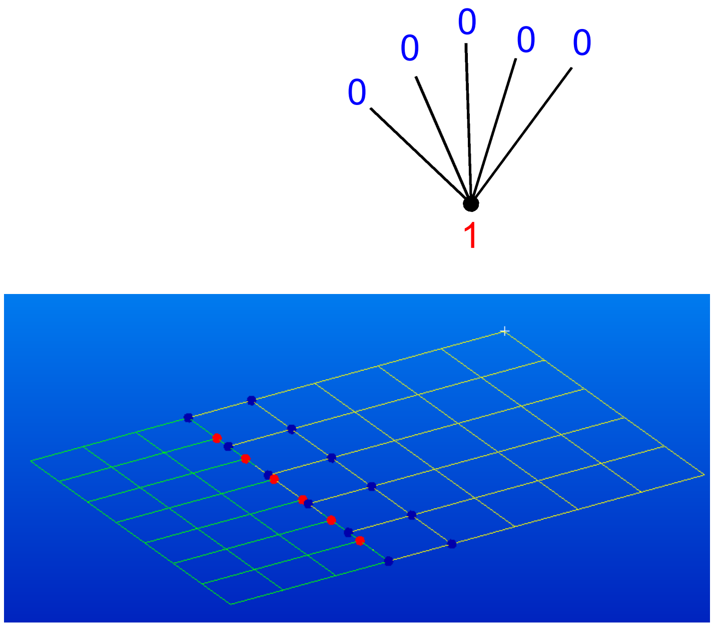




CASE STUDY: SHELL EDGE-TO-EDGE CONTACT

- **Plot the Contact Status**
  - 0 indicates a retained node (master)
  - 1 indicates a tied node (slave)





OVERVIEW

- What is and when to consider contact
- Contact Bodies (Workshop – Rubber Door Seal)
- Contact Pairs/Tables (Workshop – Deformable to Rigid Contact)
- Contact Detection
  - Node to segment
  - Segment to segment
- **Special Features**
  - Interference (Workshop – Interference Fit)
  - Stress free initial contact
  - Glued contact (Workshop – Contact Pairs)
  - Contact with Shells
  - Friction
- Convergence and Controls with contact

FRICTION

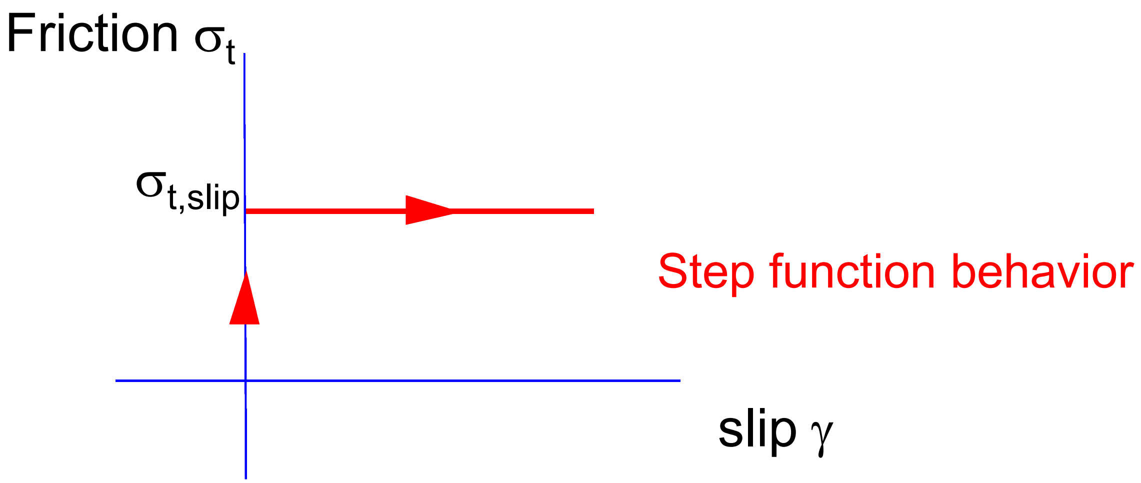
- Friction is a complex physical phenomena that is dependent on surface roughness, temperature, normal stress, and relative velocity
- Bodies in contact that move tangentially to each other will develop shear stresses at their interface - these stresses are due to friction
- If the stresses reach a critical value the bodies will slip and move relative to each other
- As long as the stresses remain below this critical value, the bodies are in “stick” status

FRICTION

- Two friction models, Bilinear Coulomb and Bilinear Shear, are supported in MSC Nastran
- Friction causes additional iterations in nonlinear analysis and usually leads to longer runtime
- Friction may make model more stable by preventing contact body free motion (such as free rotation of a pre-stressed bolt) and therefore improve convergence

COULOMB FRICTION

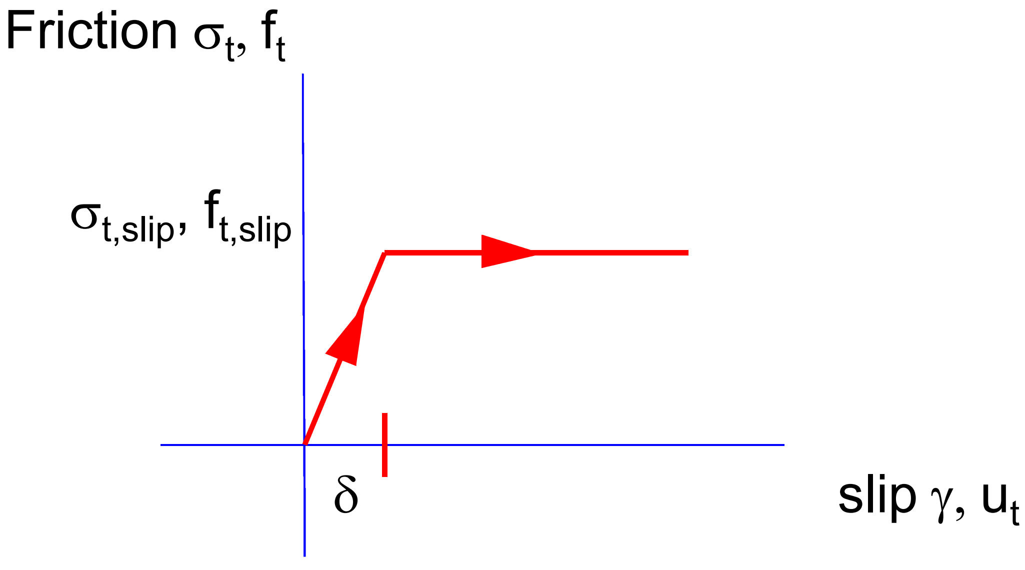
- **The critical value when slipping starts to occur usually depends on the contact stress σ_n**
- **$\sigma_{t,slip} = -\mu\ \sigma_n$ t, with μ as the friction coefficient and t as the slip direction**



- **This friction model is called Coulomb Friction**
- **The step function is discontinuous since the status is either stick or slip**

BILINEAR COULOMB FRICTION

- Since stick means infinite stiffness, a bilinear function is applied to avoid numerical difficulties
- The bilinear model assumes stick and slip conditions correspond to reversible (elastic) and permanent (plastic) relative displacements



δ = slip threshold

BILINEAR COULOMB FRICTION

- **The calculation proceeds as follows**
  - As for elastic-plastic material, the rate of the relative tangential displacement is split into an elastic (stick) and a plastic part (slip)

$$\dot{\mathbf{u}}_t = \dot{\mathbf{u}}_t^e + \dot{\mathbf{u}}_t^p$$

  - The slip criteria is defined as:

$$\phi = \left\|\mathbf{f}_t\right\| - \mu f_n \quad ; \quad \phi < 0(stick)\ ,\ \phi = 0(slip)$$

  - The rate of change of friction force vector is related to the elastic tangential displacement by

$$\dot{\mathbf{f}}_{\mathrm{t}} = \mathbf{D}\dot{\mathbf{u}}_{\mathrm{t}}^{\mathrm{e}} = \mathbf{D}(\dot{\mathbf{u}}_{\mathrm{t}} - \dot{\mathbf{u}}_{\mathrm{t}}^{\mathrm{p}})$$

BILINEAR COULOMB FRICTION

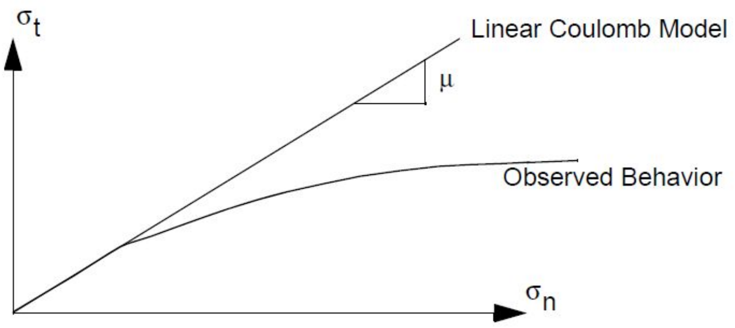
- **With these assumptions, the rate of the resulting friction force can be expressed as:**

$$\dot{\mathbf{f}}_t = \left(\mathbf{D} - \frac{\mathbf{D} \dfrac{\partial \psi}{\partial \mathbf{f}_t} \left(\dfrac{\partial \phi}{\partial \mathbf{f}_t} \right)^T \mathbf{D}}{\left(\dfrac{\partial \phi}{\partial \mathbf{f}_t} \right)^T \mathbf{D} \dfrac{\partial \psi}{\partial \mathbf{f}_t}} \right) \dot{\mathbf{u}}_t = (\mathbf{D} - \mathbf{D}^*) \dot{\mathbf{u}}_t$$

non-symmetric, but symmetrized

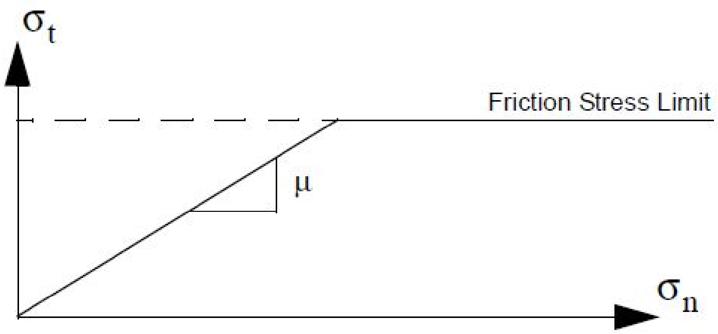
BILINEAR COULOMB FRICTION

- The Coulomb friction model may not correlate well with experiment if the frictional tractions exceed the flow stress or failure stress of the material



- Therefore, the user may define a friction stress limit $\sigma_{t,limit}$, FRLIM on the BCONPRP entry, so that the maximum friction stress will be

$$\min(\mu\sigma_n, \sigma_{t,\ limit})$$



BILINEAR SHEAR FRICTION

- **The second friction model available in MSC Nastran is the Shear Friction Model**
- **In this model the friction stress is a fraction of the equivalent stress $\bar{\sigma}$ in the material**

$$|\sigma_t| < m\frac{\bar{\sigma}}{\sqrt{3}} \ (stick) \text{ and } \sigma_t = -m\frac{\bar{\sigma}}{\sqrt{3}} \cdot \boldsymbol{t} \ (slip)$$

m is the friction factor

- **Similar to Coulomb friction, the bilinear model is adapted and the shear stress due to friction is limited by**

$$\sigma_t = min\left(m\sigma_n, m\frac{\bar{\sigma}}{\sqrt{3}}\right)$$

FRICTION INPUT

- **The Input parameters for friction are:**
  - NO friction, FTYPE=0. Default.
  - Coulomb Friction, FTYPE=6 in BCPARA
  - Shear Friction, FTYPE=7 in BCPARA
  - Friction coefficient, FRIC in BCBODY or BCONPRP (preferred)
  - Friction stress limit, FRLIM in BCONPRP
  - Slip threshold δ is input in BCPARA as
    - RVCNST for node to segment contact. Its default is 0.0025 times the average edge length of the elements defining the deformable contact bodies.
    - STKSLP for segment to segment contact. Its default is 0.0 for the maximum sticking displacement to be taken.

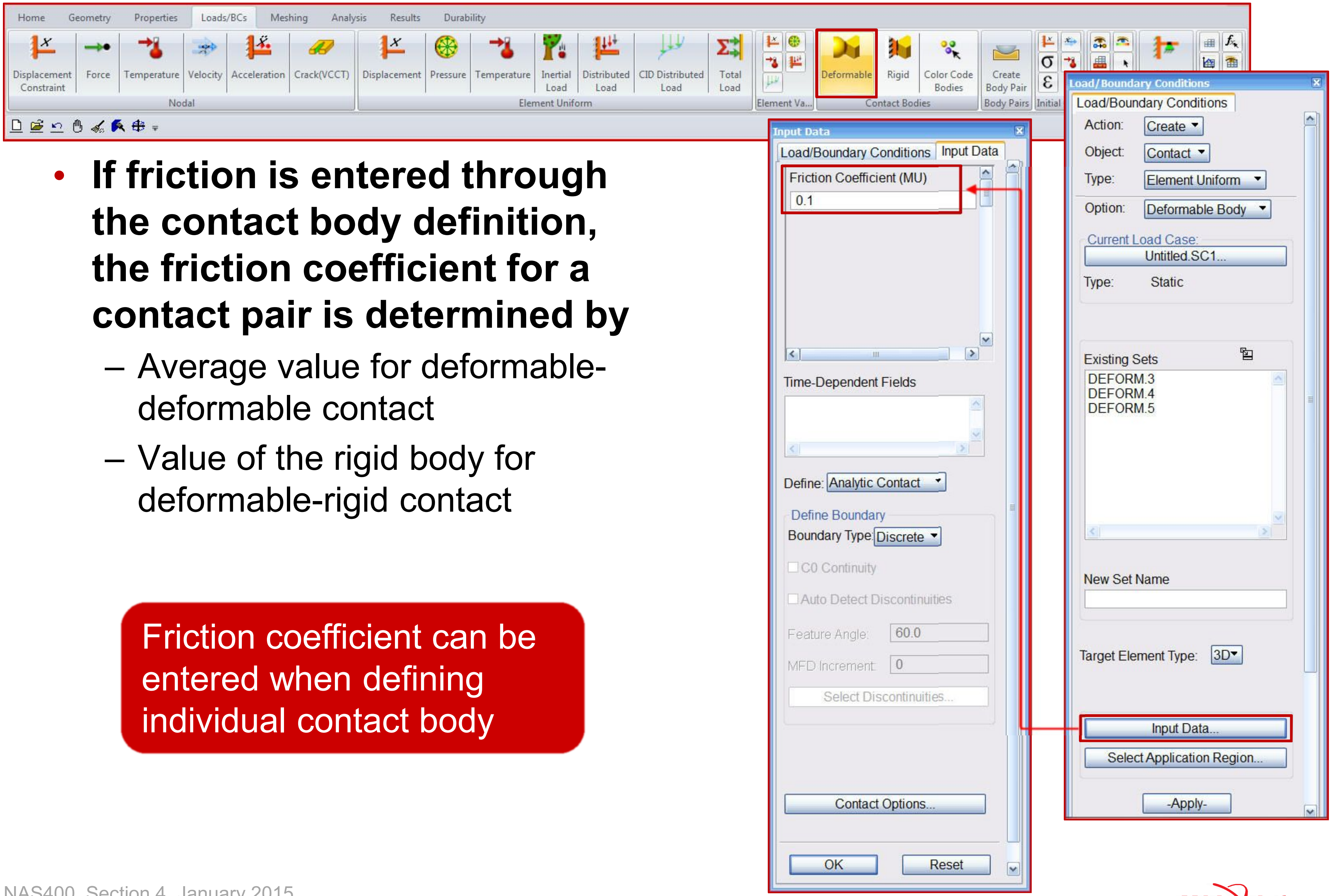
SETUP FRICTION IN PATRAN

- **Friction is not taken into account by default**
- **User needs to do two things to activate friction effect**
  - Enter friction coefficient
  - Select friction model
- **Friction coefficient can be defined in the contact body or the contact pair**
- **It is preferable to enter friction coefficient for the contact pair since friction is between contact bodies**

SETUP FRICTION IN PATRAN

- **If friction is entered through the contact body definition, the friction coefficient for a contact pair is determined by**
  - Average value for deformable-deformable contact
  - Value of the rigid body for deformable-rigid contact

Friction coefficient can be entered when defining individual contact body



Friction Coefficient (MU): 0.1

Define: Analytic Contact

Boundary Type: Discrete

Feature Angle: 60.0

MFD Increment: 0

Action: Create

Object: Contact

Type: Element Uniform

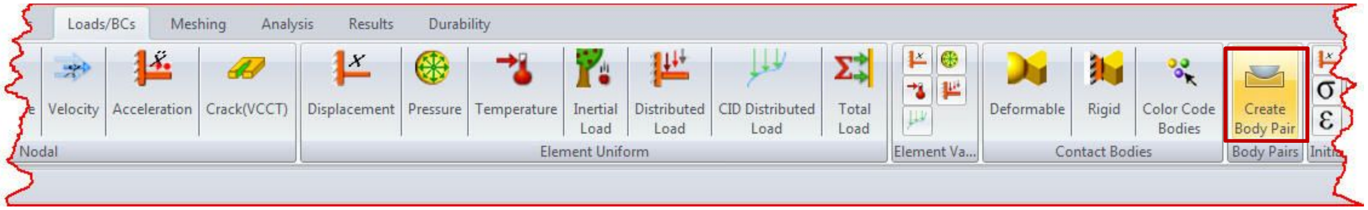
Option: Deformable Body

Type: Static

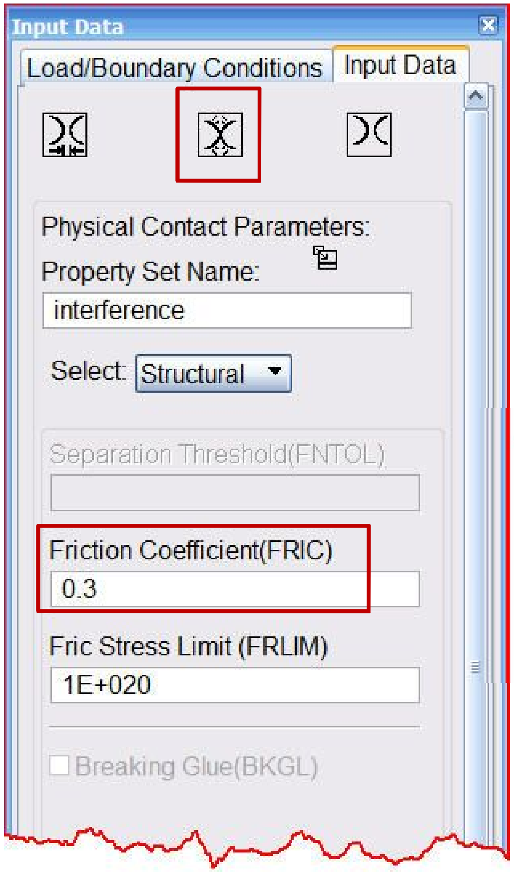
DEFORM.3
DEFORM.4
DEFORM.5

Target Element Type: 3D

SETUP FRICTION IN PATRAN



- Friction coefficient between a contact pair can be entered in the physical contact parameters form



Physical Contact Parameters:

Property Set Name: interference

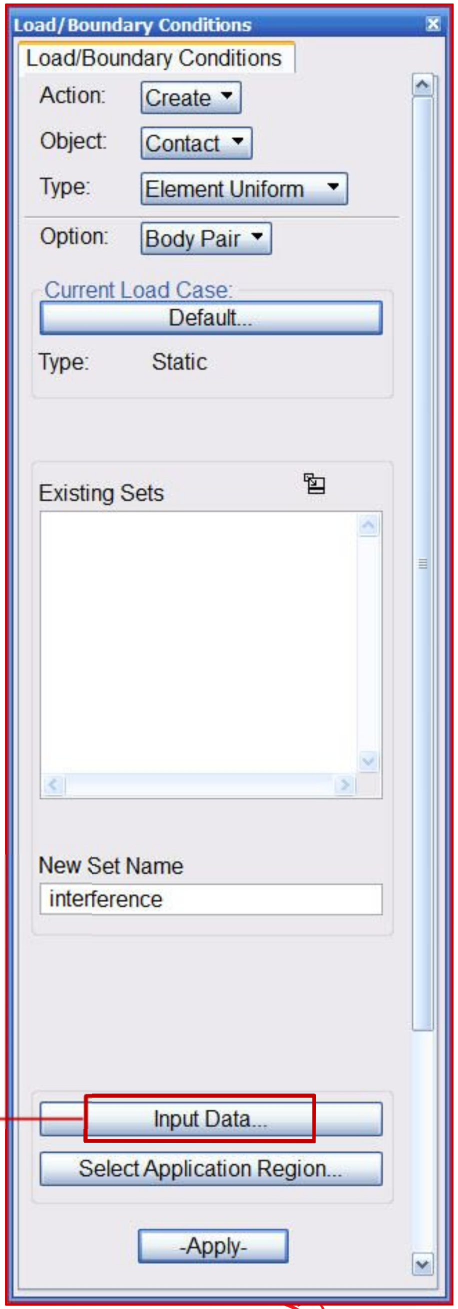
Select: Structural

Separation Threshold(FNTOL)

Friction Coefficient(FRIC): 0.3

Fric Stress Limit (FRLIM): 1E+020

☐ Breaking Glue(BKGL)



Action: Create

Object: Contact

Type: Element Uniform

Option: Body Pair

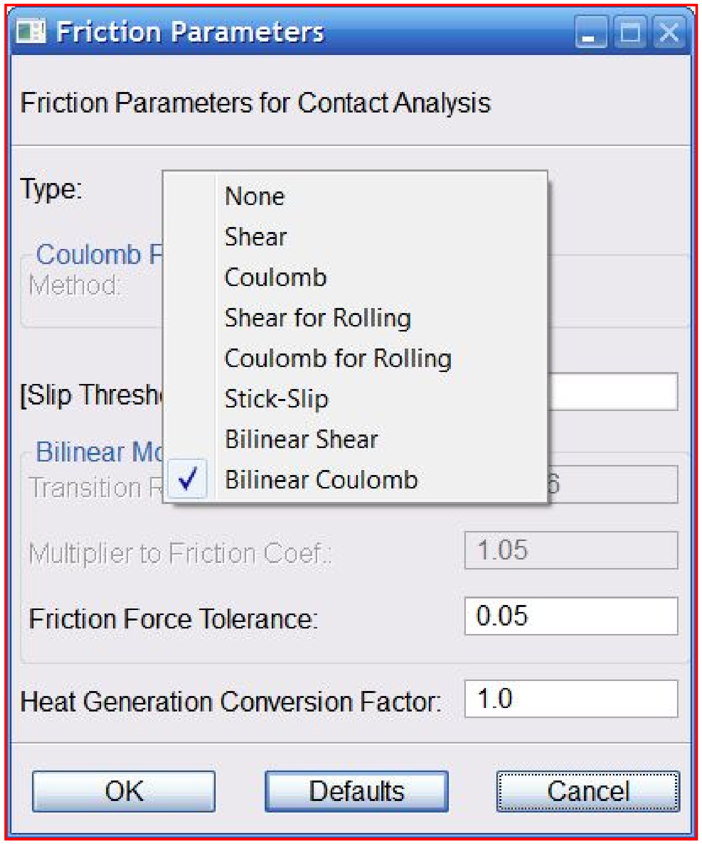
Current Load Case: Default...

Type: Static

New Set Name: interference

SETUP FRICTION IN PATRAN

- **Friction model is defined under Analysis / Solution Type / Solution Parameters / Contact Parameters / Friction**



Friction Parameters for Contact Analysis

Multiplier to Friction Coef.: 1.05

Friction Force Tolerance: 0.05

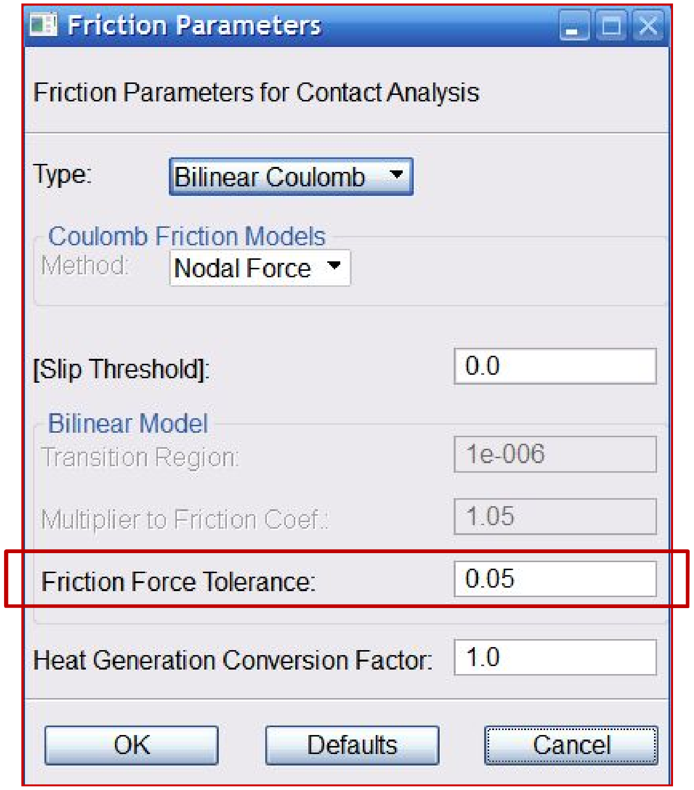
Heat Generation Conversion Factor: 1.0

Only Bilinear Coulomb and Bilinear Shear models are supported in solutions 101 and 400.

SETUP FRICTION IN PATRAN

- Friction model uses an additional check on the convergence of the friction forces
- The convergence criterion is based on relative friction force. The default convergence tolerance is 0.05

$$\frac{\left\| \|F_t\| - \|F_t^p\| \right\|}{\|F_t\|} \le e$$



Friction Parameters for Contact Analysis

Type: Bilinear Coulomb

Coulomb Friction Models

Method: Nodal Force

[Slip Threshold]: 0.0

Bilinear Model

Transition Region: 1e-006

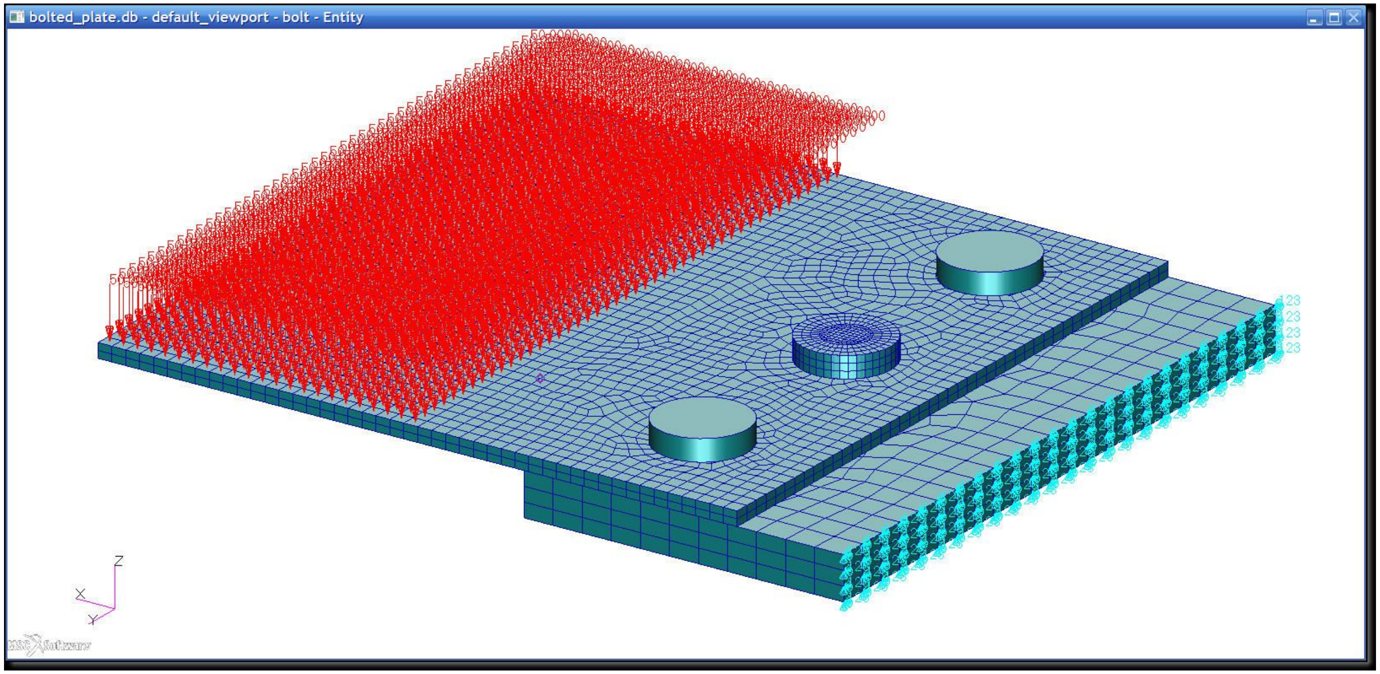
Multiplier to Friction Coef.: 1.05

Friction Force Tolerance: 0.05

Heat Generation Conversion Factor: 1.0

CASE STUDY: BOLTED PLATE

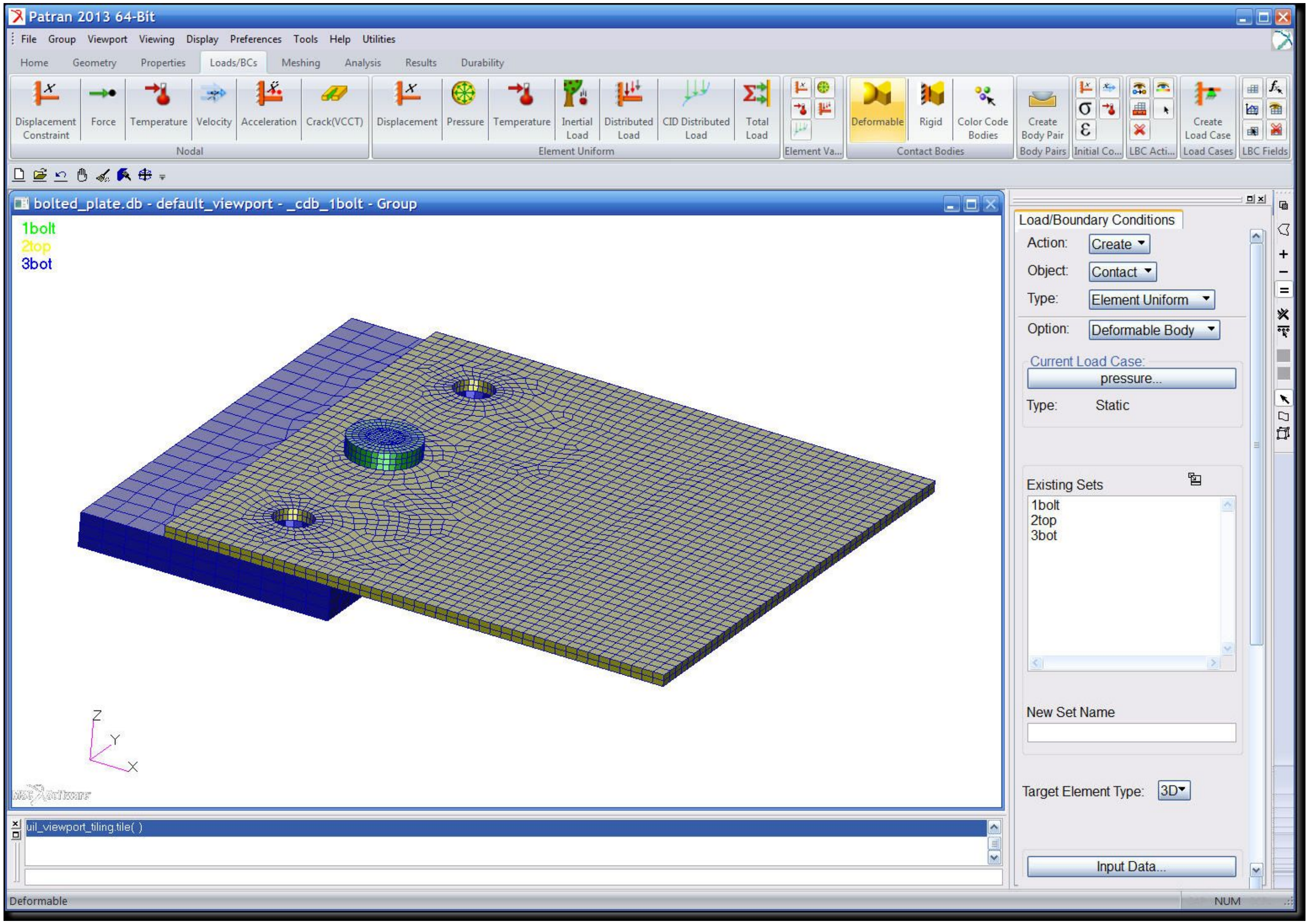
- **Two plates connected by a bolt**
- **Pressure on top plate and fixed bottom plate**



Play Case study video:
NAS133_S8_cs_bolted_plate

CASE STUDY: BOLTED PLATE

- **Create 3 contact bodies**



Action: Create

Object: Contact

Type: Element Uniform

Option: Deformable Body

Current Load Case: pressure...

Type: Static

Existing Sets

1bolt
2top
3bot

New Set Name

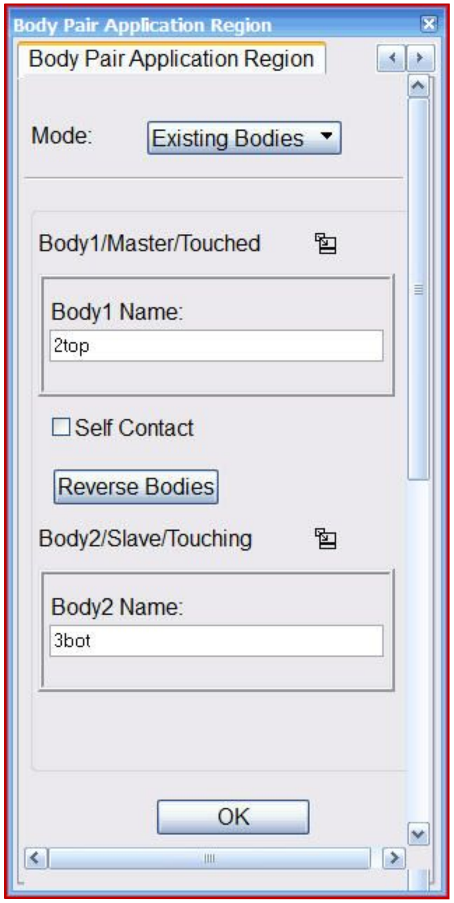
Target Element Type: 3D

Input Data...

```
uil_viewport_tiling.tile( )
```

CASE STUDY: BOLTED PLATE

- **Define 3 contact pairs**
  - Bolt and top plate
  - Bolt and bottom plate
  - Top plate and bottom plate

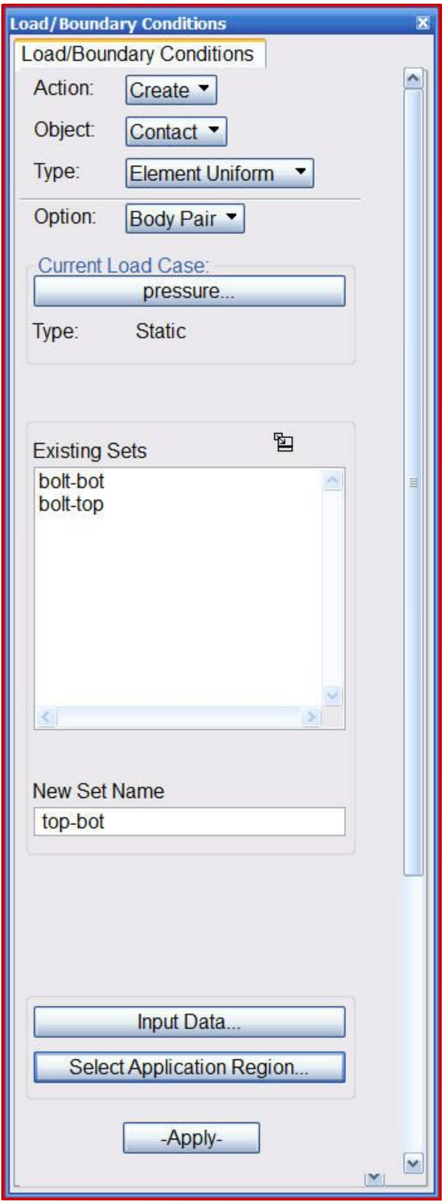


Mode: Existing Bodies

Body1 Name: 2top

☐ Self Contact

Body2 Name: 3bot



Action: Create

Object: Contact

Type: Element Uniform

Option: Body Pair

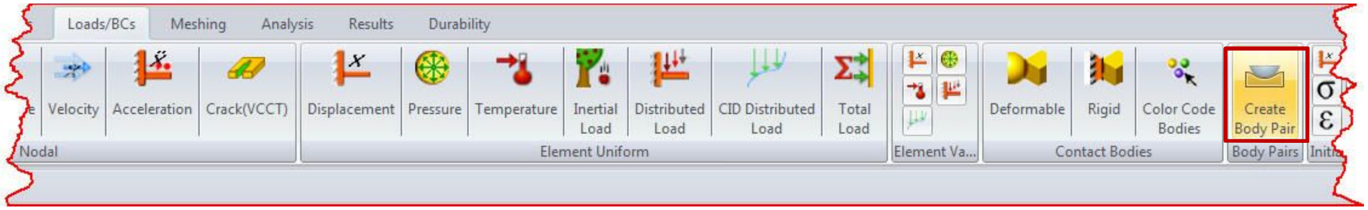
Current Load Case: pressure...

Type: Static

Existing Sets
bolt-bot
bolt-top

New Set Name: top-bot

CASE STUDY: BOLTED PLATE



- **Enter geometric and physical contact parameters**
  - Use default geometric parameters
  - Set friction coefficient to 0.3

The bolt and the top plate are free to rotate about z-axis and the model is unstable without friction

Physical Contact Parameters:

Property Set Name: bolt

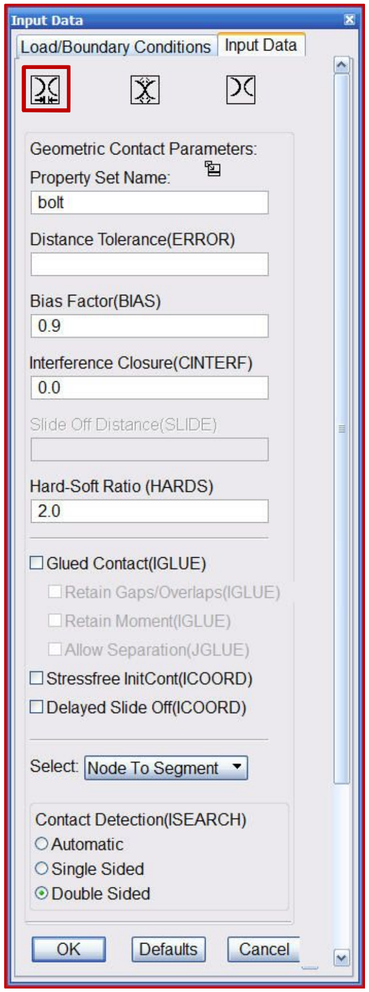
Select: Structural

Separation Threshold(FNTOL):

Friction Coefficient(FRIC): 0.3

Fric Stress Limit (FRLIM): 1E+020

☐ Breaking Glue(BKGL)



Geometric Contact Parameters:

Property Set Name: bolt

Distance Tolerance(ERROR):

Bias Factor(BIAS): 0.9

Interference Closure(CINTERF): 0.0

Slide Off Distance(SLIDE):

Hard-Soft Ratio (HARDS): 2.0

☐ Glued Contact(IGLUE)

☐ Retain Gaps/Overlaps(IGLUE)

☐ Retain Moment(IGLUE)

☐ Allow Separation(JGLUE)

☐ Stressfree InitCont(ICOORD)

☐ Delayed Slide Off(ICOORD)

Select: Node To Segment

Contact Detection(ISEARCH)

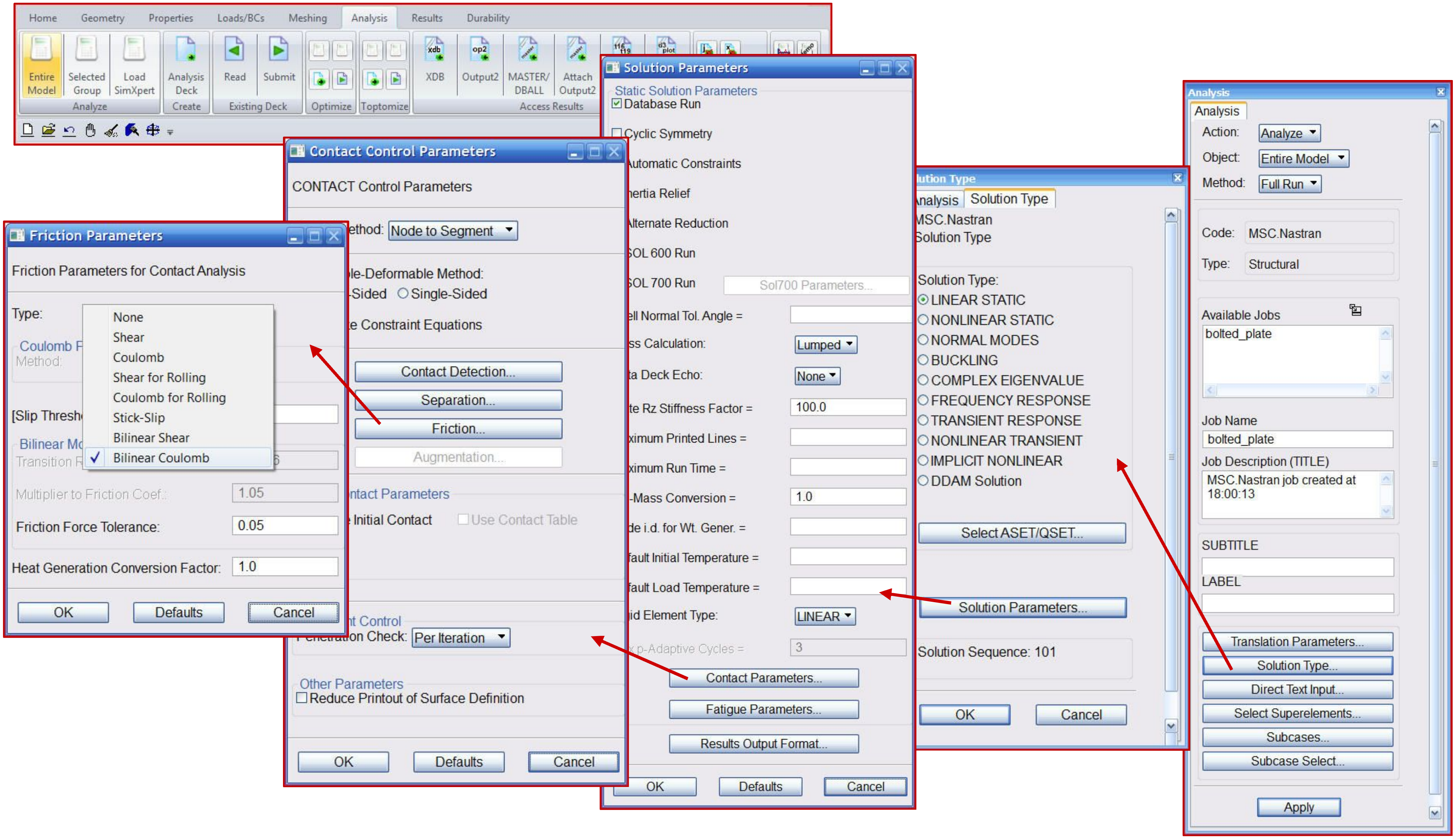
○ Automatic

○ Single Sided

● Double Sided

CASE STUDY: BOLTED PLATE

- **Select Bilinear Coulomb friction model**



Analysis

Action: Analyze
Object: Entire Model
Method: Full Run
Code: MSC.Nastran
Type: Structural
Available Jobs: bolted_plate
Job Name: bolted_plate
Job Description (TITLE): MSC.Nastran job created at 18:00:13
SUBTITLE
LABEL

Solution Type

MSC.Nastran Solution Type

Solution Type:
● LINEAR STATIC
○ NONLINEAR STATIC
○ NORMAL MODES
○ BUCKLING
○ COMPLEX EIGENVALUE
○ FREQUENCY RESPONSE
○ TRANSIENT RESPONSE
○ NONLINEAR TRANSIENT
○ IMPLICIT NONLINEAR
○ DDAM Solution

Solution Sequence: 101

Solution Parameters

Static Solution Parameters
☑ Database Run
☐ Cyclic Symmetry
Automatic Constraints
Inertia Relief
Alternate Reduction
SOL 600 Run
SOL 700 Run
Shell Normal Tol. Angle =
Mass Calculation: Lumped
Data Deck Echo: None
Plate Rz Stiffness Factor = 100.0
Maximum Printed Lines =
Maximum Run Time =
Wt.-Mass Conversion = 1.0
Node i.d. for Wt. Gener. =
Default Initial Temperature =
Default Load Temperature =
Solid Element Type: LINEAR
Max p-Adaptive Cycles = 3

Contact Control Parameters

CONTACT Control Parameters
Method: Node to Segment
Double-Deformable Method: Sided ○ Single-Sided
Constraint Equations

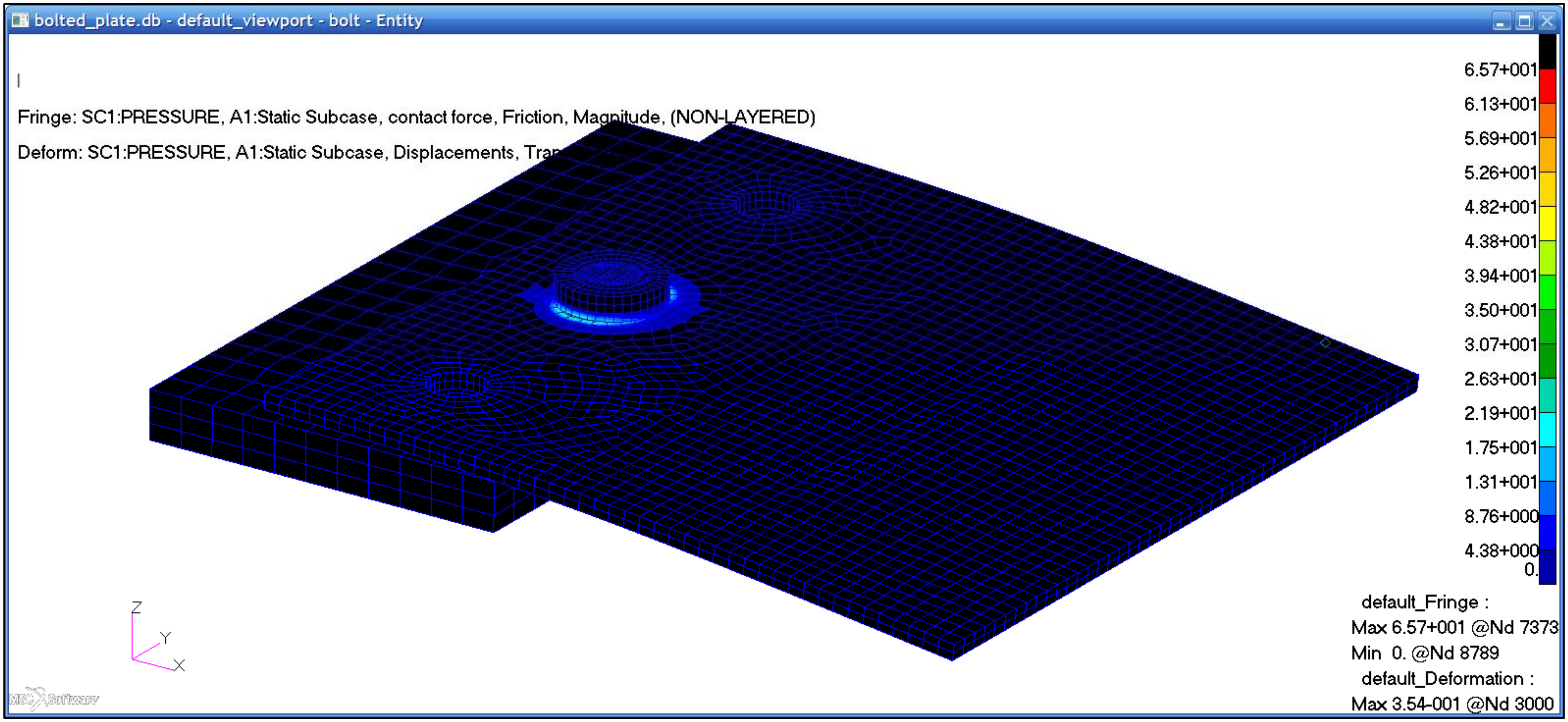
Contact Parameters
Initial Contact ☐ Use Contact Table
Control
Penetration Check: Per Iteration
Other Parameters
☐ Reduce Printout of Surface Definition

Friction Parameters

Friction Parameters for Contact Analysis
Type: Bilinear Coulomb
Coulomb F
Method:
[Slip Thresh
Bilinear Mo
Transition R
Multiplier to Friction Coef.: 1.05
Friction Force Tolerance: 0.05
Heat Generation Conversion Factor: 1.0

CASE STUDY: BOLTED PLATE

- **Plot contact friction force**



bolted_plate.db - default_viewport - bolt - Entity

Fringe: SC1:PRESSURE, A1:Static Subcase, contact force, Friction, Magnitude, (NON-LAYERED)

Deform: SC1:PRESSURE, A1:Static Subcase, Displacements, Tra

default_Fringe :
Max 6.57+001 @Nd 7373
Min 0. @Nd 8789
default_Deformation :
Max 3.54-001 @Nd 3000

OVERVIEW

- **What is and when to consider contact**
- **Contact Bodies** (Workshop – Rubber Door Seal)
- **Contact Pairs/Tables** (Workshop – Deformable to Rigid Contact)
- **Contact Detection**
  - Node to segment
  - Segment to segment
- **Special Features**
  - Interference (Workshop – Interference Fit)
  - Stress free initial contact
  - Glued contact (Workshop – Contact Pairs)
  - Contact with Shells
  - Friction
- **Convergence and Controls with contact**

CONVERGENCE AND CONTROL WHEN USING CONTACT

- **Review of Nonlinear Solution Concepts**
- **Guidelines for contact**
- **What to do if non-convergence**

SOLUTION OF CONTACT ANALYSIS

- Contact simulations are nonlinear in nature due to the potential for intermittent contact
- As such, iterative solution techniques are required that are only considered ‘successful’ if they achieve ‘convergence’
- There will be a chance of FAILED solutions

ACHIEVING CONVERGED SOLUTIONS

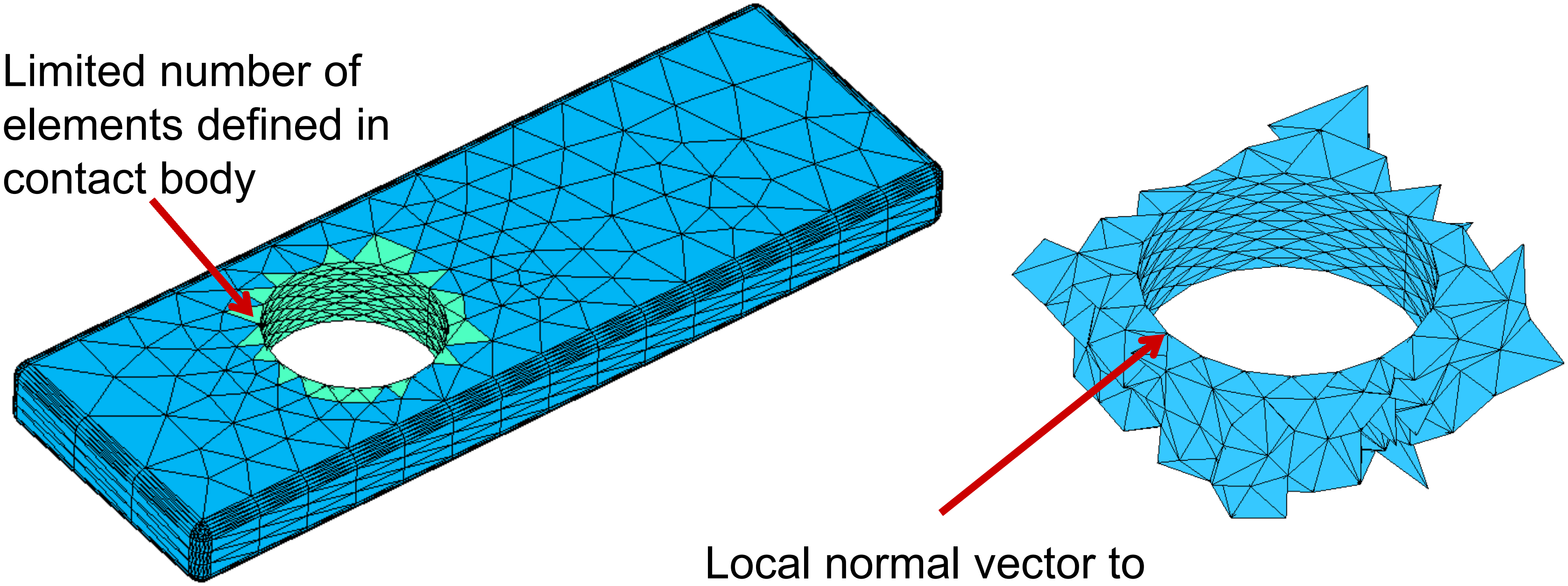
- **Follow general guidelines presented here to reduce chances of failure**
- **Confirm convergence (look in .f06 file)**
  - Just because a job completes does not mean the results are ‘good’
  - Cases where completed solutions are not ‘converged’ solutions
    - Convergence specifications too high
    - User override of abort logic forcing solution to continue

ACHIEVING CONVERGED SOLUTIONS

- **If non-convergence:**
  - Review the results that you have
  - Request intermediate results
  - Add additional output requests as necessary
  - Consider running without contact – if the problem still doesn’t run, then contact is likely not the problem (inadequate boundary conditions?)
- **Adjust as necessary based on the above output**

CONTACT GUIDELINES

- **When defining a deformable contact body, avoid selecting only a subset of elements:**



Limited number of elements defined in contact body

Local normal vector to the outer boundary may be completely wrong

CONTACT GUIDELINES

- **For contact problems, use the iteration methods FNT or PFNT**
  - CONV test flags available:
    - U = Displacement error
    - P = Load equilibrium error
    - W = Work error
    - V = Vector Component method
    - A = Auto switch
- **For contact problems with friction, use CONV test flags UPW**
- **For contact problems without friction, use CONV test flags PV**
- **When there are no external loads (for example, spring back), PV checking may not work well. Switch to UV or use PVA for Auto Switch.**
- **When there are no displacements (such as in thermal expansion), the UV checking will not work effectively. Switch to PV or use UVA.**
- **Avoid higher order shells (QUAD8) with contact**

CONTACT GUIDELINES

- **The sizes of the contact tolerances D1 and D2 have a significant impact on the computational costs and the accuracy of the solution**
- **Contact tolerances too small:**
  - Detection of contact is difficult, leading to higher costs. Initial contact might not be found.
- **Contact tolerance too large:**
  - Nodes are considered in contact prematurely, resulting in a loss of accuracy
  - Nodes might “penetrate” the surface by a large amount

DISTANCE TOLERANCES

- **Measured normal to the contacted body**
- **Input on BCPARA,0,ERROR, .005 or BCONPRG**

BCONPRG	BCGPID		PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	
	PARAM4	VAL4	PARAM5	VAL5	-etc.-				

- **Example**

BCONPRG	90		**ERROR**	**.005**					

- **By default, ERROR is evaluated from:**
  - $^{1}/_{20}$ x “smallest element edge“ for continuum elements within each contact pair
  - $^{1}/_{4}$ x “smallest thickness“ for beam and shell elements within each contact pair
- **By default, BIAS is 0.9**

CONTACT GUIDELINES

- **Set BIAS to 0.99 for frictional problems to get improved results.**
- **For shell contact set BIAS to 0.95.**
- **Use the default for the contact zone tolerance ERROR**
  - Default is 0.0 or blank
- **Use slave to master contact, single search, with**
  - The softer body as slave, for example, rubber should be slave and steel should be master
  - The finer meshed body defined as slave

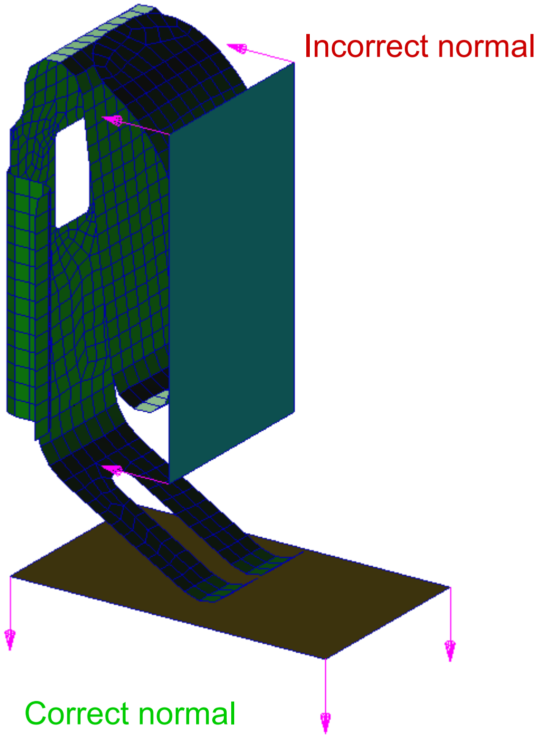
CONTACT GUIDELINES

- **Refine the mesh in the area of contact**
  - Coarse meshes can produce single point contact and promote instability
- **Consider smoothing the surfaces in contact. For example, insert a radius instead of a sharp corner for corner contact. If not possible then:**
  - define the body with sharp corners as slave body
  - increase ANG2D / ANG3D
  - set ICOORD to 2 (delay sliding off)
- **Use analytic contact (IDSPL in BCBODY)**
  - Advanced topic beyond scope of this course
- **Run without friction if possible**
  - May not be possible if friction is required for problem stability

CONTACT GUIDELINES

- **Confirm rigid body normals face away from the deformable body – otherwise the code may not detect contact**

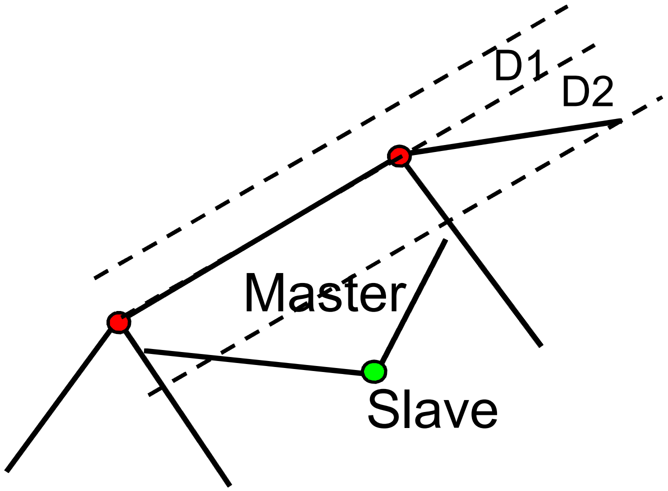
The rigid body normal should point away from the deformable body to be contacted



- **Contact can be lost or not found because of too large a load increment**
- **Analytical surface definition may be incorrect and causing “bulbous” corner/edge contact surfaces**

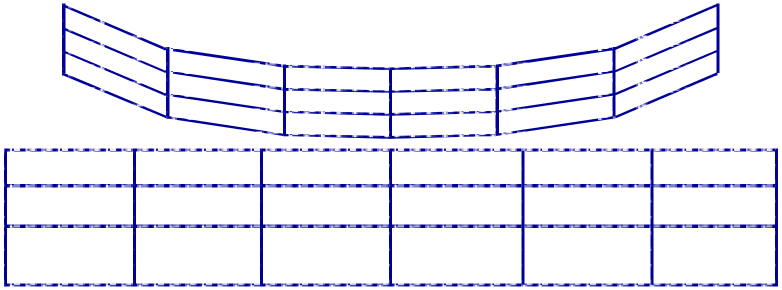
CONTACT GUIDELINES

- **Review and reconcile any initial contact over-closures and openings**
- **If nodes are modeled behind the contact zone, they will not be found**
- **If nodes move behind the contact zone during the analysis, the increment will be recycled with modified increment size**

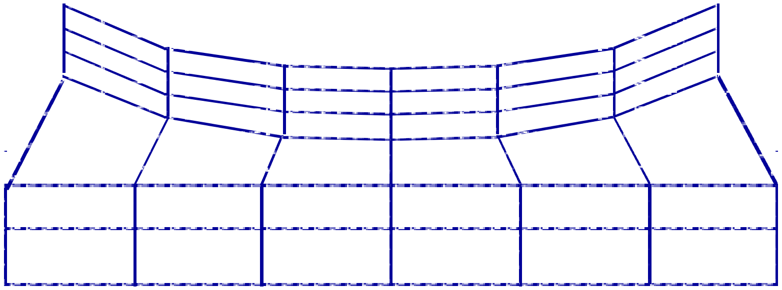


CONTACT GUIDELINES

- **Use BCPARA,0,NLGLUE,1 in SOLs 101/400:**
  - When some bodies are glued and others are not
  - Bodies are ultimately glued but there is no initial contact
- **If using Glued Contact and ICOORD: Make sure that the resulting elements will not be distorted too much**



Before



After

CONVERGENCE DIFFICULTIES

- Before embarking on a thorough investigation of why the contact behavior is not converging, confirm that the contacts are in fact the problem by running without contact activated – define a contact table with no pairs active
- Additional boundary conditions may be necessary to prevent rigid body motion if the contacts were providing support of some kind

CONVERGENCE DIFFICULTIES

- **If contacts have been identified as the source of the non-convergence**
  - Review output
    - Output files (.f06, .sts)
    - Results files (.xdb, .MASTER/.DBALL, .op2)
  - Request additional diagnostic output to .f06
  - Request intermediate steps to results files

The **Contact Output** section of this course covers these topics in detail

CONVERGENCE DIFFICULTIES

- **Options to consider**
  - Smaller timesteps
    - reduces numerical ‘effort‘ for any single increment
  - Increase separation force
    - must check results carefully to see if the results are ‘good‘
  - Adjust distance tolerance (typically increase) and bias (typically move closer to 1.)

CONVERGENCE DIFFICULTIES

- **Options to consider**
  - In extreme cases try running with all contacts GLUED and then run additional test runs releasing (make TOUCH) one contact at a time to see if the offender can be isolated

NONLINEAR CONTROL

- **Control of MSC Nastran nonlinear solutions is provided primarily by the MSC Nastran command NLSTEP**
- **Format: (For SOL 400)**

1	2	3	4	5	6	7	8	9	10
NLSTEP	ID	TOTTIME	CTRLDEF						
	“GENERAL”	MAXITER	MINITER	MAXBIS	CREEP				
	“FIXED”	NINC	NO						
	“ADAPT”	DTINITF	DTMINF	DTMAXF	NDESIR	SFACT	INTOUT	NSMAX	
		IDAMP	DAMP	CRITTID	IPHYS	LIMTAR	RSMALL	RBIG	
		ADJUST	MSTEP	RB	UTOL				
	“ARCLN”	TYPE	DTINITFA	MINALR	MAXALR	SCALEA	NDESIREA	NSMAXA	
	“HEAT”	CONVH	EPSUH	EPSPH	EPSWH	KMETHODH	KSTEPH		
		MAXQNH	MAXLSH	LSTOLH					
	“MECH”	CONV	EPSU	EPSP	EPSW	KMETHOD	KSTEP	MRCONV	
		MAXQN	MAXLS	LSTOL	FSTRESS				
	“COUP”	HGENPLAS	HGENFRIC						
	“RCHEAT”	SOLVER	DRLXCA	ARLXCA	BALENG	DAMPC	GRVCON	CSGFAC	
		NRLOOP	OUTLINV	DTIME1					

NONLINEAR OUTPUT CONTROL

- **A ‘*.sts’ status file will automatically be written for each SOL400 job**

```
information summary of job:  ./demo_400_nlio_std
version: MSC Nastran 2013.0.0, Built on May 30, 2013
date:     Mar 26, 2014; Day Time: 13:00:34

  subcase      inc  cycl  sepa cut      cycl   split   separ   cut  rmesh time step   total time  wall time   cpu time    max resp. type
  /step #       #     #    #    #        #        #      #      #     #      of          of
        1          |--of the inc--|-----------of the analysis-----------|  the inc      the job
        1        0     0    0    0        0       0       0     0     0  0.0000E+00  0.0000E+00       2.00       1.40   0.0000E+00 disp
        1        1     4    0    0        4       0       0     0     0  1.0000E+00  1.0000E+00       2.00       1.44  -1.5486E+02 disp
 Job ends with exit number :       0
           total wall time:          2.00
           total cpu  time:          1.47

           exit DEFINITION -----
           = 0  job terminates normally
           = 1  job terminates abnormally (check Fatal Error Message in F06)
```

- **Further information can be found in the .f06 file**

NONLINEAR OUTPUT CONTROL

```
N O N - L I N E A R   I T E R A T I O N   S O L U T I O N   C O N T R O L   P A R A M E T E R S

     LOOP CONTROLS FOR :   SUBCASE        1,         STEP        1,         SUBSTEP        0


     SOLUTION CONTROL PARAMETERS FROM : NLSTEP                    ID :          1

          Total Time of Loading Case (TOTTIME) ....     1.00E+00

          Maximum Number of Iterations (MAXITER) ..           10
          Minimum Number of Iterations (MINITER) ..            1
          Maximum Number of Bisection (MAXBIS) ....           10
          Creep Option (CREEP) ...................             0

          Number of Fixed Increments (NINC) .......            1
          Interval of Output (NO) ................             1

          Convergence Criteria (CONV) ............       P  V
                     - Displacement   (EPSU) ......    -1.00E-01
          Tolerance  - Residual Force (EPSP) ......     1.00E-03
                     - Work           (EPSW) ......     1.00E-01
          Option of Rotations and Moments (MRCONV).            3
          Matrix Update Option (KMETHOD) ..........         PFNT
          Matrix Update Increment (KSTEP) .........            1
          Maximum Quasi-Newton Vectors (MAXQN) ....            0
          Maximum Line Searches (MAXLS) ..........             0
          Line Search Tolerance (LSTOL) ..........      5.00E-01
          Error Tolerance in YF (FSTRESS) ........      2.00E-01

 *** USER INFORMATION MESSAGE 6204 (NL3EMA)
     0.000000E+00 SECONDS REQUIRED TO DECOMPOSE MATRIX.
%1.00000E+00   1  1 1.00E+00 2.69E+01 1.00E+00  1.000  1   1   0  4.92E+04 6.533E+06 1.06E+01 -1.587E+02        6 2  1.00 0  0    0    1
%1.00000E+00   1  2 9.33E-02 1.62E+00 8.19E+02  0.060  0   1   0  2.41E+03 3.233E+06 1.13E+01 -1.558E+02        6 2  1.00 0  0    1    2
%1.00000E+00   1  3 5.83E-03 1.66E-01 2.07E-02  0.063  0   1   0  1.60E+00 3.233E+06 1.12E+01 -1.549E+02        6 2  1.00 0  0    2    3
%1.00000E+00   1  4 3.07E-06 7.29E-07 8.63E-03  0.093  0   1   0  2.71E-05 3.233E+06 1.12E+01 -1.549E+02        6 2  1.00 0  0    3    4

 *** JOB CONVERGES FOR THE CURRENT STEP.
 *** SUBCASE      1 STEP      1 IS COMPLETED.
```

Standard .f06 Output

NONLINEAR OUTPUT CONTROL

- **LOAD STEP**
  - step number minus 1 plus fraction of step, i.e. 0.08 = 8% of first step
- **NO. INC**
  - increment number
- **ITR**
  - iteration number
- **DISP LOAD WORK**
  - displacement, load and energy errors, must be smaller than the tolerances EPSU, EPSP and EPSW
- **CONV RATE**
  - should be between 0 and 1, bigger than 1 means, the solution will never converge

```
        LOAD  NO.       - - ERROR FACTORS - -    CONV  ITR MAT NO.    AVG     TOTL        - - - - - DISP - - - - - -  LINE_S  NO. TOT TOT
        STEP  INC ITR  DISP     LOAD      WORK   RATE  DIV DIV BIS  R_FORCE   WORK      AVG         MAX     AT GRID C FACT NO QNV KUD ITR
%1.00000E+00   1  1 1.00E+00 2.69E+01 1.00E+00  1.000  1   1   0   4.92E+04 6.533E+06 1.06E+01 -1.587E+02        6 2  1.00 0  0   0    1
```

NONLINEAR OUTPUT CONTROL

- **ITR DIV**
  - divergence counter, > MAXDIV triggers the divergence process
- **MAT DIV**
  - divergence counter for element and material routines
- **NO. BIS**
  - number of bisections
- **AVG R_FORCE**
  - average residual force (forces and moments); should be small
- **TOTL WORK**
  - approximate total work
- **DISPLACEMENTS: AVG MAX AT GRID C**
  - average, maximum displacement at grid in direction c

```
        LOAD  NO.       - - ERROR FACTORS - -    CONV  ITR MAT NO.    AVG     TOTL       - - - - - DISP - - - - - -   LINE_S  NO. TOT TOT
        STEP  INC ITR  DISP     LOAD      WORK   RATE  DIV DIV BIS  R_FORCE   WORK      AVG        MAX     AT  GRID C FACT NO QNV KUD ITR
%1.00000E+00   1  1 1.00E+00 2.69E+01 1.00E+00  1.000   1   1   0 4.92E+04 6.533E+06 1.06E+01 -1.587E+02        6 2  1.00  0 0   0    1
```

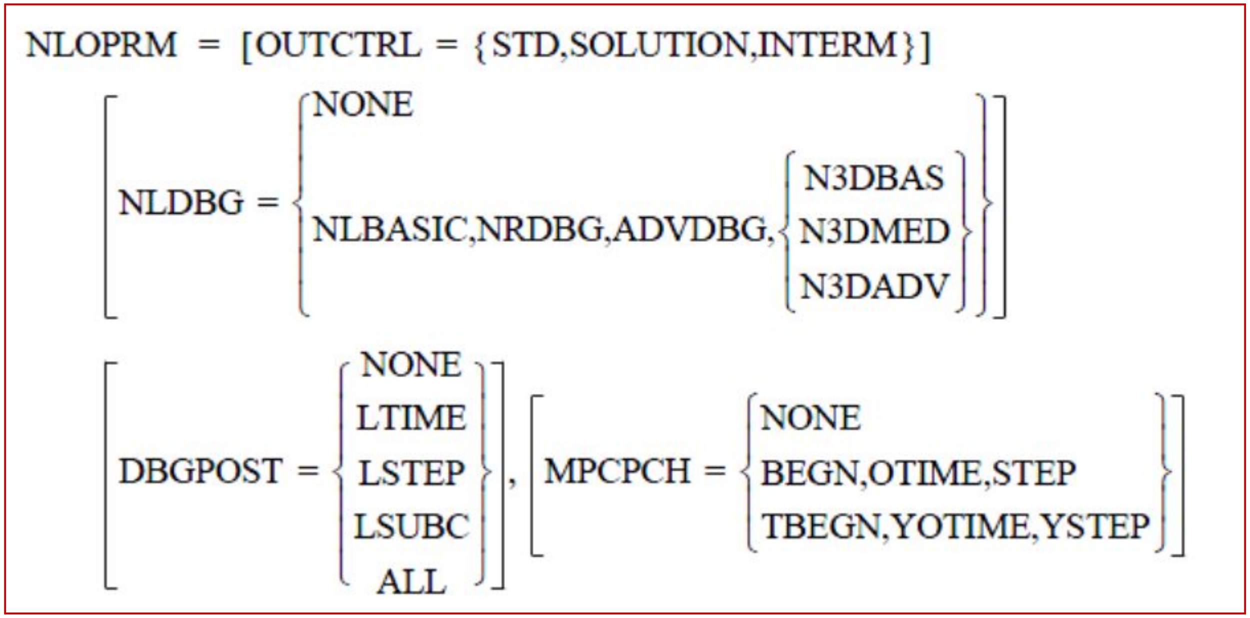
NONLINEAR OUTPUT CONTROL

- **LINE_S: FACT NO**
  - line search factor a and number of line searches
- **NO. QNV**
  - number of Quasi Newton Vectors
- **TOT KUD / ITR**
  - total of stiffness updates / iterations

```
        LOAD   NO.        - - ERROR FACTORS - -      CONV   ITR MAT NO.     AVG      TOTL         - - - - - DISP - - - - - -   LINE_S  NO. TOT TOT
        STEP   INC ITR   DISP      LOAD       WORK    RATE   DIV DIV BIS  R_FORCE    WORK       AVG          MAX     AT GRID C FACT NO QNV KUD ITR
%1.00000E+00    1   1 1.00E+00 2.69E+01 1.00E+00  1.000  1    1    0   4.92E+04 6.533E+06 1.06E+01 -1.587E+02         6 2  1.00 0  0    0     1
```

NONLINEAR OUTPUT CONTROL

- **Case Control command NLOPRM is also available**
- **Format**



$$\text{NLOPRM} = \left[\text{OUTCTRL} = \{\text{STD,SOLUTION,INTERM}\}\right]$$

$$\left[\text{NLDBG} = \left\{\begin{array}{l}\text{NONE}\\ \text{NLBASIC,NRDBG,ADVDBG,}\left\{\begin{array}{c}\text{N3DBAS}\\ \text{N3DMED}\\ \text{N3DADV}\end{array}\right\}\end{array}\right\}\right]$$

$$\left[\text{DBGPOST} = \left\{\begin{array}{c}\text{NONE}\\ \text{LTIME}\\ \text{LSTEP}\\ \text{LSUBC}\\ \text{ALL}\end{array}\right\}\right], \left[\text{MPCPCH} = \left\{\begin{array}{l}\text{NONE}\\ \text{BEGN,OTIME,STEP}\\ \text{TBEGN,YOTIME,YSTEP}\end{array}\right\}\right]$$

- **Examples**

```
NLOPRM          OUTCTRL=STD,SOLUTION          DBGPOST=LTIME
NLOPRM          OUTCTRL=(SOLUTION,INTERM),  MPCPCH=(OTIME,STEP)
```

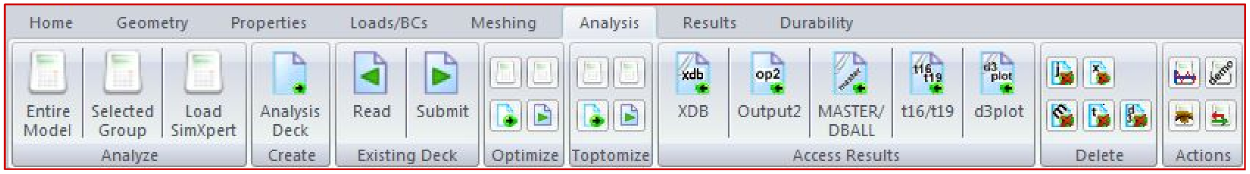
NONLINEAR OUTPUT CONTROL

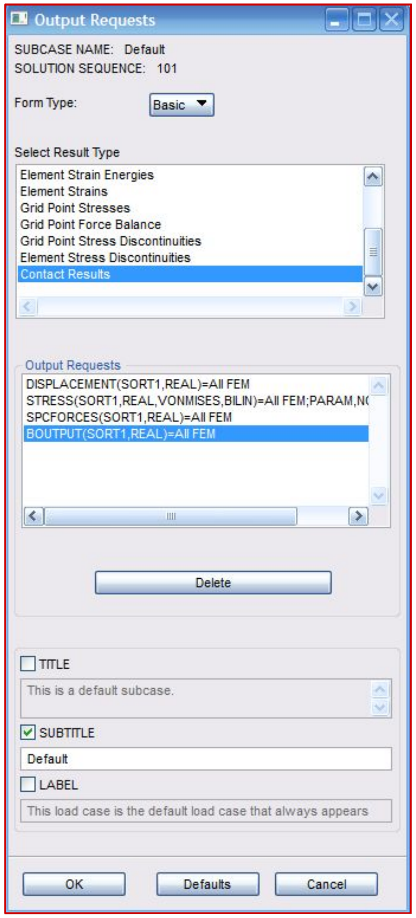
- **Example using Case Control: NLOPRM NLDBG=ADVDBG**

.f06 Output:
First iteration

```
 *** USER INFORMATION MESSAGE 6204 (NL3EMA)
     0.000000E+00 SECONDS REQUIRED TO DECOMPOSE MATRIX.

             maximum residual force  at node         6 degree of freedom  1 is equal to    5.566E+05
             maximum reaction force  at node         1 degree of freedom  1 is equal to    2.072E+04
             residual convergence ratio    2.687E+01

             maximum residual moment at node         5 degree of freedom  6 is equal to    4.656E+04
             maximum reaction moment at node         1 degree of freedom  6 is equal to    9.982E+04
             residual convergence ratio    4.664E-01

             maximum displacement  change   at node         6 degree of freedom  2 is equal to    1.587E+02
             maximum displacement increment at node         6 degree of freedom  2 is equal to    1.587E+02
             displacement convergence ratio   1.000E+00

             maximum   rotation    change   at node         6 degree of freedom  6 is equal to    2.381E-01
             maximum   rotation   increment at node         6 degree of freedom  6 is equal to    2.381E-01
             rotation convergence ratio   1.000E+00

             strain energy change at this iteration is    6.53309E+06
             strain energy change at this increment is    6.53309E+06
             relative energy error is                     1.00000E+00
%1.00000E+00   1  1 1.00E+00 2.69E+01 1.00E+00  1.000  1   1   0  4.92E+04 6.533E+06 1.06E+01 -1.587E+02        6 2  1.00 0   0    0    1
```

BOUTPUT DEFINITION IN PATRAN





SUBCASE NAME: Default
SOLUTION SEQUENCE: 101

Form Type: Basic

Select Result Type
- Element Strain Energies
- Element Strains
- Grid Point Stresses
- Grid Point Force Balance
- Grid Point Stress Discontinuities
- Element Stress Discontinuities
- Contact Results

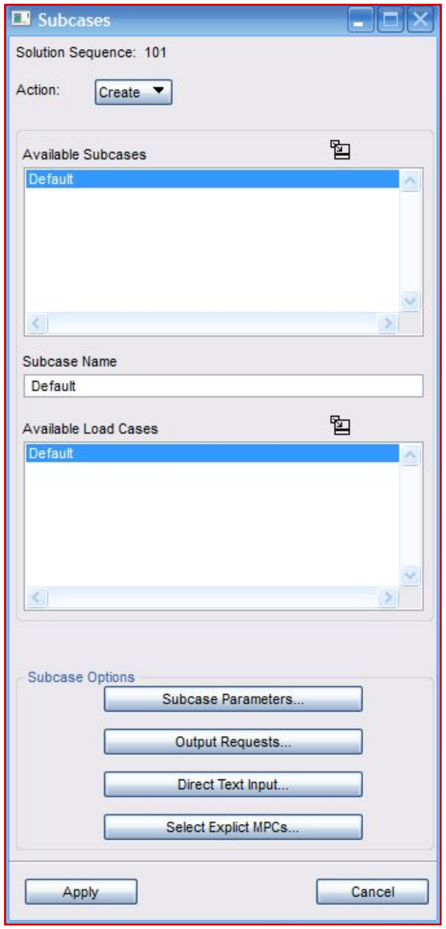
Output Requests

```
DISPLACEMENT(SORT1,REAL)=All FEM
STRESS(SORT1,REAL,VONMISES,BILIN)=All FEM;PARAM,NO
SPCFORCES(SORT1,REAL)=All FEM
BOUTPUT(SORT1,REAL)=All FEM
```

☐ TITLE
This is a default subcase.

☑ SUBTITLE
Default

☐ LABEL
This load case is the default load case that always appears



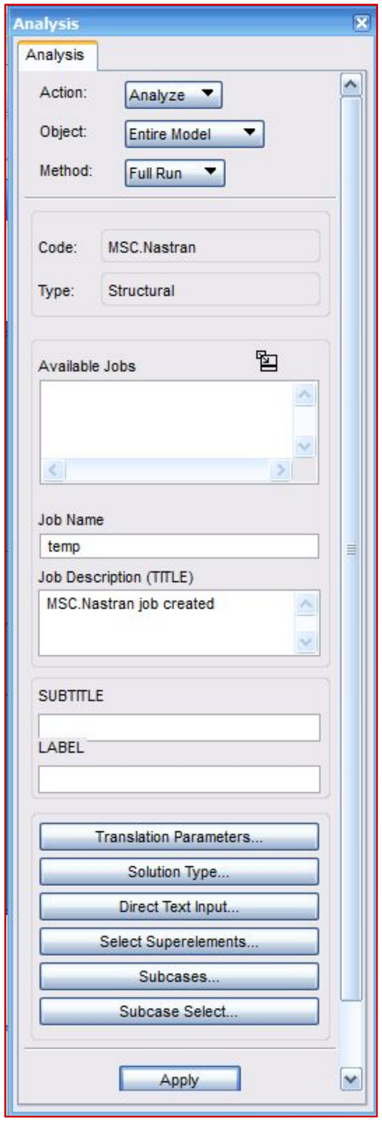
Solution Sequence: 101

Action: Create

Available Subcases
- Default

Subcase Name
Default

Available Load Cases
- Default



Action: Analyze
Object: Entire Model
Method: Full Run

Code: MSC.Nastran
Type: Structural

Available Jobs

Job Name
temp

Job Description (TITLE)
MSC.Nastran job created

SUBTITLE

LABEL

FUNDAMENTAL CONTROLS IN CONTACT – SUMMARY

- **Contact analyses are nonlinear and require a nonlinear solution**
- **NLSTEP controls all nonlinear behavior**
- **Smart logic is built into NLSTEP and is a good starting point**
- **Additional output requests are available by NLOPRM if difficulties are encountered in gaining convergence**
- **General guidelines detailed in this section show path to convergence**

SECTION 5

ADVANCED NONLINEAR MATERIALS

OVERVIEW

- Large Strain Elastic-Plastic Material – MATEP
- Nonlinear Elastic Material – MATS1/3/8/ORT
- Advanced Hyperelastic Material – MATHE
- Composite Failure – MATF
- Gasket Material – MATG
- Crack Materials – MCOHE & VCCT
- Creep Material – MATVP
- Table Input for nonlinear materials
- Guidelines and Limitations

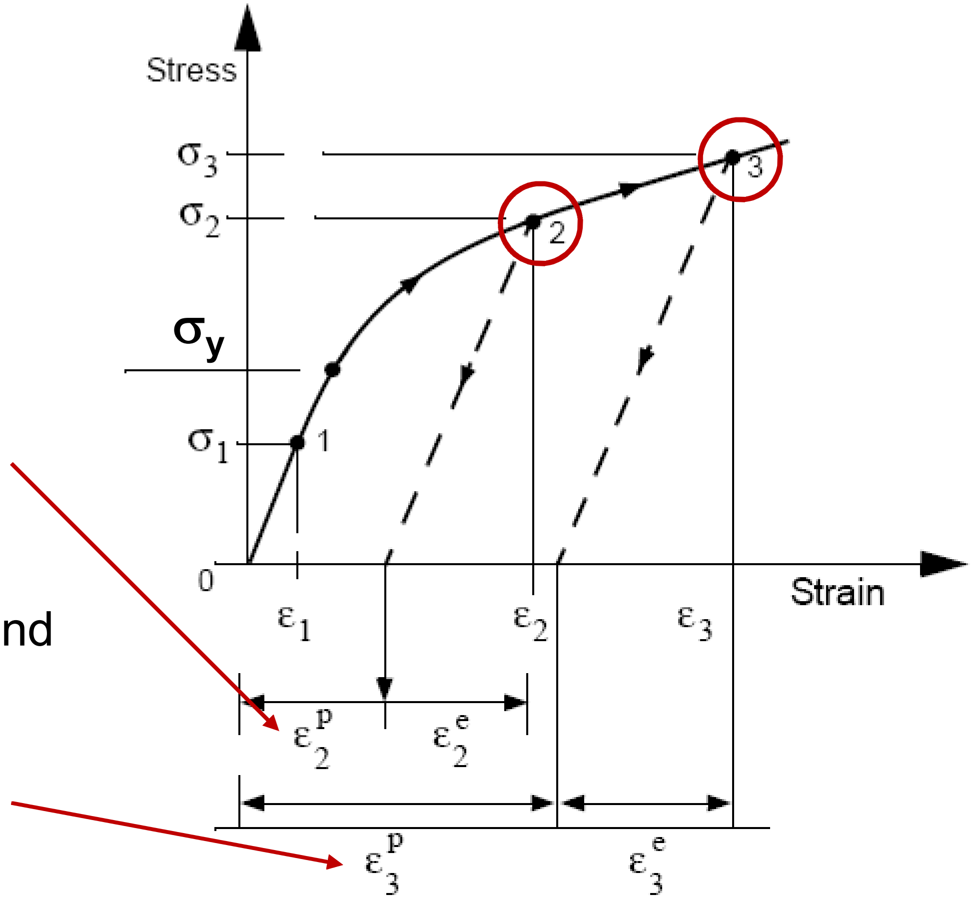
ADVANCED NONLINEAR MATERIALS

- The Advanced Nonlinear Materials presented in this chapter are normally invoked through a nonlinear property extension, for example PSLDN1 for solid elements
- Starting with 2010, the nonlinear property extensions are automatically added if advanced nonlinear materials are used in a model
- In addition the use of NLMOPTS Bulk Data Card may be required when the defaults are not appropriate, for example LRGSTRN=1 for large strain plasticity

LARGE STRAIN ELASTIC-PLASTIC MATERIAL

UNIAXIAL TEST

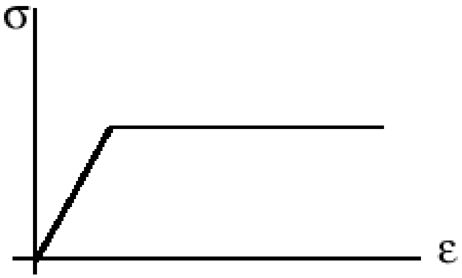
- **Point 1**
  - elastic, no plasticity at all
  - loading = unloading
- **Point 2**
  - initial yield stress σ_y has grown due to hardening
  - after unloading, plastic deformation remains
- **Point 3**
  - yield stress has further grown and after unloading plastic deformation remains which is larger than before



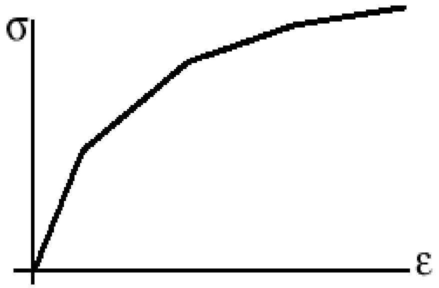
$$\varepsilon_2 = \varepsilon_2^p + \varepsilon_2^e$$

$$\varepsilon_3 = \varepsilon_3^p + \varepsilon_3^e$$

TYPICAL STRESS-STRAIN CURVES

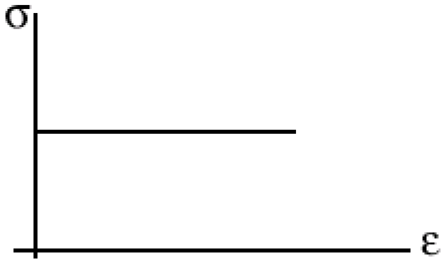


elastic – perfectly plastic



piecewise linear

bilinear



perfectly or rigid plastic

RULES FOR ELASTIC-PLASTIC BEHAVIOR

- **A Yield Criterion to determine the yield surface**
  - von Mises
  - Hill
  - Barlat

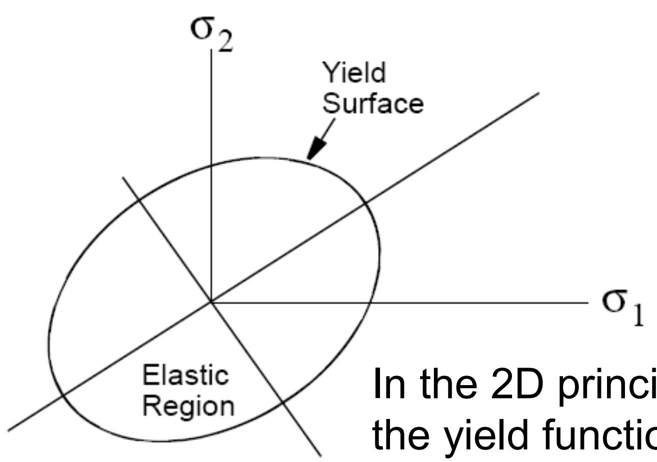
  metals

  - Mohr-Coulomb **soil, rock, ice, concrete**
- **A Hardening Rule for the change of yield stress**
  - Isotropic
  - Kinematic
  - Combined
- **A Flow Rule for plastic deformation**
  - Associated flow rule

VON MISES YIELD STRESS (1928)

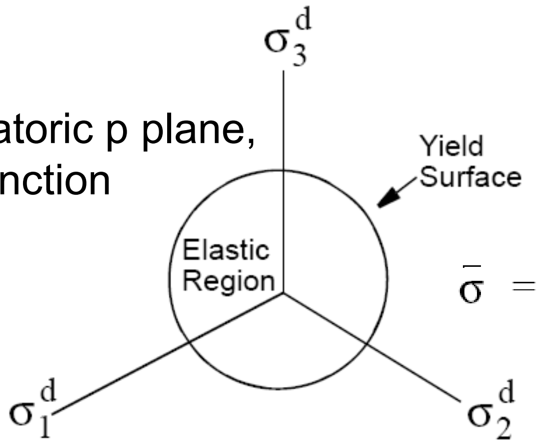
- **For isotropic yield**

$$\bar{\sigma} = \sqrt{\frac{1}{2}[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2] + 3(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}$$



In the 2D principal stress plane, the yield function is an ellipse

In the deviatoric p plane, the yield function is a circle



$$\bar{\sigma} = \sqrt{\frac{3}{2}\sigma_{ij}^d \sigma_{ij}^d}$$

MATEP	MID		Y0		vMISES				

HILL’S YIELD FUNCTION (1948)

- **for anisotropic yielding:**

$$\bar{\sigma} = \sqrt{F(\sigma_2 - \sigma_3)^2 + G(\sigma_3 - \sigma_1)^2 + H(\sigma_1 - \sigma_2)^2 + 2L\tau_{23}^2 + 2M\tau_{31}^2 + 2N\tau_{12}^2}$$

- **with the ratios of actual to isotropic yield:**

$$F = \frac{1}{2}\left(\frac{1}{R_{22}^2} + \frac{1}{R_{33}^2} - \frac{1}{R_{11}^2}\right) \qquad G = \frac{1}{2}\left(\frac{1}{R_{33}^2} + \frac{1}{R_{11}^2} - \frac{1}{R_{22}^2}\right) \qquad H = \frac{1}{2}\left(\frac{1}{R_{11}^2} + \frac{1}{R_{22}^2} - \frac{1}{R_{33}^2}\right)$$

$$L = \frac{3}{2R_{23}^2},\ M = \frac{3}{2R_{31}^2},\ N = \frac{3}{2R_{12}^2}$$

HILL'S YIELD FUNCTION (CONT.)

- **The stress ratios of actual yield stresses in various material directions to the reference yield stress**

$$R_{11} = \frac{Y_1}{Y_a},\ R_{22} = \frac{Y_2}{Y_a},\ R_{33} = \frac{Y_3}{Y_a} \quad , \quad R_{12} = \frac{\sqrt{3}\,T_{12}}{Y_a},\ R_{23} = \frac{\sqrt{3}\,T_{23}}{Y_a},\ R_{31} = \frac{\sqrt{3}\,T_{31}}{Y_a}$$

- **where Y_a = average (or reference) yield stress in all directions (to be input as initial yield stress)**

MATEP	MID		Y0		HILL				
	ANISO		R11	R22	R33	R12	R23	R31	

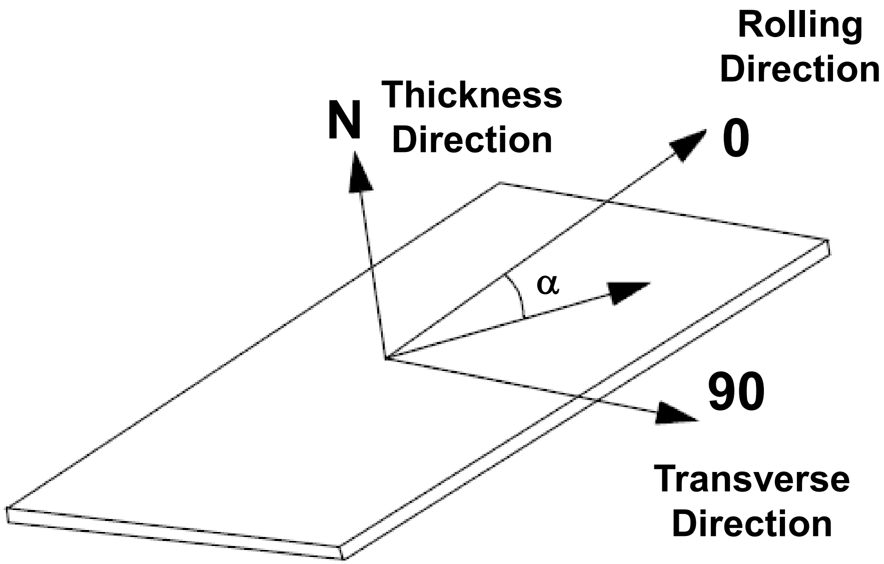
HILL’S YIELD FUNCTION (CONT.)

- Experimental data can be easily obtained for the shells or plane stress case
- Sample of tensile coupon cut from a sheet in the three directions, $\alpha = 0^\circ, 45^\circ$, and 90° is tested to obtain stresses

$$\sigma = \sigma_0, \sigma_{45}, \text{ and } \sigma_{90}$$

- Similarly, strain ratio is measured in the three directions

$$r = \frac{\varepsilon_{\text{width}}}{\varepsilon_{\text{thickness}}}$$



HILL'S YIELD FUNCTION (CONT.)

- The yield stress in the third (thickness) direction can be written as

$$\sigma_N = \sigma_0 \sqrt{\frac{r_{90}(1 + r_0)}{r_0 + r_{90}}} = \sigma_{90} \sqrt{\frac{r_0(1 + r_{90})}{r_0 + r_{90}}}$$

- The yield stress ratios are then calculated as

$$R_{11} = \frac{\sigma_0}{Y_a}, R_{22} = \frac{\sigma_{90}}{Y_a}, R_{33} = \frac{\sigma_N}{Y_a}$$

$$R_{12} = R_{33} \sqrt{\frac{3}{2r_{45} + 1}}$$

$$R_{23} = R_{31} = 1.0$$

BARLAT’S YIELD FUNCTION (1991)

- Planar anisotropic yield, suitable for aluminum alloy sheets. The yield function reads:

$$f = |S_1 - S_2|^m + |S_2 - S_3|^m + |S_3 - S_1|^m = 2\bar{\sigma}^m$$

- Si are the principal values of the following tensor, with the Cauchy stress tensor s:

$$S = \begin{bmatrix} \frac{C_3(\sigma_{xx} - \sigma_{yy}) - C_2(\sigma_{zz} - \sigma_{xx})}{3} & C_6\sigma_{xy} & C_5\sigma_{zx} \\ C_6\sigma_{xy} & \frac{C_1(\sigma_{yy} - \sigma_{zz}) - C_3(\sigma_{xx} - \sigma_{yy})}{3} & C_4\sigma_{zy} \\ C_5\sigma_{zx} & C_4\sigma_{zy} & \frac{C_2(\sigma_{zz} - \sigma_{xx}) - C_2(\sigma_{yy} - \sigma_{zz})}{3} \end{bmatrix}$$

x,y,z referring to the rolling, transverse and thickness direction of the sheet

BARLAT’S YIELD FUNCTION (Count.)

- **Material coefficients Ci represent anisotropic properties and m is an exponent**
- **Recommended m values are 6 for steel and 8 for aluminum**
- **C1, C2, C3 and C6 have to be defined using four stresses from the experimental data (C4=C5=1)**
- **When Ci=1, plastic deformation is isotropic**
  - m=1: Tresca yield function
  - m=2 or 4: von Mises yield function

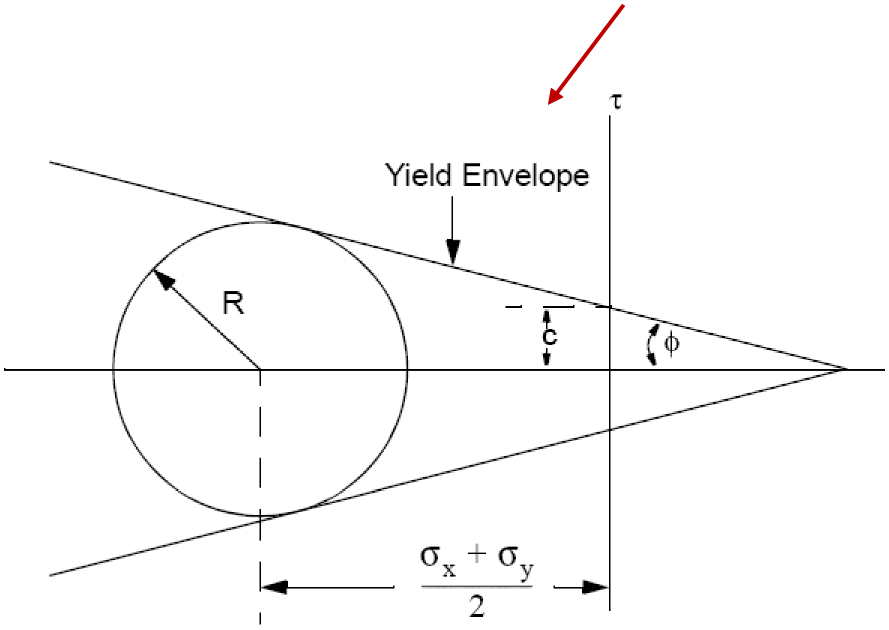
MATEP	MID		Y0		BARLAT				
	ANISO		M	C1	C2	C3	C6		

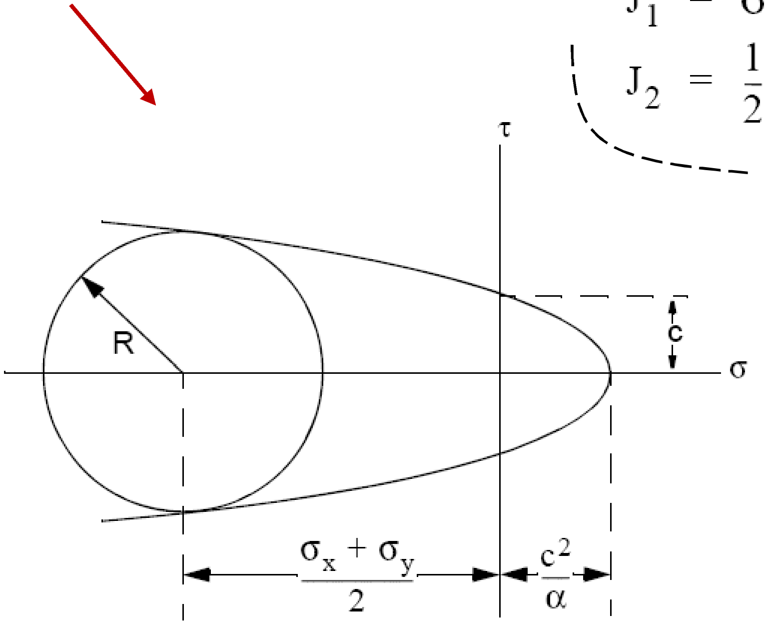
MOHR-COULOMB YIELD FUNCTION

- The yield stress is pressure dependent
- Version of Drucker-Prager is implemented
- The function can be linear or parabolic

$$J_1 = \sigma_{ii}$$

$$J_2 = \frac{1}{2}\sigma_{ij}^{d}\sigma_{ij}^{d}$$





$$f = \alpha J_1 + J_2^{1/2} - \frac{\bar{\sigma}}{\sqrt{3}} = 0$$

$$f = (3J_2 + \sqrt{3}\beta\bar{\sigma}J_1)^{1/2} - \bar{\sigma} = 0$$

MOHR-COULOMB (CONT.)

- **Required input date are $\bar{\sigma}$, α and β which are related to cohesion c and angle of friction Φ**

linear case: $c = \dfrac{\bar{\sigma}}{[3(1-12\alpha^2)]^{1/2}}$; $\dfrac{3\alpha}{(1-3\alpha^2)^{1/2}} = \sin\phi$

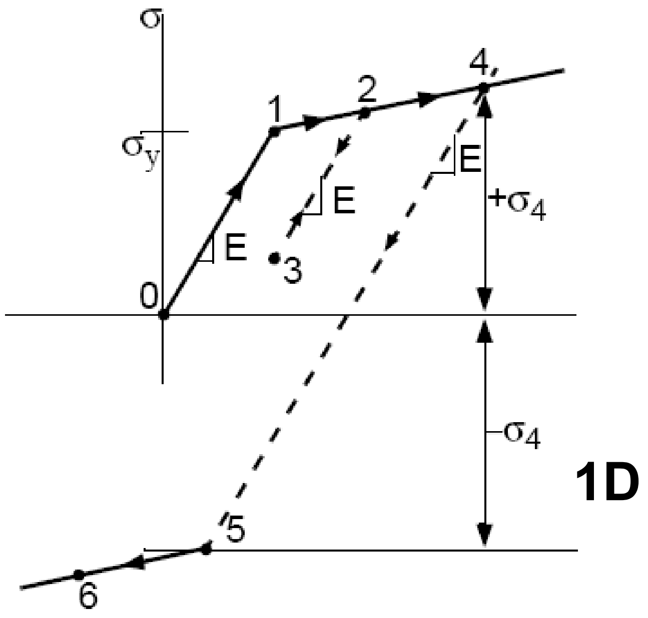
MATEP	MID		Y0		LINMOHR				
	PRESS	LIN	ALPHA						

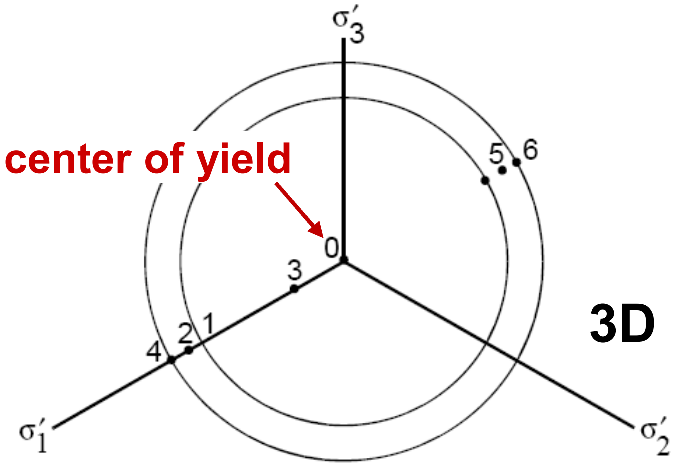
parabolic case: $\bar{\sigma}^2 = 3\left(c^2 - \dfrac{\alpha^2}{3}\right)$ $\beta = \dfrac{\alpha}{(3(3c^2-\alpha^2))^{1/2}}$

MATEP	MID		Y0		PBLMOHR				
	PRESS	PBL	ALPHA	BETA					

ISOTROPIC HARDENING

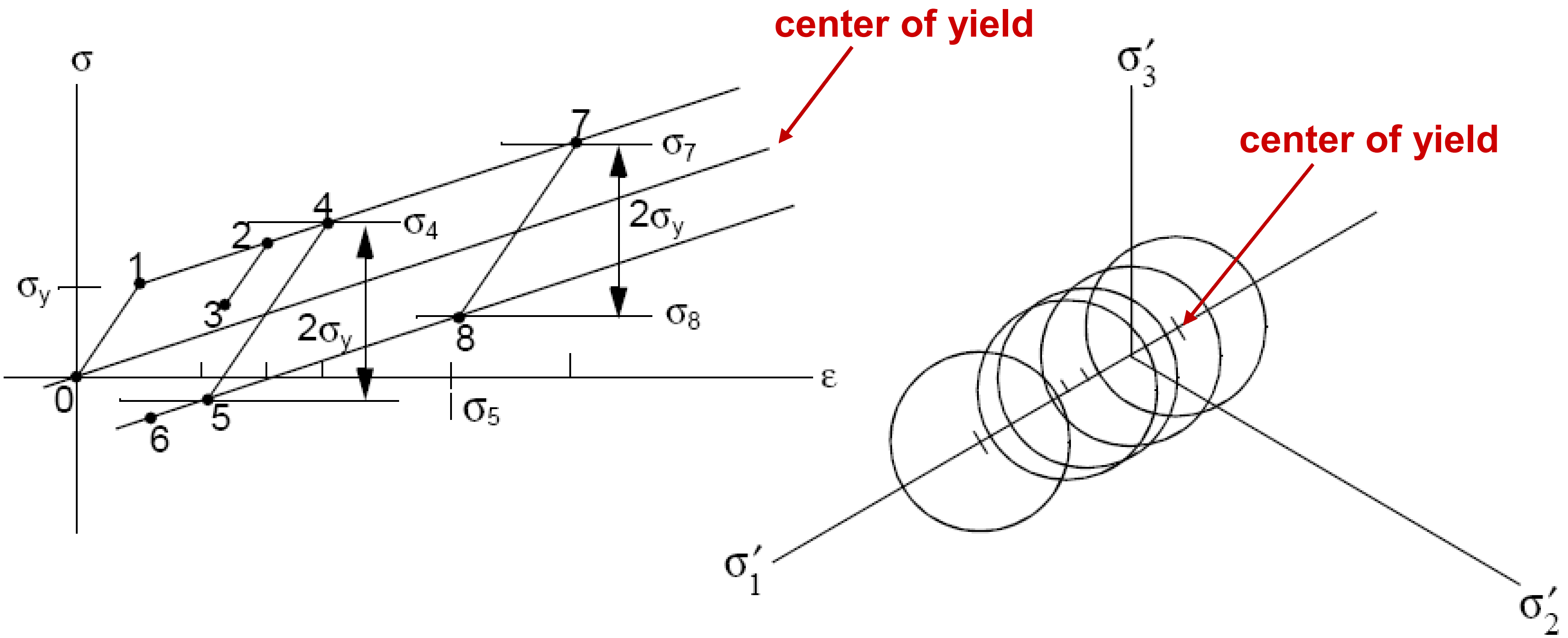
- **Point 1**
  - material starts yielding
- **Points 2 – 3**
  - hardening and elastic unloading
- **Points 3 – 2 – 4**
  - elastic loading followed by new hardening, yield stress becomes σ_4
- **Point 5**
  - load direction reversed
  - material behaves linear elastic until $-\sigma_4$ is being reached
- **Point 6**
  - for further negative loading the yield stress increases





KINEMATIC HARDENING

- **The center of yield moves in the stress space, but the surface remains the same**
  - this type of hardening is better suited when cyclic loading occurs

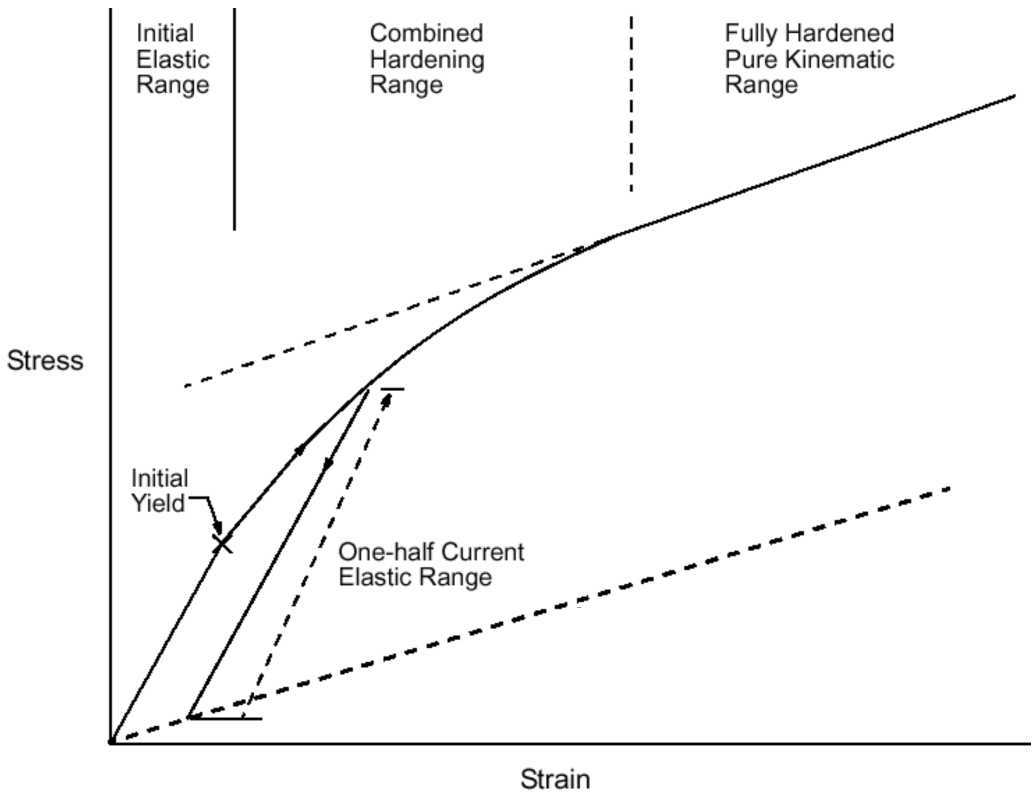


center of yield

center of yield

COMBINED HARDENING

- **This is a combination of isotropic and kinematic hardening which is also suited for cyclic load**



- **After the initial yield, the material starts hardening predominantly isotropic**
- **However, isotropic hardening decays as a function of the equivalent plastic strain**
- **After large plastic straining, the hardening rule becomes purely kinematic**
- **Applies to the yield functions of Hill and Barlat**

HARDENING INPUT

1	2	3	4	5	6	7	8	9	10
MATEP	MID	TABLE				WKHARD			

ISOTROP	**for isotropic hardening (Default)**
KINEM	**for kinematic hardening**
COMBINE	**for combination between kinematic and isotropic hardening**

HARDENING – EXAMPLE

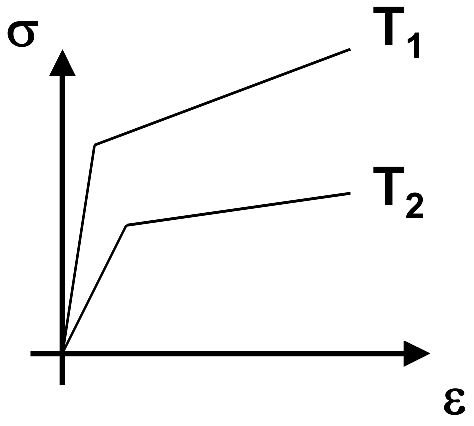
- **Isotropic Hardening**

```
BEGIN BULK
PARAM,  LGDISP, 1
NLMOPTS,LRGS,   1
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
MAT1    1       70500.          .342    1.
MATEP   1       TABLE   1       ISOTROP
TABLES1 1       2
+       0.      194.    .02     230.043 .04     268.496 .06     293.904
+       .08     313.378 .1      329.365 .2      384.423 .3      420.802
+       .4      448.681 .5      471.573 .6      491.14  .7      508.317
+       .8      523.682 .9      537.619 1.      550.399 1.1     562.224
+       1.2     573.239 1.3     583.564 1.4     593.287 ENDT
```

stress vs. plastic strain table

THERMO PLASTICITY

- **The stress-strain curve depends on temperature**
- **The change of stresses reads**



$$\dot{\sigma}_{ij} = L_{ijkl}\dot{\varepsilon}_{kl} + h_{ij}\dot{T}$$

with

$$L_{ijkl} = C_{ijkl} - \left(C_{ijmn}\frac{\partial\bar{\sigma}}{\partial\sigma_{mn}}\frac{\partial\bar{\sigma}}{\partial\sigma_{pq}}C_{pqkl}\right)/D$$

$$h_{ij} = X_{ij} - C_{ijkl}\boxed{\alpha_{kl}} - \left(C_{ijkl}\frac{\partial\bar{\sigma}}{\partial\sigma_{kl}}\left(\sigma_{pq}X_{pq} - \frac{2}{3}\bar{\sigma}\frac{\partial\bar{\sigma}}{\partial T}\right)\right)/D$$

change of yield stress

$$X_{ij} = \boxed{\frac{\partial C_{ijkl}}{\partial T}}\varepsilon^{e}_{kl}$$

change of elastic moduli

$$D = \frac{4}{9}\bar{\sigma}^2\frac{\partial\bar{\sigma}}{\partial\bar{\varepsilon}^p} + \frac{\partial\bar{\sigma}}{\partial\sigma_{ij}}C_{ijkl}\frac{\partial\bar{\sigma}}{\partial\sigma_{kl}}$$

α_{kl}: coefficients of thermal expansion

THERMO PLASTICITY – INPUT EXAMPLE

```
BEGIN BULK
PARAM,  LGDISP, 1
NLMOPTS,LRGS,   1
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
MAT1    1       2.0+5           0.3             1.2E-6
MATT1   1       7
TABLEM1 7
+       20.0    2.0+5   1000.   1.65+5  1500.   1.5+5   ENDT
MATEP   1       TABLE                           ISOTROP
MATTEP  1                       1
TABLEST 1
+       20.0    20      1000.   1000    1500.   1500    ENDT
TABLES1 20      2
+       0.      350.    0.01    360.    0.1     450.    1.      500.
+       2.0     500.    ENDT
TABLES1 1000    2
+       0.      330.    0.01    340.    0.1     430.    1.      480.
+       2.0     480.    ENDT
TABLES1 1500    2
+       0.      300.    0.01    310.    0.1     400.    1.      450.
+       2.0     450.    ENDT
```

temperature dependent Young's modulus

STRAIN RATE DEPENDENT PLASTICITY

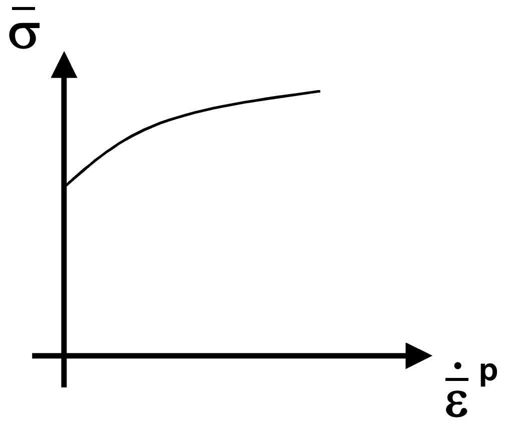
- **The yield stress depends on the strain rate**
- **The change of stresses reads**

$$\dot{\sigma}_{ij} = L_{ijkl}\dot{\varepsilon}_{kl} + r_{ij}\dot{\bar{\varepsilon}}^{p}$$

with

$$r_{ij} = C_{ijmn}\frac{\partial\bar{\sigma}}{\partial\sigma_{mn}}\frac{2}{3}\bar{\sigma}\boxed{\frac{\partial\bar{\sigma}}{\partial\dot{\bar{\varepsilon}}^{p}}}/D$$

change of yield stress due to equivalent plastic strain rate



STRAIN RATE DEPENDENCY – EXAMPLE

```
BEGIN BULK
PARAM,  LGDISP, 1
NLMOPTS,LRGS,   1
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
MAT1    1       2.0+5           0.3
MATEP   1       TABLE           1               ISOTROP
+       REFFECT TABLE   2
TABLES1 1       2
+       0.      350.    0.01    360.    0.1     450.    1.      500.
+       2.0     500.    ENDT
TABLES1 2
+       0.      350.    0.1     400.    1.0     500.    2.0     500.
+       ENDT
```

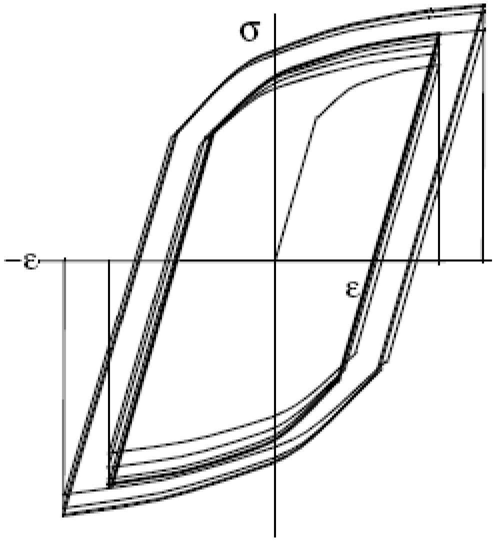
yield stress vs. plastic strain rate

stress-strain curve

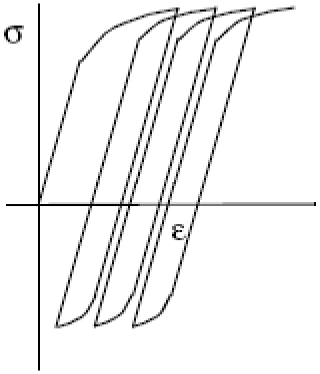
matches initial yield stress

CYCLIC PLASTICITY (CHABOCHE)

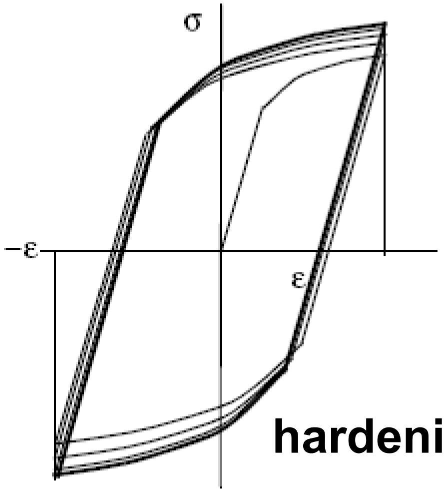
- The stress-strain curve mutates due to cyclic loading
- Several effects can be observed:



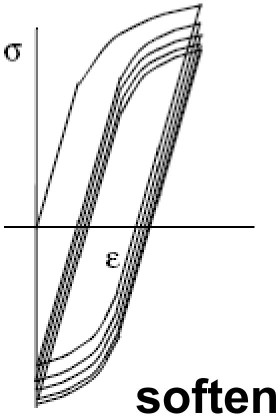
multiple loading



ratcheting



hardening



softening

CYCLIC PLASTICITY (CONT.)

- **Material Constants**
  - R_o, b, R_{oo}, C, g, Q_o, Q_m, m
- **Temperature Dependency (combine with MATTEP)**

MATEP	MID	Form	Y0	TID		Chaboche		H	
	Chaboche	R_0	R_{oo} or Q_o	b	C	γ	K	n	
		Q_m	μ	η					

+

MATTEP	MID	Form	T(Y0)	T(FID)		Chaboche		T(H)	
	Chaboche	$T(R_0)$	$T(R_{oo}/Q_o)$	T(b)	T(C)	$T(\gamma)$	T(K)	T(n)	
		$T(Q_m)$	$T(\mu)$	$T(\eta)$					

FLOW RULE

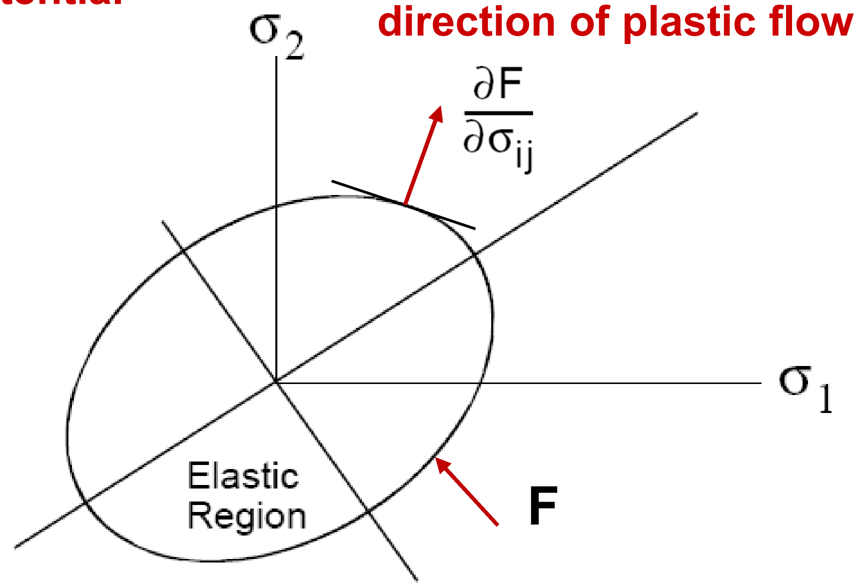
- **An associative flow rule is being used**
  - the flow potential equals the yield surface
  - when the material yields, the plastic strain is perpendicular to the yield surface

flow potential

$$d\varepsilon_{ij}^{P} = d\lambda \frac{\partial F}{\partial \sigma_{ij}}$$

sort of scale factor (determines the amount of plastic strain)

direction of plastic flow



LARGE STRAIN OPTIONS

- **NLMOPTS Bulk Data Card**

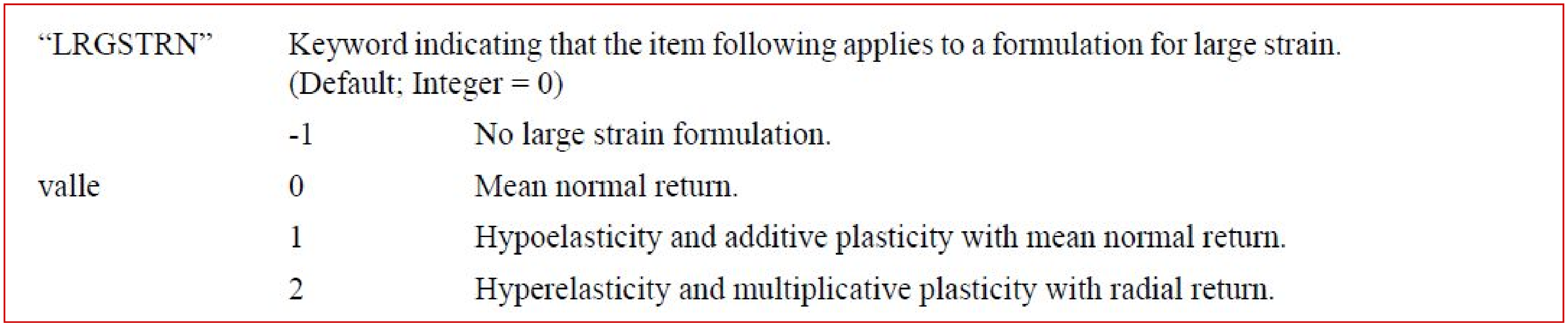
1	2	3	4	5	6	7	8	9	10
NLMOPTS	“CREEP”	valc1	valc2	valc3	valc4				
	“ASSM”	vala							
	“TSHEAR”	vals							
	“LRGSTRN”	valle							
	HEMICUBE	value	NPIXEL		CUTOFF	FRACTION	FACCNT	FACTOL	

Field	Contents
CREEP	Keyword indicating that the formulation for creep analysis. (Character Default CREEP)

•

•

•



“LRGSTRN”	Keyword indicating that the item following applies to a formulation for large strain. (Default; Integer = 0)		
	-1	No large strain formulation.	
valle	0	Mean normal return.	
	1	Hypoelasticity and additive plasticity with mean normal return.	
	2	Hyperelasticity and multiplicative plasticity with radial return.	

INITIAL PLASTIC STRAIN

- **Plastic strains at the beginning of an analysis**

1	2	3	4	5	6	7	8	9	10
IPSTRAIN	EID1	EID2	INT1	INTN	LAY1	LAYN	STRAIN		

- **Example:**

IPSTRAIN	2001	2020	1	4	1	5	0.025		

Field	Contents
EID1	First Element ID to which these strains apply. (Integer > 0)
EID2	Last Element ID to which these strains apply. (Integer; Default = EID1)
INT1	First Integration point for which the strain applies. (Integer > 0; Default = 1)
INTN	Last Integration point for which the strain applies. (Integer > 0, Default = 4)
LAY1	First element layer for which the strain applies. (Integer >0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
LAYN	Last element layer for which the strain applies. (Integer >0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
STRAIN	Equivalent plastic strain value at start of analysis. (Real; Default is 0.0)

INITIAL STRESS

- **Stresses at the beginning of an analysis**

1	2	3	4	5	6	7	8	9	10
ISTRESS	EID1	EID2	INT1	INTN	LAY1	LAYN	STRESS1	STRESS2	
	STRESS3	STRESS4	STRESS5	STRESS6	STRESS7				

Field	Contents
EID1	First Element ID to which these stresses apply. (Integer > 0)
EID2	Last Element ID to which these stresses apply. (Integer; Default = EID1)
INT1	First Integration point for which the stress applies. (Integer > 0; Default = 1)
INTN	Last Integration point for which the stress applies. (Integer > 0; Default = 4)
LAY1	First element layer for which the stress applies. (Integer >0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
LAYN	Last element layer for which the stress applies. (Integer >0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
STRESS(i)	Up to 7 stress components may be entered. (Real; Default = 0.0)

STRESS-STRAIN CURVE

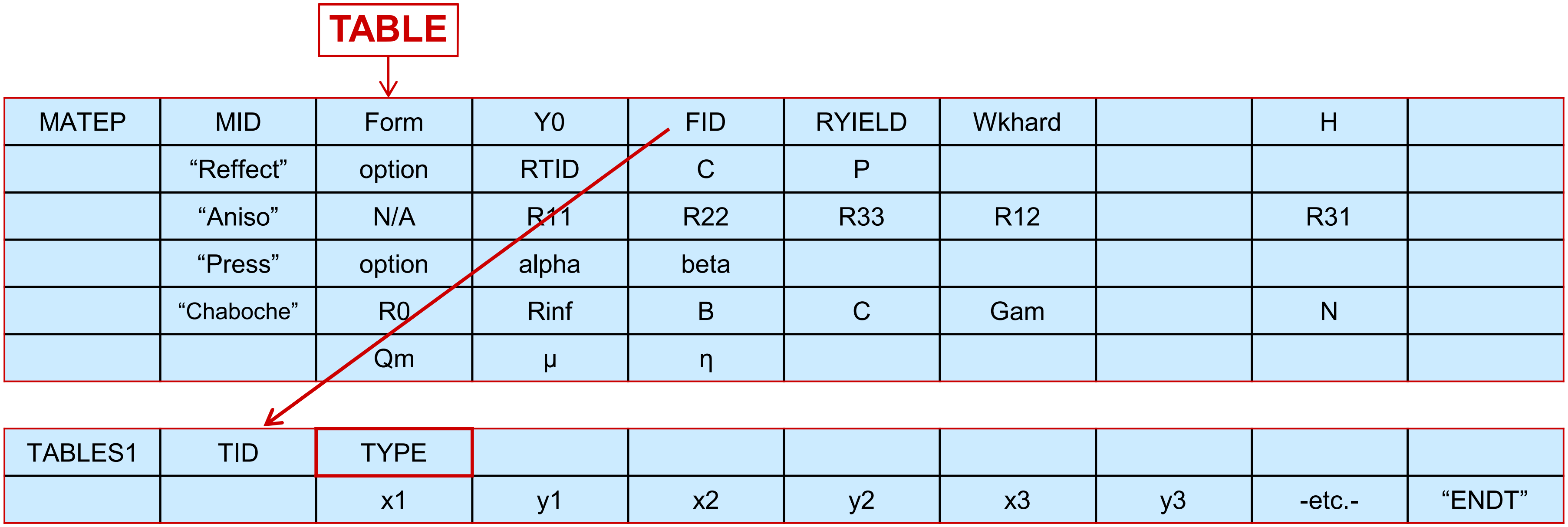
- Typically, laboratory measurements are expressed in engineering (nominal) stress and strain
- True (Cauchy) stress and true (Log) strain are required in SOL 400
- Engineering stress and strain can be converted to true stress and strain by:

$$\sigma_{true} = \sigma_{nom}(1 + \varepsilon_{nom})$$

$$\varepsilon_{true} = \ln(1 + \varepsilon_{nom})$$

STRESS-STRAIN CURVE (CONT.)

- **Stress-strain curve is entered in table form**
- **Two types of stress-strain curve can be used in SOL 400**
  - True stress vs true total strain (TYPE = 1, default)
  - True stress vs true plastic strain (TYPE = 2)



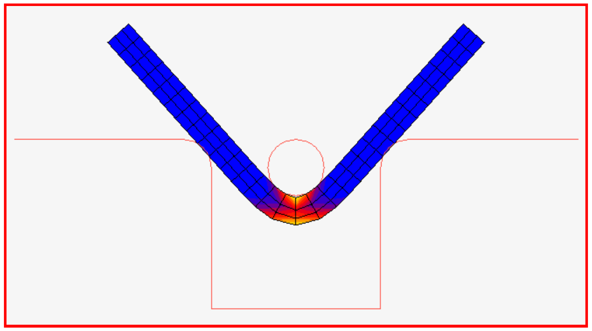
TABLE

MATEP	MID	Form	Y0	FID	RYIELD	Wkhard		H	
	"Reffect"	option	RTID	C	P				
	"Aniso"	N/A	R11	R22	R33	R12		R31	
	"Press"	option	alpha	beta					
	"Chaboche"	R0	Rinf	B	C	Gam		N	
		Qm	μ	η					

TABLES1	TID	TYPE							
		x1	y1	x2	y2	x3	y3	-etc.-	"ENDT"

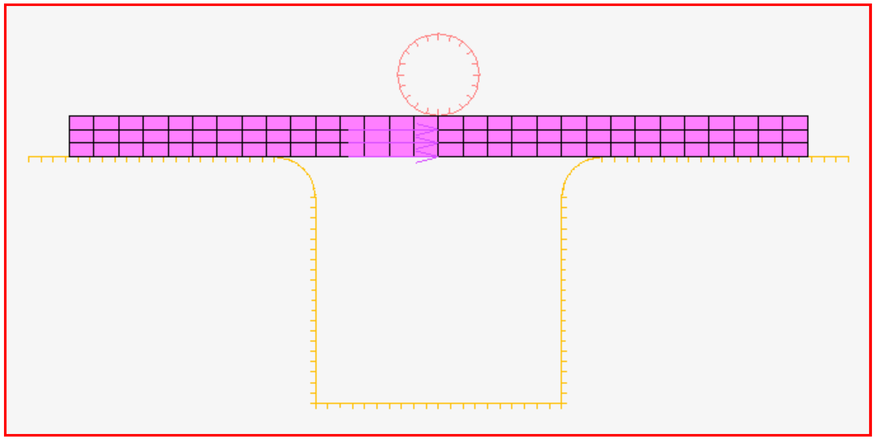
CASE STUDY - METAL FORMING

- **Model description**
  - A flat sheet is formed into an angled bracket by punching it through a hole in a table using the contact option
  - The cylindrical punch drives the sheet down into the hole of the table to a total stroke of 0.3 inches
  - The material is elastic-plastic with work-hardening
  - At the bottom of the stroke, the total strain is nearly 60% - a large strain problem



CASE STUDY - METAL FORMING (CONT.)

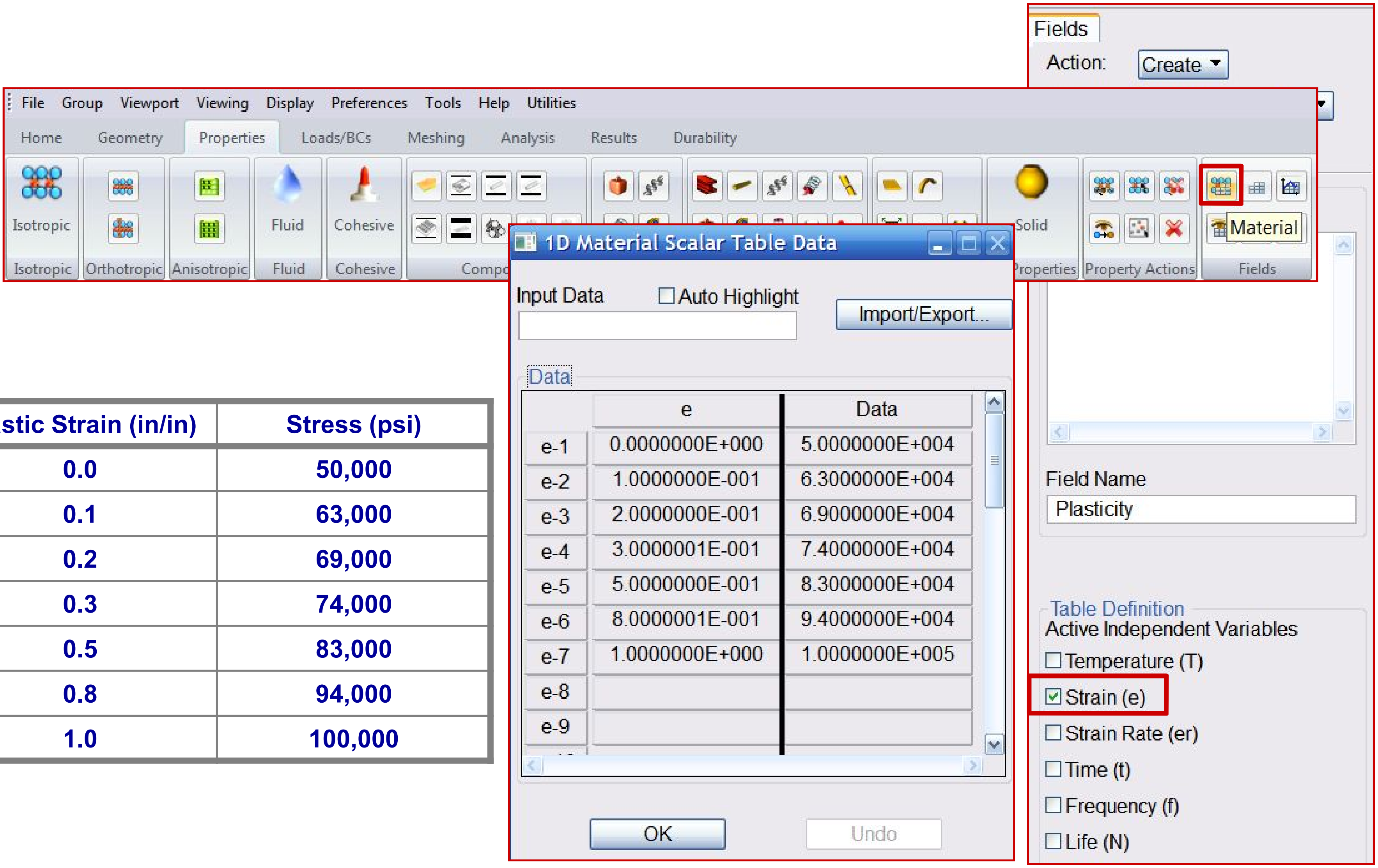
- **Model Review**
  - 2D (plane strain) model
  - Three contact bodies: plate (deformable), punch (rigid with motion control) and table (rigid)
  - The plate is constrained in x-direction
  - Elastic material is defined, need to add plasticity



CASE STUDY - METAL FORMING (CONT.)

- **Create material field**

Plastic Strain (in/in)	Stress (psi)
0.0	50,000
0.1	63,000
0.2	69,000
0.3	74,000
0.5	83,000
0.8	94,000
1.0	100,000



1D Material Scalar Table Data

Input Data ☐ Auto Highlight

Data

	e	Data
e-1	0.0000000E+000	5.0000000E+004
e-2	1.0000000E-001	6.3000000E+004
e-3	2.0000000E-001	6.9000000E+004
e-4	3.0000001E-001	7.4000000E+004
e-5	5.0000000E-001	8.3000000E+004
e-6	8.0000001E-001	9.4000000E+004
e-7	1.0000000E+000	1.0000000E+005
e-8		
e-9		

Fields

Action: Create

Field Name: Plasticity

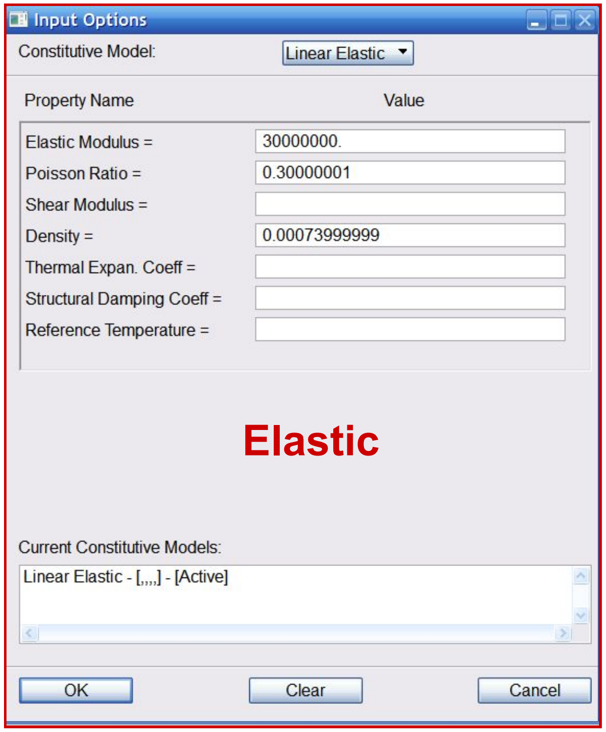
Table Definition

Active Independent Variables

☐ Temperature (T)
☑ Strain (e)
☐ Strain Rate (er)
☐ Time (t)
☐ Frequency (f)
☐ Life (N)

CASE STUDY - METAL FORMING (CONT.)

- **Add material plasticity**
  - stress-strain curve
  - von Mises yield criterion
  - isotropic hardening



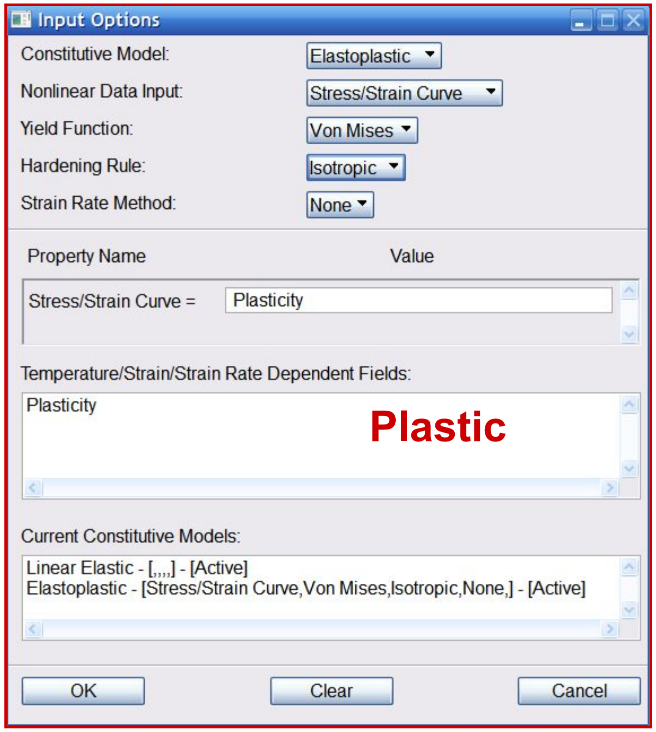
Constitutive Model: Linear Elastic

Property Name	Value
Elastic Modulus =	30000000.
Poisson Ratio =	0.30000001
Shear Modulus =	
Density =	0.00073999999
Thermal Expan. Coeff =	
Structural Damping Coeff =	
Reference Temperature =	

Current Constitutive Models:

Linear Elastic - [,,,,] - [Active]

Elastic



Constitutive Model: Elastoplastic

Nonlinear Data Input: Stress/Strain Curve

Yield Function: Von Mises

Hardening Rule: Isotropic

Strain Rate Method: None

Property Name	Value
Stress/Strain Curve =	Plasticity

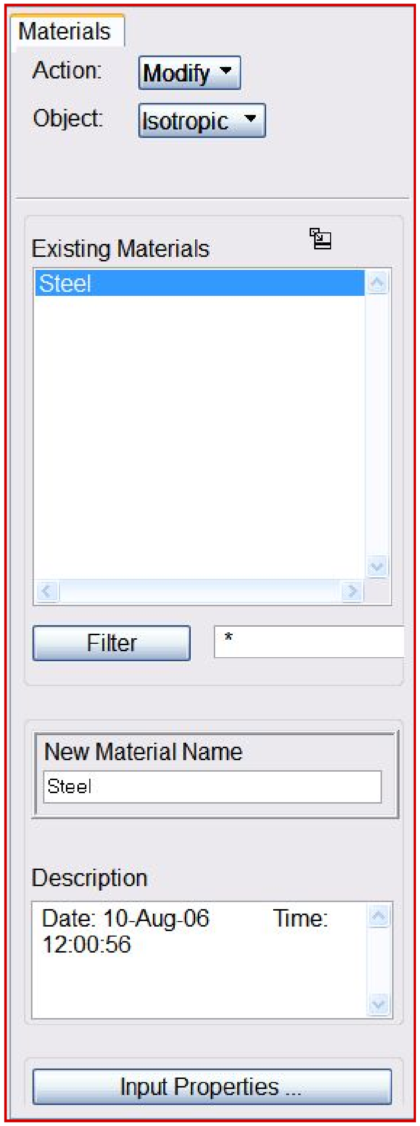
Temperature/Strain/Strain Rate Dependent Fields:

Plasticity

Current Constitutive Models:

Linear Elastic - [,,,,] - [Active]
Elastoplastic - [Stress/Strain Curve,Von Mises,Isotropic,None,] - [Active]

Plastic



Action: Modify

Object: Isotropic

Existing Materials

Steel

*

New Material Name

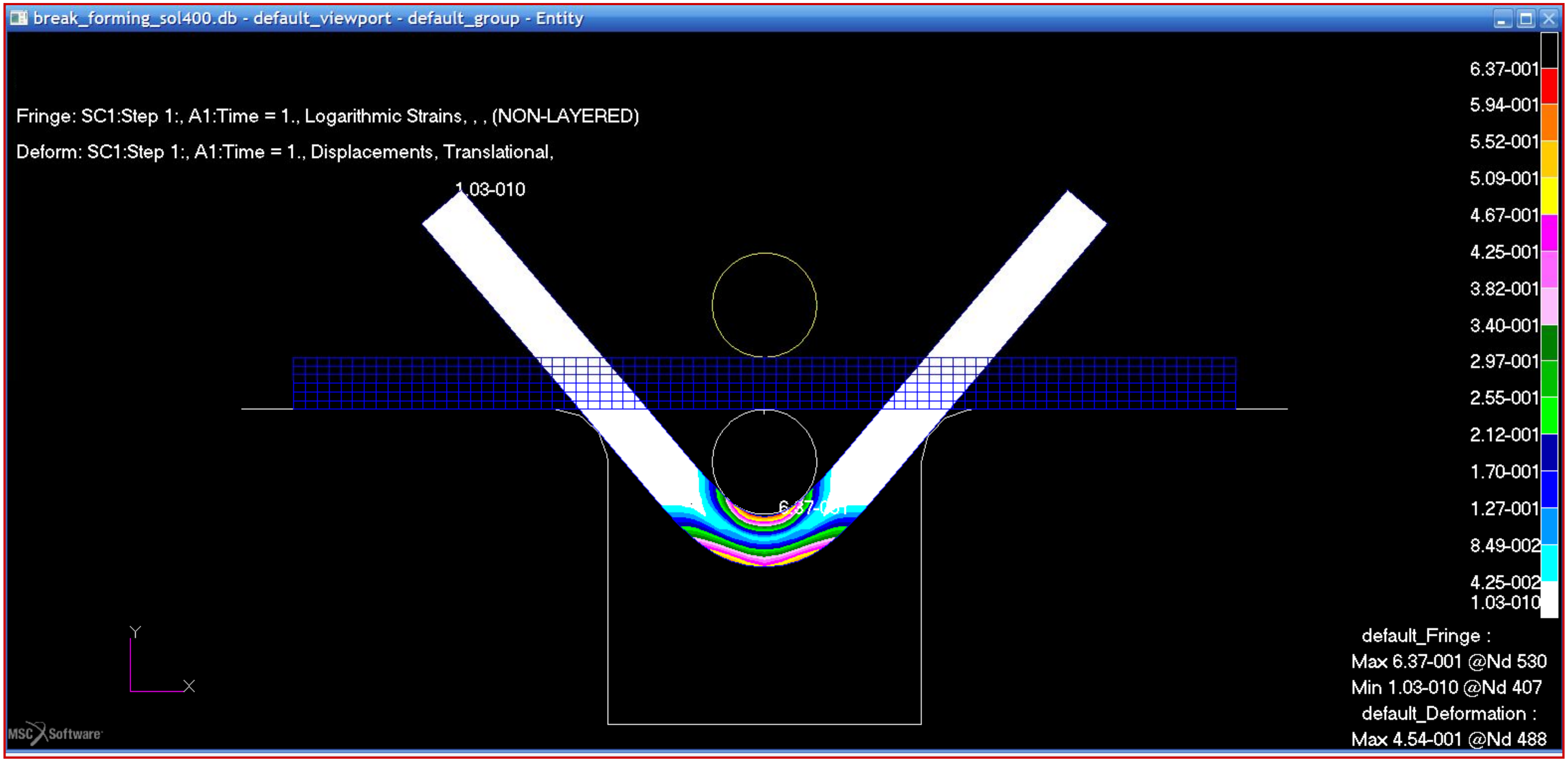
Steel

Description

Date: 10-Aug-06 Time: 12:00:56

CASE STUDY - METAL FORMING (CONT.)

- **Run the job and post-process results**



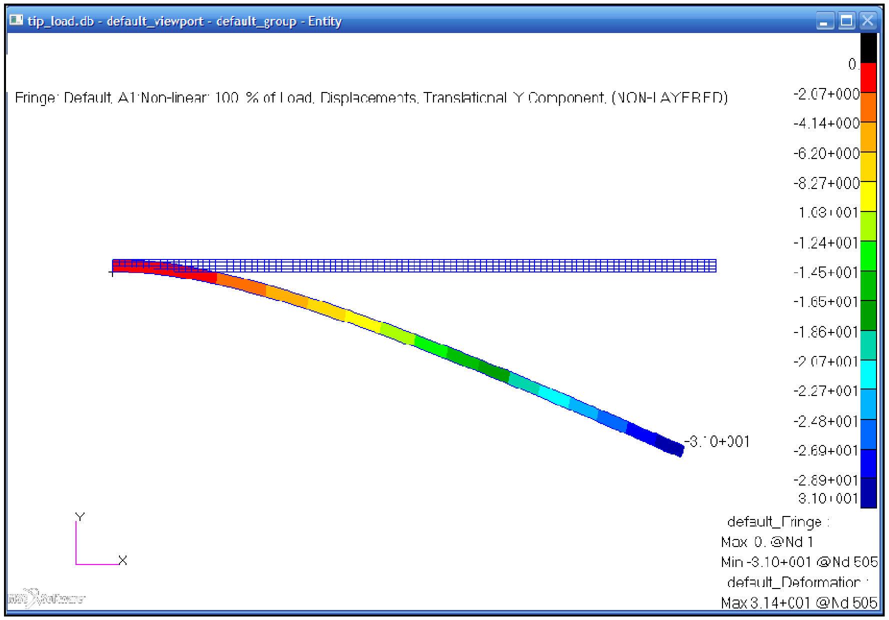
break_forming_sol400.db - default_viewport - default_group - Entity

Fringe: SC1:Step 1:, A1:Time = 1., Logarithmic Strains, , , (NON-LAYERED)

Deform: SC1:Step 1:, A1:Time = 1., Displacements, Translational,

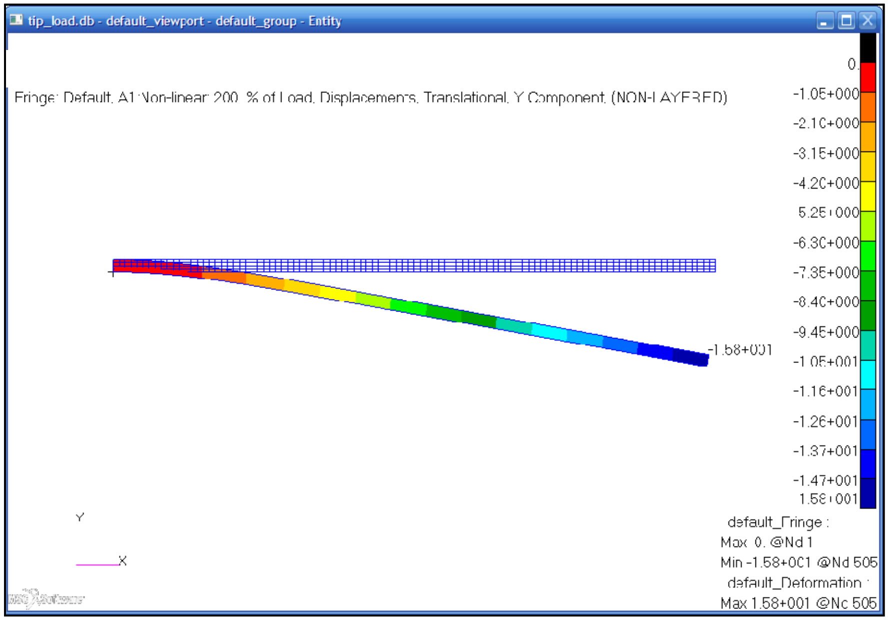
default_Fringe :
Max 6.37-001 @Nd 530
Min 1.03-010 @Nd 407
default_Deformation :
Max 4.54-001 @Nd 488

WORKSHOP 6
PLASTIC DEFORMATION



Fringe: Default, A1:Non-linear: 100. % of Load, Displacements, Translational, Y Component, (NON-LAYERED)

default_Fringe :
Max 0. @Nd 1
Min -3.10+001 @Nd 505
default_Deformation :
Max 3.14+001 @Nd 505



Fringe: Default, A1:Non-linear: 200. % of Load, Displacements, Translational, Y Component, (NON-LAYERED)

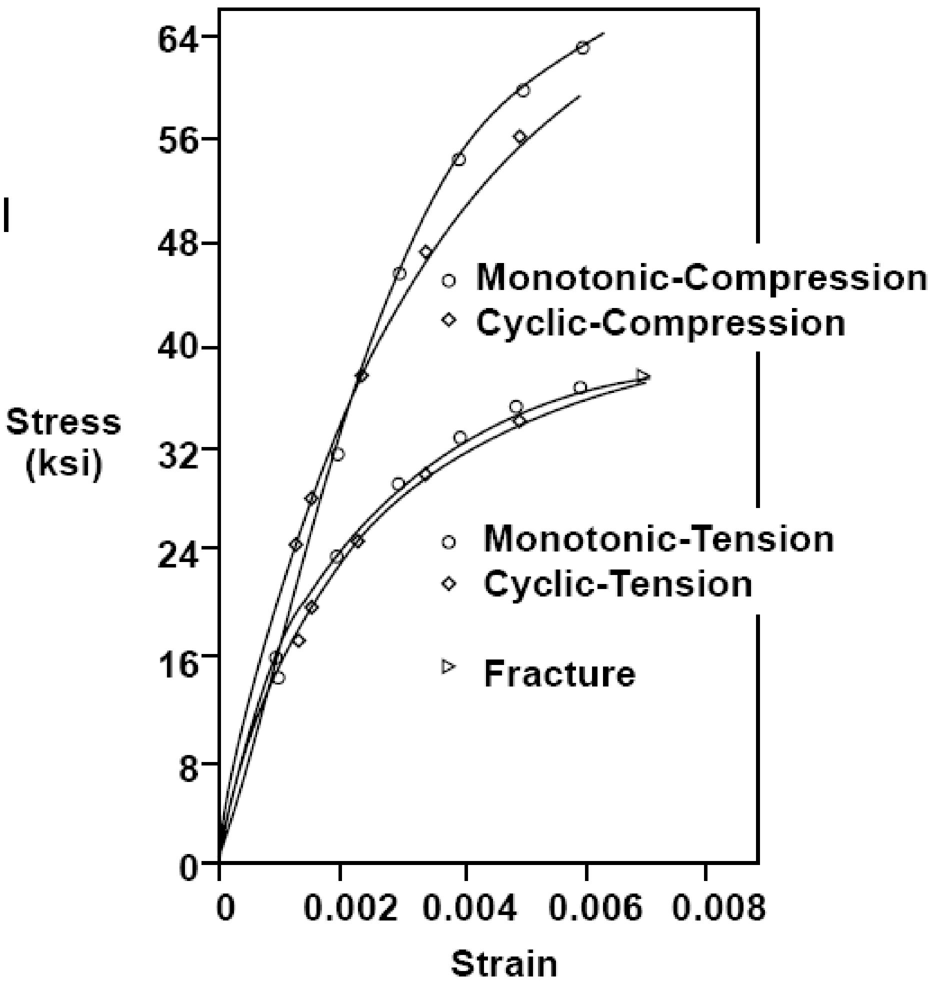
default_Fringe :
Max 0. @Nd 1
Min -1.58+001 @Nd 505
default_Deformation :
Max 1.58+001 @Nd 505

NONLINEAR ELASTIC MATERIAL

BASICS OF NLELAST MATERIAL

- **Applications**
  - Plastics
  - Metals
- **Loading & Unloading**
  - Different to elastic-plastic material the loading & unloading curves remain the same
  - However, tension and compression curves may be different
  - Should be used for small strains only (e < 10%)

Example of Gray Case Iron



APPLICABILITY OF NLELAST

- **isotropic material – all advanced nonlinear elements**

MATS1	MID	TID	NLELAST						

- **orthotropic – adv. axisymmetric solid**

MATS3	MID	TEX	TETH	TEZ	TNUXTH	TNUTHZ	TNUZX	TRHO	
			TGZX	TAX	TATH	TAZ			

- **orthotropic – adv. plane strain, stress, 3D solid**

MATSORT	MID	TE1	TE2	TE3	TNU12	TNU23	TNU31	TRHO	
	TG12	TG23	TG31	TA1	TA2	TA3			

- **orthotropic – adv. shell**

MATS8	MID	TE1	TE2	TNU12	TG12	TG1Z	TG2Z	TRHO	
	TA1	TA2							

T stands for TABLES1, TABLEST or TABL3Di

MATS1 INPUT EXAMPLES

- **Typical input of a nonlinear stress-strain curve**

```
BEGIN BULK
PARAM,  LGDISP, 1
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
MAT1    1       35000.          0.3
MATS1   1       10      NLELAST
TABLES1 10
+       -0.02   -400.   -0.015  -375.   -0.01   -300.   -0.005  -175.
+       0.0     0.0     0.02    200.    ENDT
```

strain – stress – data

MATS1 INPUT EXAMPLES (CONT.)

- **Temperature dependent stress-strain curve**

```
BEGIN BULK
PARAM,  LGDISP, 1
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
MAT1    1       35000.          0.3
MATS1   1       1       NLELAST
TABLEST 1
+       20.0    20      100.    100     200.    200     300.    300
+       ENDT
TABLES1 20
+       -0.02   -400.   -0.015  -375.   -0.01   -300.   -0.005  -175.
+       0.0     0.0     0.02    200.    ENDT
TABLES1 100
+       -0.02   -300.   -0.015  -281.25 -0.01   -225.   -0.005  -131.25
+       0.0     0.0     0.02    150.    ENDT
TABLES1 200
+       -0.02   -200.   -0.015  -187.5  -0.01   -150.   -0.005  -87.5
+       0.0     0.0     0.02    100.    ENDT
TABLES1 300
+       -0.02   -100.   -0.015  -93.75  -0.01   -75.    -0.005  -43.75
+       0.0     0.0     0.02    50.     ENDT
```

temperature

strain – stress – data

MATORT INPUT EXAMPLES

- **Typical input of orthotropic nonlinear elastic materials**

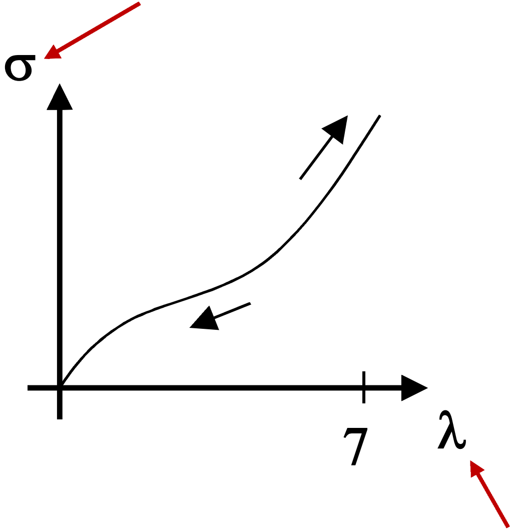
```
BEGIN BULK
PARAM,  LGDISP, 1
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
MATORT  1       35000.  35000.  3500.   0.3     0.3     0.3
+       13461.5 13461.5 1346.15
MATSORT 1       1       1       2
+       3       3       4
TABL3D0 1               69
+       -0.02   5000.   -0.015  15000.  -0.01   25000.  -0.005  35000.
+       0.0     22500.  0.02    10000.  1.0     10000.  ENDT
TABL3D0 2               69
+       -0.02   500.    -0.015  1500.   -0.01   2500.   -0.005  3500.
+       0.0     2250.   0.02    1000.   1.0     1000.   ENDT
TABL3D0 3               69
+       -0.02   1923.1  -0.015  5769.2  -0.01   9615.4  -0.005  13461.5
+       0.0     8653.9  0.02    3846.2  1.0     3846.2  ENDT
TABL3D0 4               69
+       -0.02   192.31  -0.015  576.92  -0.01   961.54  -0.005  1346.15
+       0.0     865.39  0.02    384.62  1.0     384.62  ENDT
```

strain

ADVANCED HYPERELASTIC MATERIAL

BASICS OF HYPERELASTIC MATERIAL

- **Applications**
  - Rubber & elastomers:
    - O-rings, bushings, gaskets, seals, boots, tires
  - Plastics
  - Glass
  - Solid propellant
- **Material Behavior**
  - large strain nonlinear elastic
  - incompressible or nearly
  - incompressible
  - loading = unloading



Cauchy stress

stretch

VOLUMETRIC LOCKING

- Due to incompressibility ($\nu \approx 0.5$), standard formulation would lead to volumetric locking or even instability

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \tau_{12} \\ \tau_{23} \\ \tau_{31} \end{Bmatrix} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{1-\nu} & \frac{\nu}{1-\nu} & 0 & 0 & 0 \\ & 1 & \frac{\nu}{1-\nu} & 0 & 0 & 0 \\ & & 1 & 0 & 0 & 0 \\ & \text{SYM} & & \frac{1-2\nu}{2(1-\nu)} & 0 & 0 \\ & & & & \frac{1-2\nu}{2(1-\nu)} & 0 \\ & & & & & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \\ \gamma_{12} \\ \gamma_{23} \\ \gamma_{31} \end{Bmatrix}$$

Singular!

STRAIN ENERGY FUNCTION

- **Therefore for nearly-incompressible materials strain energy functions are used, like:**

strain energy

polynomial coefficients

bulk modulus

$$W = W_{deviatoric} + W_{volumetric} = \sum_{m=1}^{N} \sum_{n=1}^{N} C_{mn} (\dot{I}_1 - 3)^m (\dot{I}_2 - 3)^n + \frac{9K}{2}\left(J^{\frac{1}{3}} - 1\right)^2$$

deviatoric stress invariants

$J = \lambda_1 \lambda_2 \lambda_3$ (Jacobian)

λ_i (principal stretches)

- **Stresses are calculated from:**

2nd Piola Kirchhoff

Green Lagrange

$$S_{ij} = \frac{\partial W}{\partial E_{ij}}$$

Total Lagrangian Formulation

Cauchy

Left Cauchy Green

$$\sigma_{ij} = \frac{2}{J} \frac{\partial W}{\partial b_{ik}} b_{kj}$$

Updated Lagrangian Formulation

MATERIAL MODELS

- Mooney-Rivlin Model
- Ogden Model
- Arruda-Boyce[1]
- Gent[1]

[1]Arruda-Boyce and Gent yield better results when only a tension test is available

MOONEY-RIVLIN MODEL

- **Generalized Mooney-Rivlin, as used in SOL 400**

$$W^{tod}_{devratoric} = C_{10}(\dot{I}_1 - 3) + C_{01}(\dot{I}_2 - 3) + C_{11}(\dot{I}_1 - 3)(\dot{I}_2 - 3) + C_{20}(\dot{I}_1 - 3)^2 + C_{30}(\dot{I}_1 - 3)^3$$

- **Format 1 (Default): Generalized Mooney-Rivlin model (Model = Mooney)**

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model	N/A	K	RHO	Texp	Tref	GE	
	C10	C01	N/A	TAB1	TAB2	TAB3	TAB4	TABD	
	C20	C11	N/A	N/A	N/A	N/A			
	C30	N/A	N/A	N/A	N/A	N/A			

- **When a subset of coefficients is being used:**

$$W^{nh}_{deviatoric} = C_{10}(\dot{I}_1 - 3) \qquad \text{Neo-Hookean}$$

$$W^{mr}_{deviatoric} = C_{10}(\dot{I}_1 - 3) + C_{01}(\dot{I}_2 - 3) \qquad \text{Mooney-Rivlin}$$

MOONEY-RIVLIN MODEL (CONT.)

- **Instead of the coefficients, test data can be input**

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model	N/A	K	RHO	Texp	Tref	GE	
	C10	C01	N/A	TAB1	TAB2	TAB3	TAB4	TABD	
	C20	C11	N/A	N/A	N/A	N/A			
	C30	N/A	N/A	N/A	N/A	N/A			

- **Up to 4 tests for deviatoric part and one for volumetric part are possible**
  - TAB1, simple tension
  - TAB2, equal biaxial
  - TAB3, simple shear
  - TAB4, pure shear
  - TABD, volumetric

Input via TABLES1

Attention: not available for MATHE, use MATHP instead

MOONEY-RIVLIN MODEL (CONT.)

- **Mooney-Rivlin – Input Example**

```
BEGIN BULK
PARAM,  LGDISP, 1
NLMOPTS,LRGS,   2
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
MATHE   1       MOONEY                                  1.-6
+       0.5     0.125
```

K=blank for fully incompressible

OGDEN MODEL

- **Strain Energy Function**

$$W_{deviatoric}^{ogden} = \sum_{k=1}^{N} \frac{\mu_k}{\alpha_k}\left(\bar{\lambda}_1^{\alpha_k} + \bar{\lambda}_2^{\alpha_k} + \bar{\lambda}_3^{\alpha_k} - 3\right)$$

with: $\bar{\lambda}_i^{\alpha_k} = J^{-\frac{\alpha_k}{3}} \lambda_i^{\alpha_k}$ ($\bar{\lambda}_i$ are the deviatoric stretches)

- **Format 2: Ogden Model or Foam Model**

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model	NOT	K	RHO	Texp	Trep	GE	
	Mu1	Alpha1	Beta1	TAB1	TAB2	TAB3	TAB4	TABD	
	Mu2	Alpha2	Beta2	Mu3	Alpha3	Beta3			
	Mu4	Alpha4	Beta4	Mu5	Alpha5	Beta5			

FOAM MODEL

- **Strain Energy Function**

$$W = \sum_{n=1}^{N} \frac{\mu_n}{\alpha_n}\left(\lambda_1^{\alpha_n} + \lambda_2^{\alpha_n} + \lambda_3^{\alpha_n} - 3\right)$$

$$+ \sum_{n=1}^{N} \frac{\mu_n}{\beta_n}\left(1 - J^{\beta_n}\right)$$

- **Format 2: Ogden Model or Foam Model**

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model	NOT	K	RHO	Texp	Trep	GE	
	Mu1	Alpha1	Beta1	TAB1	TAB2	TAB3	TAB4	TABD	
	Mu2	Alpha2	Beta2	Mu3	Alpha3	Beta3			
	Mu4	Alpha4	Beta4	Mu5	Alpha5	Beta5			

ARRUDA-BOYCE AND GENT MODELS

- **Strain Energy Function (Arruda-Boyce)**

$$W_{dev}^{Arruda\text{-}Boyce} = nk\Theta\left[\frac{1}{2}(\dot{I}_1 - 3) + \frac{1}{20N}(\dot{I}_1^2 - 9) + \frac{11}{1050N^2}(\dot{I}_1^3 - 27) + \frac{19}{7000N^3}(\dot{I}_1^4 - 81) + \frac{519}{673750N^4}(\dot{I}_1^5 - 243)\right]$$

NKT
- **n, chain density**
- **k, Boltzmann constant**
- **θ, temperature**
- **N, number of statistical links**

- **Strain Energy Function (Gent)**

$$W_{dev}^{Gent} = \frac{-EI_m}{6}\log\left(1 - \frac{I_1^*}{I_m}\right)$$ **with** $$I_1^* = \dot{I}_1 - 3$$

- **Format 3: Arruda-Boyce model or Gent Model (Model = Aboyce or Gent)**

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model	N/A	K	RHO	Texp	Tref	GE	
	NKT	N/E	Im	TAB1	TAB2	TAB3	TAB4	TABD	

TEMPERATURE DEPENDENT MATHE

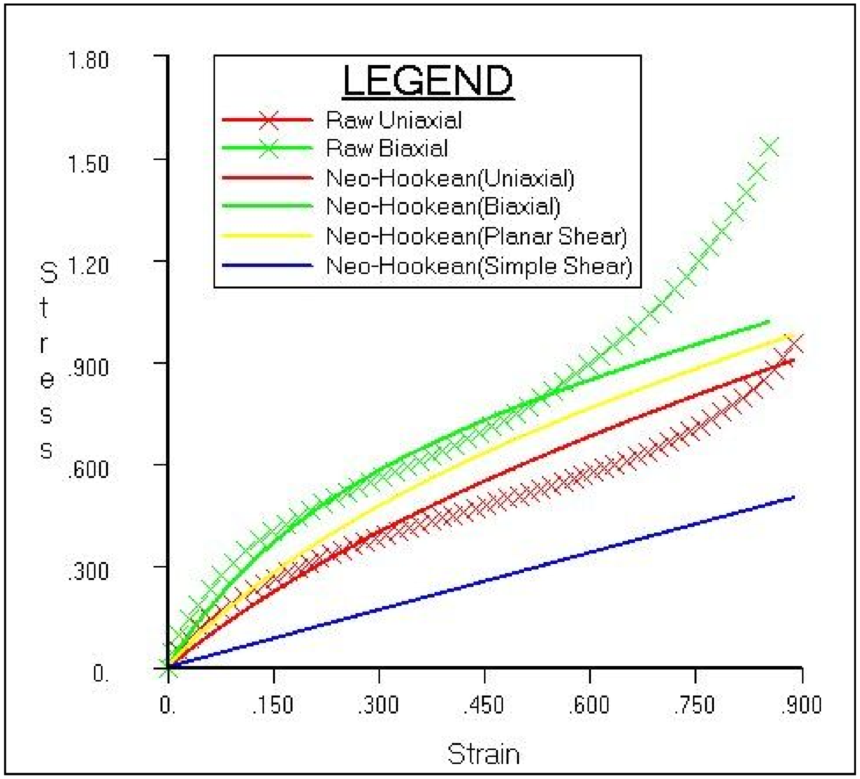
- **Input via MATTHE**

1	2	3	4	5	6	7	8	9	10
MATTHE	MID	N/A	N/A	N/A	N/A	T(Texp)	N/A	N/A	
	T(XX)	T(YY)	T(ZZ)	T(TAB1)	T(TAB2)	T(TAB3)	T(TAB4)	T(TABD)	

- **T(XX) – ID of TABLEMi, applicable to Mooney, Aboyce option depending on the MATHE "Model" field**
  - for MOONEY it is C10, for ABOYCE it is NKT
- **T(YY) – ID of TABLEMi, applicable to Mooney, Aboyce, or Gent option depending on the MATHE "Model" field**
  - for MOONEY it is C01, for ABOYCE it is N, for GENT is E
- **T(ZZ) – ID of TABLEMi, applicable to Gent option depending on the MATHE "Model" field**
  - for GENT it is LM

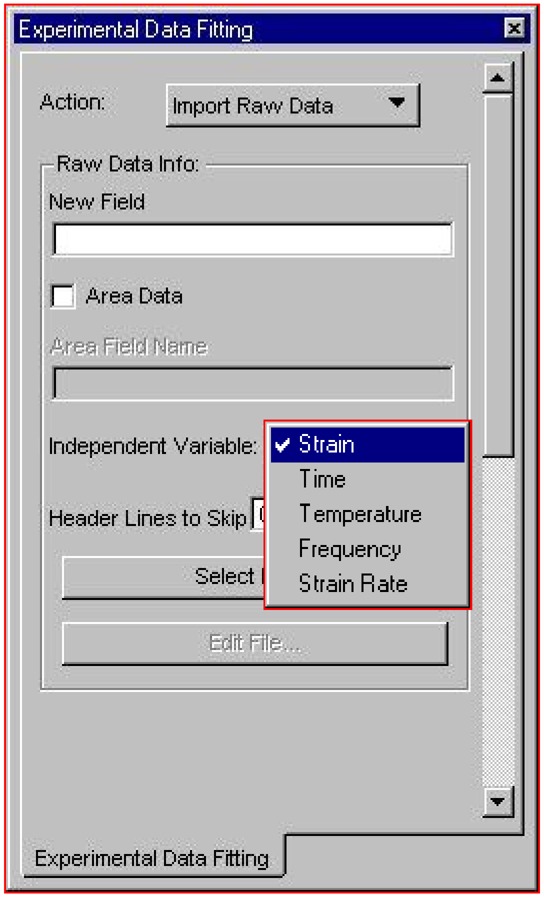
EXPERIMENTAL DATA FITTING

- **Material constants are usually obtained from experimental data fitting**
- **Patran provides a easy graphical interface**
- **Required testing data must are engineering stresses vs engineering strains**



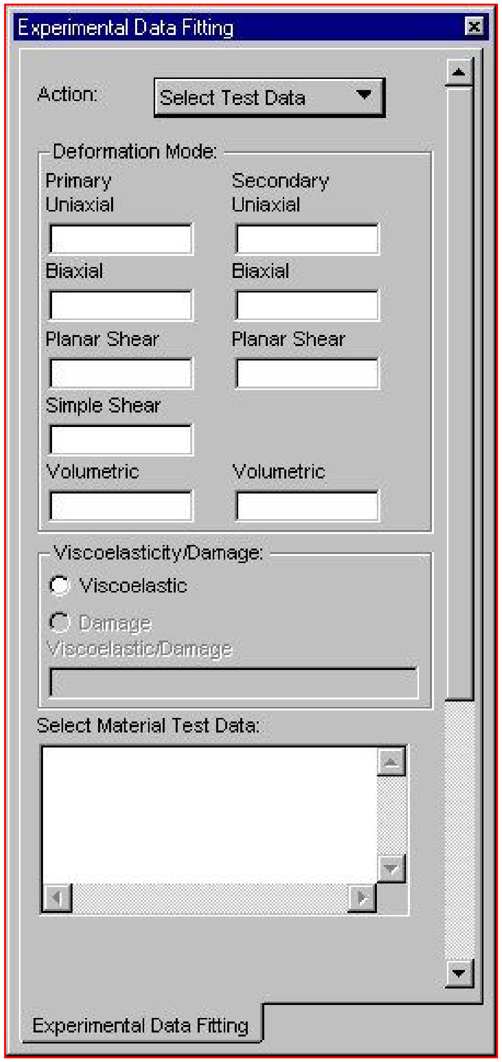
EXPERIMENTAL DATA FITTING (CONT.)

- **Read In Raw Material Data**
  - Import Raw Data as Fields
    - Strain
    - Strain Rate
    - Time
    - Frequency
    - Temperature
  - CSV Files Supported



EXPERIMENTAL DATA FITTING (CONT.)

- **Select Fields and Associate to Tests**
  - Uniaxial / Biaxial
  - Planar / Simple Shear
  - Volumetric
  - Visco-elastic Relaxation (Bulk, Shear, Energy)

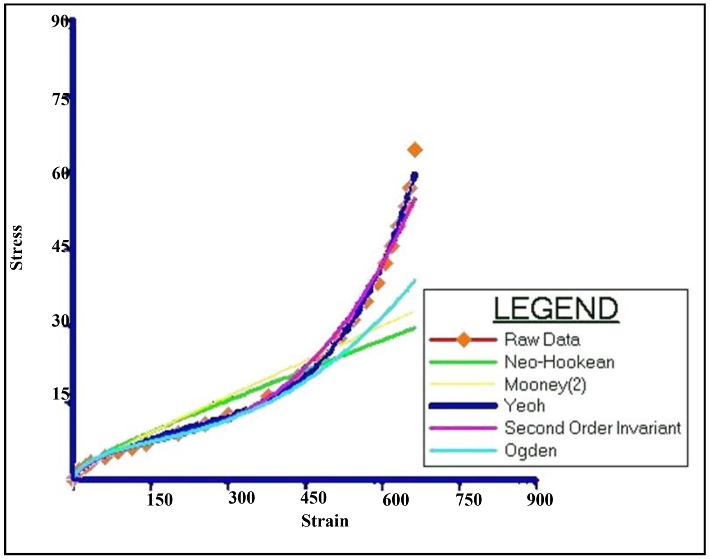


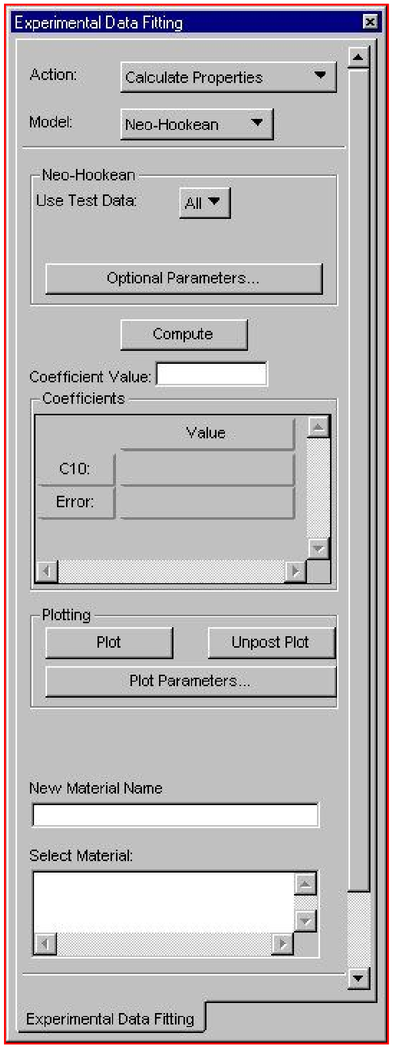
Experimental Data Fitting

Action: Select Test Data

EXPERIMENTAL DATA FITTING (CONT.)

- **Calculate Properties based on Selected Material Models**
  - Neo-Hookean / Mooney / Yeoh / Ogden / Gent
  - Shear / Bulk / Energy Relaxation (Viscoelastic)
- **Plot the Curve Fit Results and Save Constitutive Model into Database**



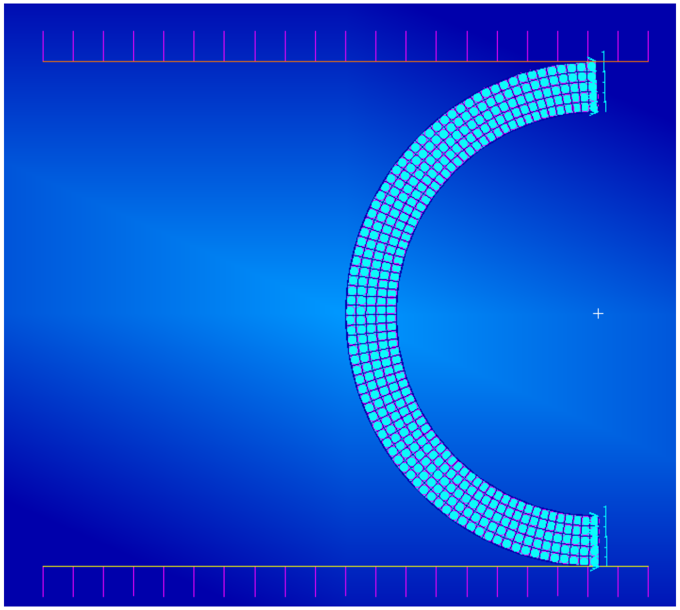

Action: Calculate Properties

Model: Neo-Hookean

Use Test Data: All

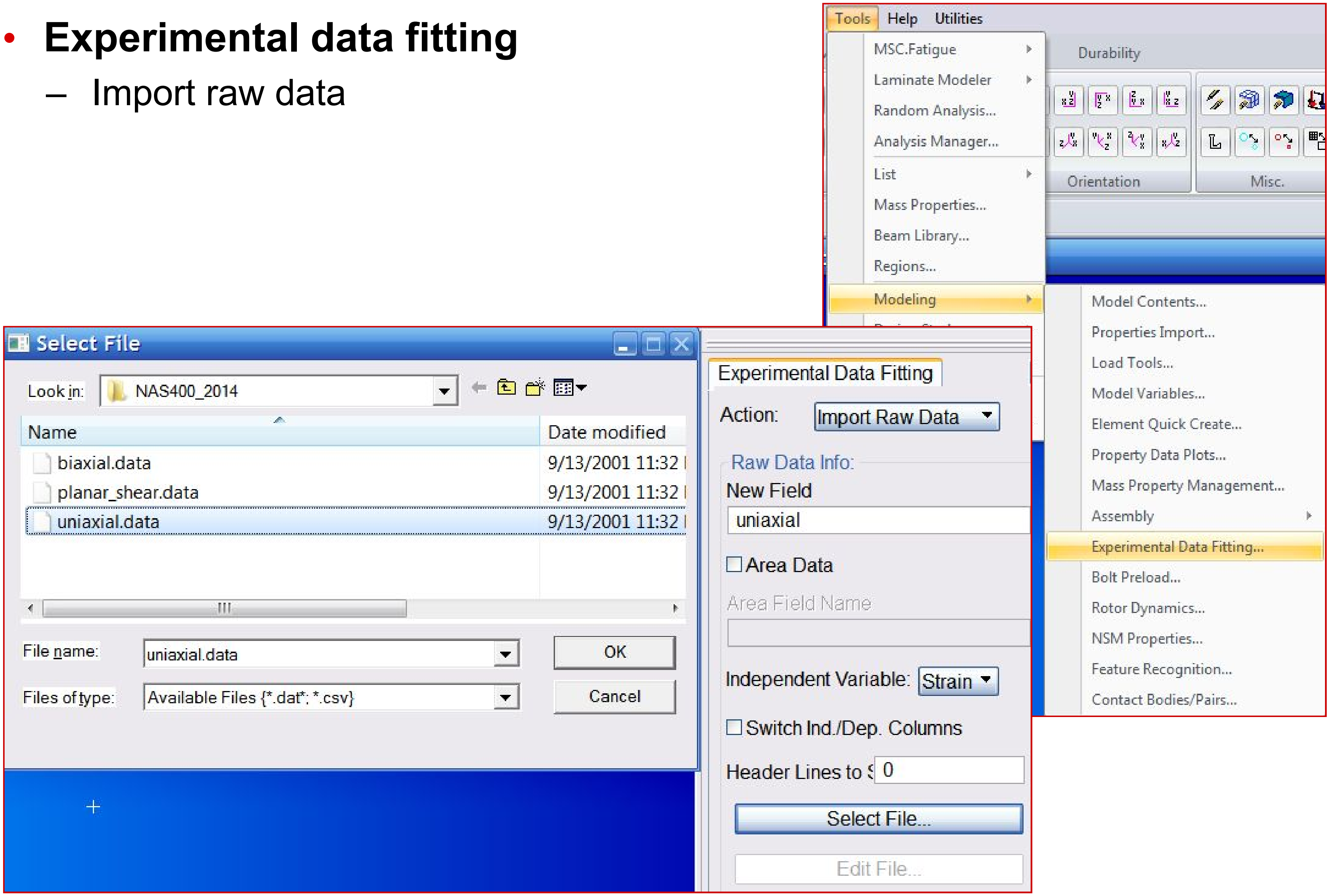
CASE STUDY – RUBBER TUBE

- A rubber tube is compressed by two rigid bodies on the top and bottom
- Experimental data fitting is used to define the rubber material



CASE STUDY – RUBBER TUBE (CONT.)

- **Experimental data fitting**
  - Import raw data



Look in: NAS400_2014

Name	Date modified
biaxial.data	9/13/2001 11:32
planar_shear.data	9/13/2001 11:32
uniaxial.data	9/13/2001 11:32

File name: uniaxial.data

Files of type: Available Files {*.dat*; *.csv}

Action: Import Raw Data

New Field: uniaxial

☐ Area Data

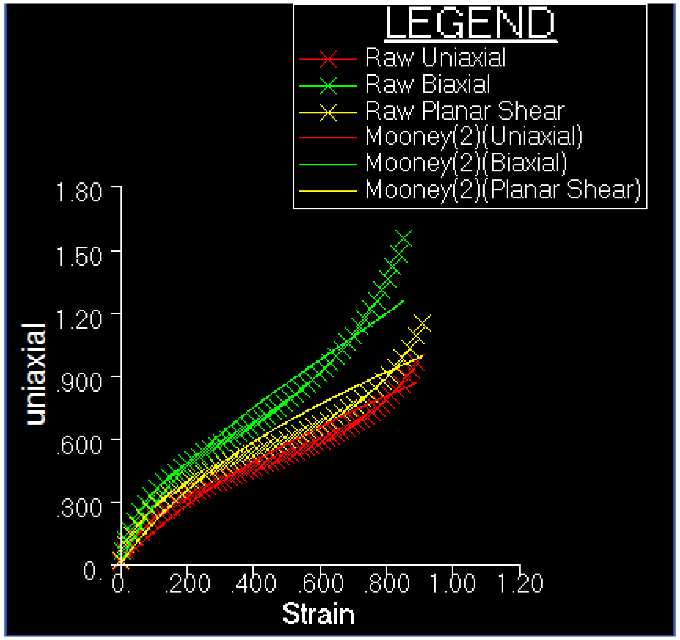
Independent Variable: Strain

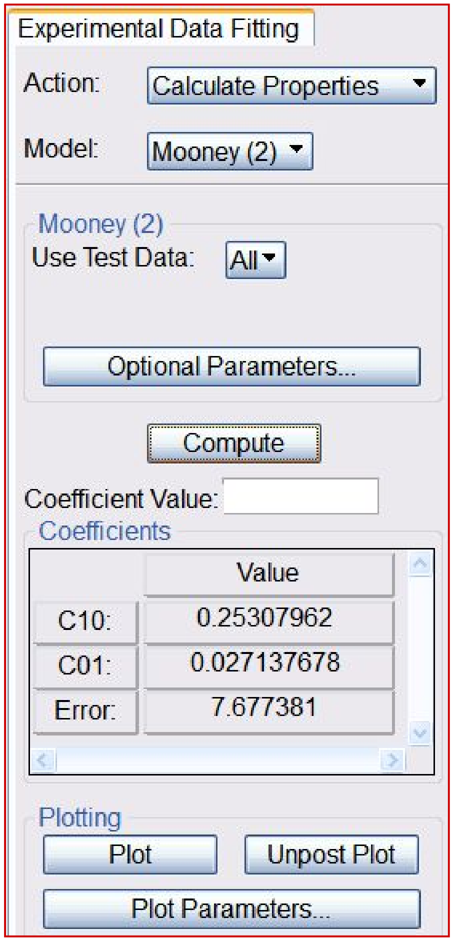
☐ Switch Ind./Dep. Columns

Header Lines to S: 0

CASE STUDY – RUBBER TUBE (CONT.)

- **Experimental data fitting**
  - Select test data
  - Calculate property
  - Plot fitting curves



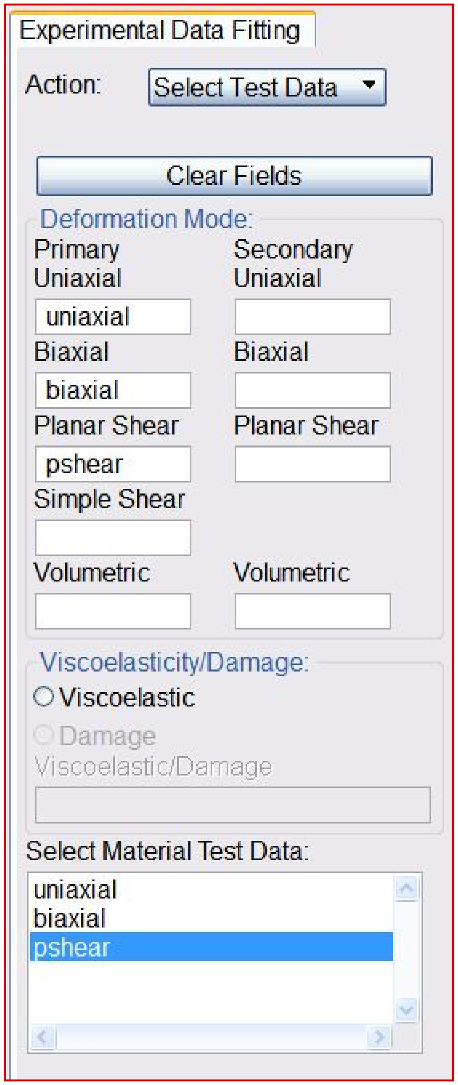


Action: Calculate Properties

Model: Mooney (2)

Use Test Data: All

	Value
C10:	0.25307962
C01:	0.027137678
Error:	7.677381



Action: Select Test Data

Primary Uniaxial: uniaxial

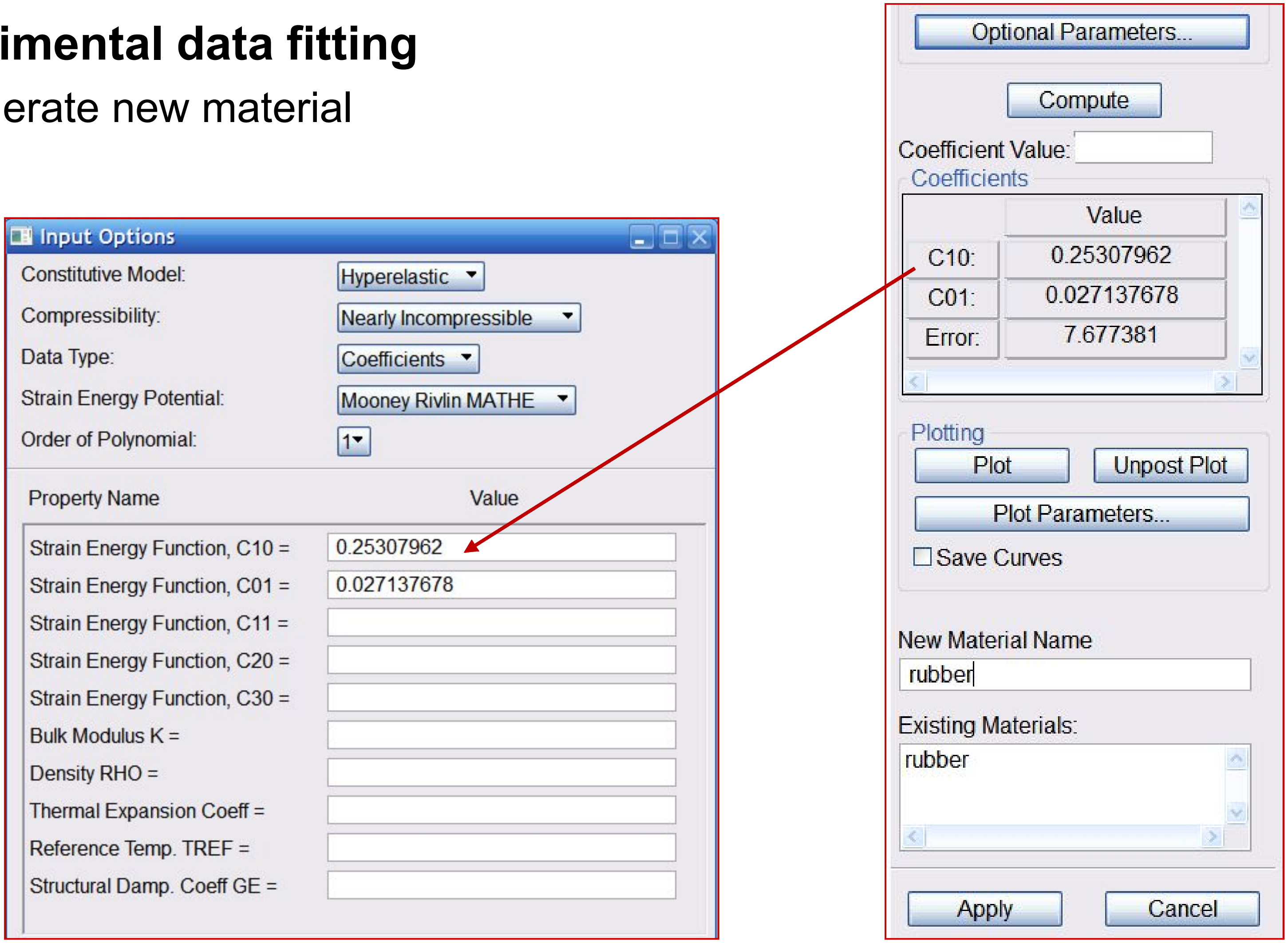
Biaxial: biaxial

Planar Shear: pshear

Select Material Test Data:
uniaxial
biaxial
pshear

CASE STUDY – RUBBER TUBE (CONT.)

- **Experimental data fitting**
  - Generate new material



Input Options

Constitutive Model: Hyperelastic

Compressibility: Nearly Incompressible

Data Type: Coefficients

Strain Energy Potential: Mooney Rivlin MATHE

Order of Polynomial: 1

Property Name	Value
Strain Energy Function, C10 =	0.25307962
Strain Energy Function, C01 =	0.027137678
Strain Energy Function, C11 =	
Strain Energy Function, C20 =	
Strain Energy Function, C30 =	
Bulk Modulus K =	
Density RHO =	
Thermal Expansion Coeff =	
Reference Temp. TREF =	
Structural Damp. Coeff GE =	

Coefficient Value:

	Value
C10:	0.25307962
C01:	0.027137678
Error:	7.677381

New Material Name

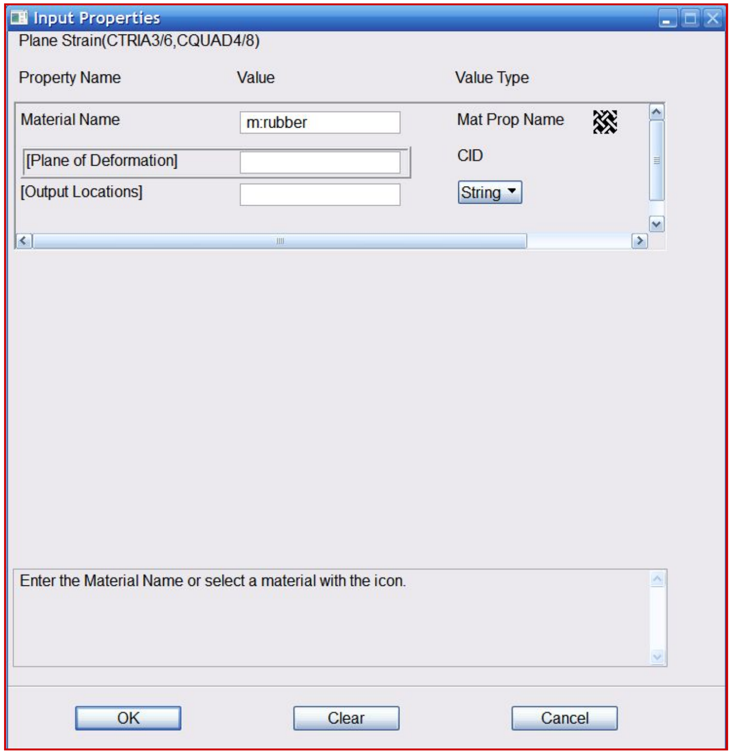
rubber

Existing Materials:

rubber

CASE STUDY – RUBBER TUBE (CONT.)

- **Define element property**
  - 2D solid (plane strain)
  - Hyperelastic formulation

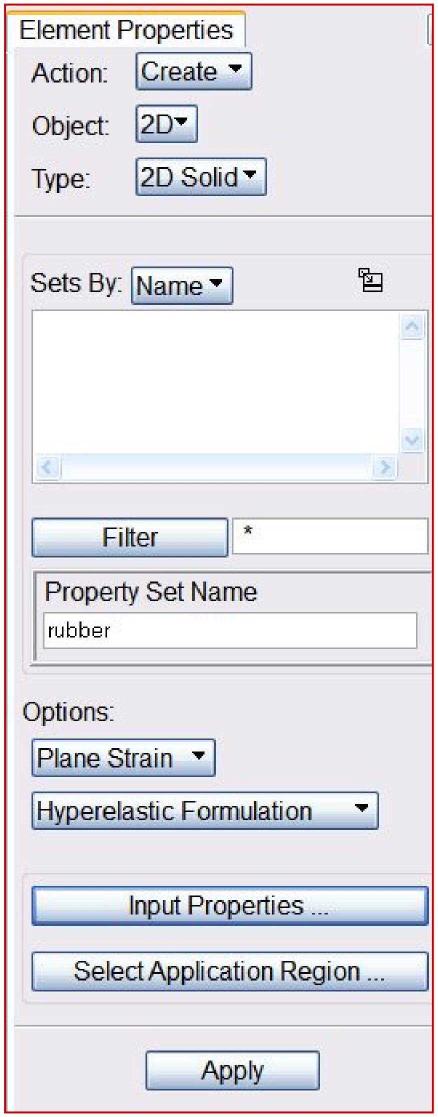


Input Properties

Plane Strain(CTRIA3/6,CQUAD4/8)

Property Name	Value	Value Type
Material Name	m:rubber	Mat Prop Name
[Plane of Deformation]		CID
[Output Locations]		String

Enter the Material Name or select a material with the icon.



Element Properties

Action: Create

Object: 2D

Type: 2D Solid

Sets By: Name

Filter: *

Property Set Name: rubber

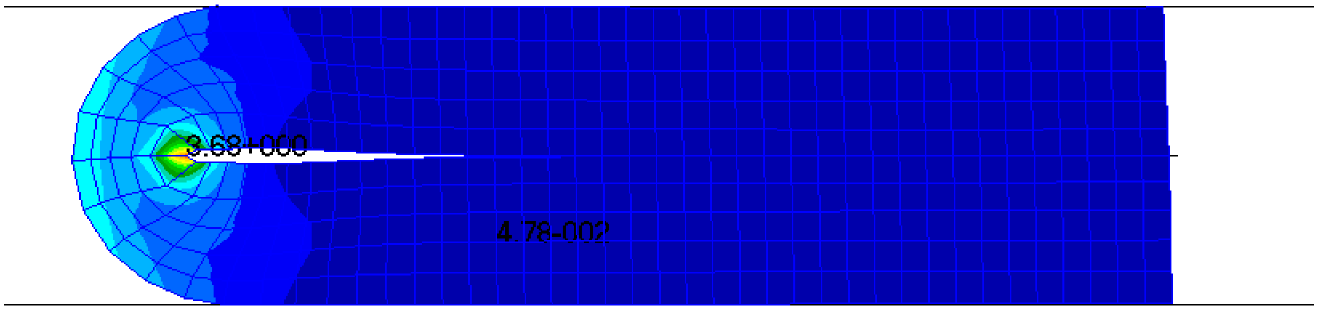
Options:

Plane Strain

Hyperelastic Formulation

CASE STUDY – RUBBER TUBE (CONT.)

- Setup implicit nonlinear (SOL 400) job
- Run the job and post-process results

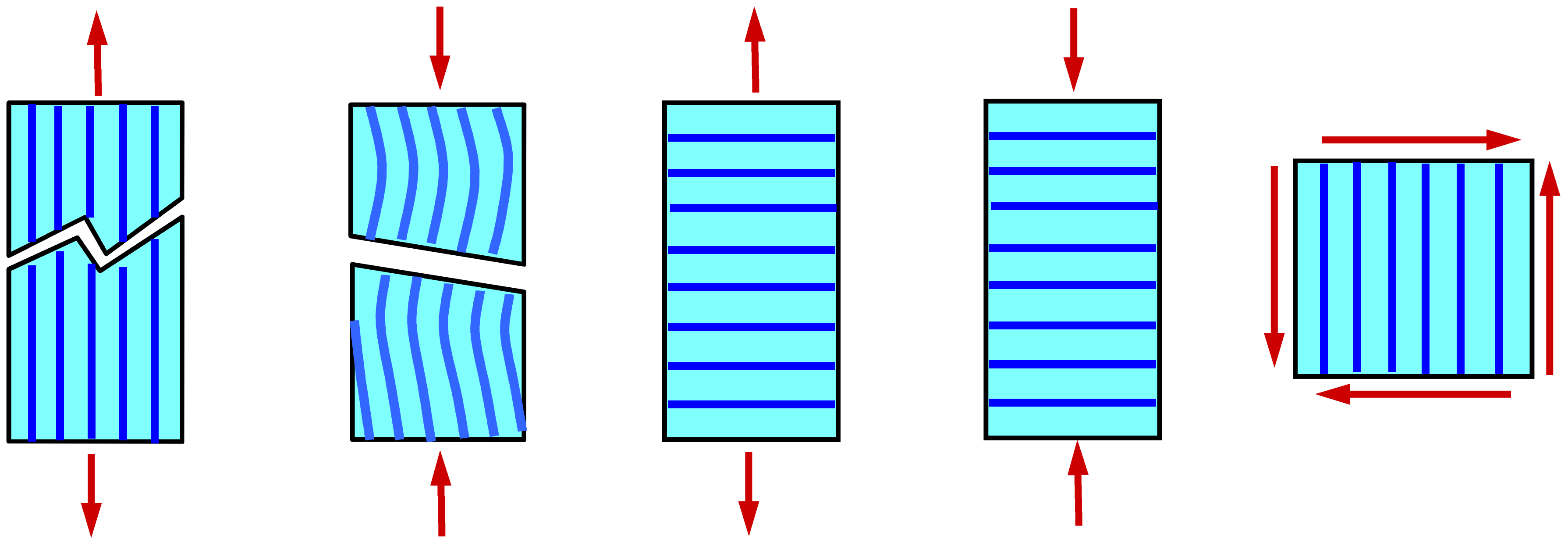


COMPOSITE FAILURE

COMPOSITE ANALYSIS

- **Traditional composite analysis**
  - Linear analysis
  - Compute ply failure index and/or strength ratio
  - Compute bonding failure index and strength ratio
- **Incentives for going beyond linear analysis**
  - Evaluate the ultimate capacity of a composite structure by progressively failing the plies
  - Simulate delamination/debonding
  - Simulate crack propagation

FAILURE MECHANISMS



Longitudinal Tension	***Longitudinal Compression***	***Transverse Tension***	***Transverse Compression***	***In-Plane Shear***
Fibre Failure	***Fibre Failure***	***Matrix Failure***	***Matrix Failure***	***Fibre-Matrix Interface Failure***

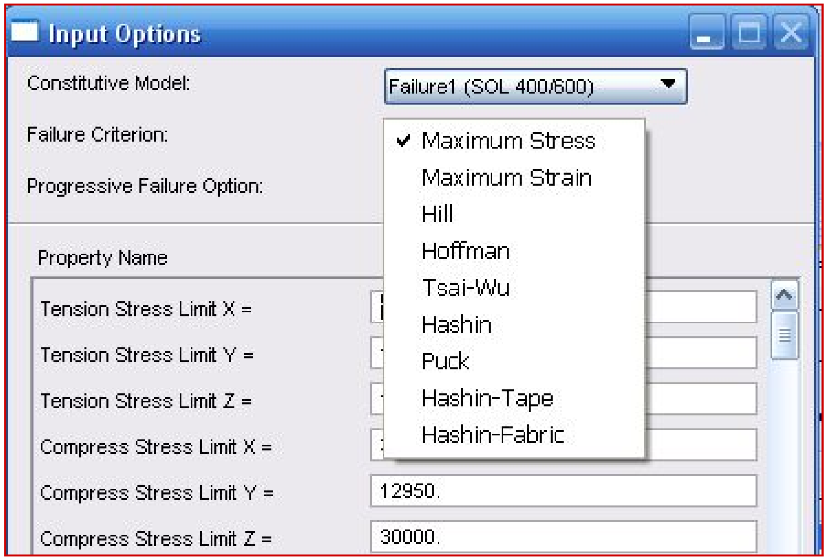
INPUT CARD FOR COMPOSITE FAILURE

- **Failure model properties are entered in MATF card**
- **Up to three different failure criteria can be specified**
- **Compute failure indices only if ITYPE=0 (default)**

MATF	MID	ITYPE							
	"CRI"	Criterion	V_1^1	V_2^1	V_3^1	V_4^1	V_5^1	V_6^1	1st
	V_7^1	V_8^1	V_9^1	Find^1	V_{10}^1	V_{11}^1	V_{12}^1	W_1^1	
	W_2^1	W_3^1	W_4^1	W_5^1	W_6^1	W_7^1	W_8^1	W_9^1	
	"PF"	A1	A2	A3	A4	A5			
	"CRI"	Criterion	V_1^2	V_2^2	V_3^2	V_4^2	V_5^2	V_6^2	2nd
	V_7^2	V_8^2	V_9^2	Find^1	V_{10}^2	V_{11}^2	V_{12}^2	W_1^2	
	W_2^2	W_3^2	W_4^2	W_5^2	W_6^2	W_7^2	W_8^2	W_9^2	
	"CRI"	Criterion	V_1^3	V_2^3	V_3^3	V_4^3	V_5^3	V_6^3	3rd
	V_7^3	V_8^3	V_9^3	Find^1	V_{10}^3	V_{11}^3	V_{12}^3	W_1^3	
	W_2^3	W_3^3	W_4^3	W_5^3	W_6^3	W_7^3	W_8^3	W_9^3	

FAILURE CRITERIA

- **The available failure criteria are:**
  - 1: Maximum stress criterion
  - 2: Maximum strain criterion
  - 3: Hill failure criterion
  - 4: Hoffman failure criterion
  - 5: Tsai-Wu failure criterion
  - 7: Hashin failure criteria
  - 8: Puck failure criteria
  - 10: Hashin-Tape
  - 11: Hashin-Fabric



Constitutive Model: Failure1 (SOL 400/600)

Compress Stress Limit Y = 12950.

Compress Stress Limit Z = 30000.

FAILURE CRITERIA (CONT.)

1. Maximum stress

stress$_i$ / allowed stress$_i$ > F$_{ind}$

Failure Index F$_{ind}$
(default = 1.0)

2. Maximum strain

strain$_i$ / allowed strain$_i$ > F$_{ind}$

3. Hill

$$\frac{\sigma_x^2}{X^2}+\frac{\sigma_y^2}{Y^2}+\frac{\sigma_z^2}{Z^2}-\left(\frac{1}{X^2}+\frac{1}{Y^2}-\frac{1}{Z^2}\right)\sigma_x\sigma_y-\left(\frac{1}{Y^2}+\frac{1}{Z^2}-\frac{1}{X^2}\right)\sigma_y\sigma_z\ -\left(\frac{1}{Z^2}+\frac{1}{X^2}-\frac{1}{Y^2}\right)\sigma_z\sigma_x+\frac{\tau_{xy}^2}{S_{xy}^2}+\frac{\tau_{yz}^2}{S_{yz}^2}+\frac{\tau_{zx}^2}{S_{zx}^2}>F_{ind}$$

4. Hoffman

$$C_x(\sigma_x-\sigma_y)^2+C_y(\sigma_y-\sigma_z)^2+C_z(\sigma_z-\sigma_x)^2+\left(\frac{1}{X_t}-\frac{1}{X_c}\right)\sigma_x+\left(\frac{1}{Y_t}-\frac{1}{Y_c}\right)\sigma_y+\left(\frac{1}{Z_t}-\frac{1}{Z_c}\right)\sigma_z+\frac{\tau_{xy}^2}{S_{xy}^2}+\frac{\tau_{yz}^2}{S_{yz}^2}+\frac{\tau_{zx}^2}{S_{zx}^2}>F_{ind}$$

5. Tsai-Wu

$$\left(\frac{1}{X_t}-\frac{1}{X_c}\right)\sigma_x+\left(\frac{1}{Y_t}-\frac{1}{Y_c}\right)\sigma_y+\left(\frac{1}{Z_t}-\frac{1}{Z_c}\right)\sigma_z+\frac{\sigma_x^2}{X_tX_c}+\frac{\sigma_y^2}{Y_tY_c}+\frac{\sigma_z^2}{Z_tZ_c}+\frac{\tau_{xy}^2}{S_{xy}^2}+\frac{\tau_{yz}^2}{S_{yz}^2}+\frac{\tau_{zx}^2}{S_{zx}^2}+2F_{xy}\sigma_x\sigma_y+2F_{yz}\sigma_y\sigma_z+2F_{zx}\sigma_x\sigma_z>F_{ind}$$

FAILURE CRITERIA (CONT.)

7. Hashin

Fiber Modes

Tension fiber mode,

① $$\left(\frac{\sigma_{11}}{X_t}\right)^2 + \frac{1}{S^2}(\sigma_{12}^2 + \sigma_{13}^2) = 1 \text{ or } \sigma_{11} = X_T$$

Compressive fiber mode, $\sigma_1 < 0$

② $$\frac{|\sigma_{11}|}{X_c} = X_c$$

Matrix Modes

Tensile matrix mode, $\sigma_2 + \sigma_3 > 0$

③ $$\left[\frac{1}{Y_t}(\sigma_2 + \sigma_3)^2 + \frac{1}{S_{23}^2}(\sigma_{23}^2 - \sigma_2\sigma_3) + \frac{1}{S_{12}}(\sigma_{12}^2 + \sigma_{13}^2)\right]$$

Compressive matrix mode, $\sigma_2 + \sigma_3 < 0$

④ $$\left[\frac{1}{Y_c}\left(\left(\frac{Y_c}{2S_{23}}\right)^2 - 1\right)(\sigma_2 + \sigma_3) + \frac{1}{4S_{23}^2}(\sigma_2 + \sigma_3)^2 + \frac{1}{S_{23}^2}(\sigma_{23}^2 - \sigma_2\sigma_3 + \frac{1}{S_{12}^2}(\sigma_{12}^2 + \sigma_{13}^2)\right]$$

FAILURE CRITERIA (CONT.)

8. Puck

Two modes of fiber failure

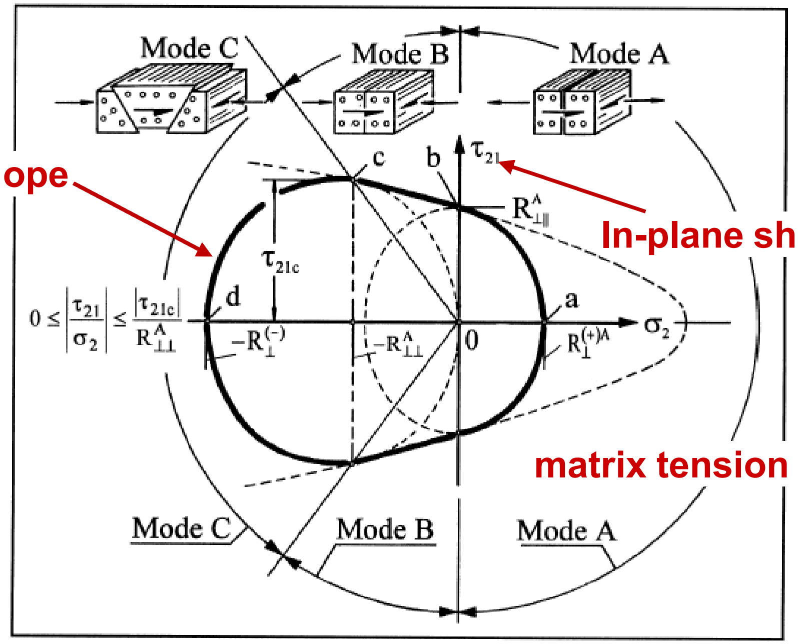
Tensile fiber mode, $\sigma_1 > 0$

① $\frac{\sigma_1}{X_t}$

Compressive fiber mode, $\sigma_1 < 0$

② $\frac{|\sigma_1|}{X_c}$

Three modes of matrix failure†



Failure envelope

In-plane shear

matrix tension

Mode C is the most dangerous due to “wedge effect” which can lead to catastrophic failure.

---

†Puck, Shurmann: Failure Analysis of FRP Laminates by means of Phenomenological Models,Composites Science and Technology, 58 (1998), p. 1052.

FAILURE CRITERIA (CONT.)

8. Puck (cont.)

$$\text{Mode A, } \sigma_2 > 0\,,\ \theta_{fp} \ = \ 0$$

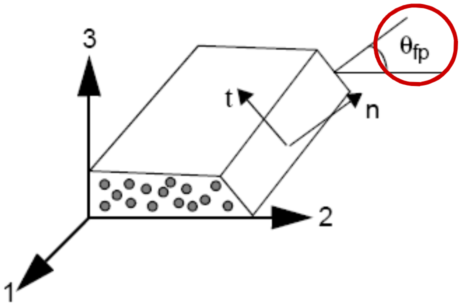
③ $$\left[\sqrt{\left(\frac{\sigma_{12}}{S_{12}}\right)^2 + \left(1 - p_{12t}\frac{Y_t}{S_{12}}\right)^2\left(\frac{\sigma_2}{Y_t}\right)^2} + p_{12t}\frac{\sigma_2}{S_{12}}\right]$$

$$\text{Mode B, } \sigma_2 < 0 \text{ and } 0 \le \left|\frac{\sigma_2}{\sigma_{12}}\right| \le \frac{R^A}{\sigma_{21c}}\,,\ \theta_{fp} \ = \ 0$$

④ $$\left[\frac{1}{S_{12}}\left(\sqrt{\sigma_{12}^2 + (p_{12c}\sigma_2)^2} + p_{12c}\sigma_2\right)\right]$$

$$\text{Mode C, } \sigma_2 < 0 \text{ and } 0 \le \left|\frac{\sigma_{12}}{\sigma_2}\right| \le \frac{\sigma_{21c}}{R^A}\,.$$

⑤ $$\left[\left(\left(\frac{\sigma_{12}}{2(1 + p_{23c}S_{12})}\right)^2 + \left(\frac{\sigma_2}{Y_c}\right)^2\right)\frac{Y_c}{|\sigma_2|}\right]$$



- Failure Angle θ_{fp}
- Analytical calculation for plane stress, numerical calculation otherwise
- The Failure Index is calculated as a function of this angle (output as 6th mode)

Additional inputs are p12c, p12t, p23c, p23t to describe the failure envelope, typically p12c = p12t = 0.35; p23c = p23t = 0.27

FAILURE CRITERIA (CONT.)

10. Hashin-Tape (1: fiber, 2: matrix, 3: thickness direction)

Fiber Modes

Tensile fiber mode, $\sigma_1 > 0$

① $$\left[\left(\frac{\sigma_1}{X_t}\right)^2 + \left(\frac{\sigma_{12}}{S_{12}}\right)^2 + \left(\frac{\sigma_{13}}{S_{13}}\right)^2\right]$$

Compressive fiber mode, $\sigma_1 < 0$

② $$\left[\left(\frac{\sigma_1}{X_c}\right)^2 + \left(\frac{\sigma_{12}}{S_{12}}\right)^2 + \left(\frac{\sigma_{13}}{S_{13}}\right)^2\right]$$

Matrix Modes

Tensile matrix mode, $\sigma_2 + \sigma_3 > 0$

③ $$\left[\left(\frac{\sigma_2 + \sigma_3}{Y_t}\right)^2 - \frac{\sigma_{22}\sigma_{33}}{S_{23}^2} + \left(\frac{\sigma_{12}}{S_{12}}\right)^2 + \left(\frac{\sigma_{13}}{S_{13}}\right)^2 + \left(\frac{\sigma_{23}}{S_{23}}\right)^2\right]$$

Compressive matrix mode, $\sigma_2 + \sigma_3 < 0$

④ $$\left[\left(\left(\frac{Y_c}{2S_{23}}\right)^2 - 1\right)\left(\frac{\sigma_2 + \sigma_3}{Y_c}\right) + \left(\frac{\sigma_2 + \sigma_3}{2S_{23}}\right)^2 - \frac{\sigma_2\sigma_3}{S_{23}^2} + \left(\frac{\sigma_{12}}{S_{12}}\right)^2 + \left(\frac{\sigma_{13}}{S_{13}}\right)^2 + \left(\frac{\sigma_{23}}{S_{23}}\right)^2\right]$$

FAILURE CRITERIA (CONT.)

11. <u>Hashin-Fabric</u> (1 and 2: fiber directions, 3: thickness direction)

Fiber Modes

Tensile fiber 1 mode, $\sigma_1 > 0$

① $$\left[\left(\frac{\sigma_1}{X_t}\right)^2 + \left(\frac{\sigma_{12}}{S_{12}}\right)^2 + \left(\frac{\sigma_{13}}{S_{13}}\right)^2\right]$$

Compressive fiber 1 mode, $\sigma_1 < 0$

② $$\left[\left(\frac{\sigma_1}{X_c}\right)^2 + \left(\frac{\sigma_{12}}{S_{12}}\right)^2 + \left(\frac{\sigma_{13}}{S_{13}}\right)^2\right]$$

Tensile fiber 2 mode, $\sigma_2 > 0$

③ $$\left[\left(\frac{\sigma_2}{Y_t}\right)^2 + \left(\frac{\sigma_{12}}{S_{12}}\right)^2 + \left(\frac{\sigma_{13}}{S_{13}}\right)^2\right]$$

Compressive fiber 1 mode, $\sigma_2 < 0$

④ $$\left[\left(\frac{\sigma_2}{Y_c}\right)^2 + \left(\frac{\sigma_{12}}{S_{12}}\right)^2 + \left(\frac{\sigma_{13}}{S_{13}}\right)^2\right]$$

Matrix Modes

Tensile matrix mode, $\sigma_3 > 0$

⑤ $$\left[\left(\frac{\sigma_3}{Z_t}\right)^2 + \left(\frac{\sigma_{12}}{S_{12}}\right)^2 + \left(\frac{\sigma_{13}}{S_{13}}\right)^2 + \left(\frac{\sigma_{23}}{S_{23}}\right)^2\right]$$

Compressive matrix mode, $\sigma_3 < 0$

⑥ $$\left[\left(\frac{\sigma_3}{Z_c}\right)^2 + \left(\frac{\sigma_{12}}{S_{12}}\right)^2 + \left(\frac{\sigma_{13}}{S_{13}}\right)^2 + \left(\frac{\sigma_{23}}{S_{23}}\right)^2\right]$$

PROGRESSIVE FAILURE

- **For FI > 1 the material stiffness will be degraded**
  - controlled via ITYPE and PF part on MATF
    - ITYPE=2 – gradually
      - so that the largest FI becomes 1 within an increment
      - moduli are reduced by $1 - e^{1-FI}$ based on rules for the criteria
      - the stiffness is not reduced to more than A1. Default = 0.01.
    - ITYPE=3 – immediately
      - fraction of initial stiffness upon failure is A1 based on rules for the criteria. Default = 0.01.
  - For maximum stress and strain
    - Each modulus is reduced separately for each respective failure mode according to ITYPE and A1
    - For example, if failure in the 2-direction is found, only E_2 is being reduced

PROGRESSIVE FAILURE (CONT.)

- **Stiffness Degradation (cont.)**
  - Hill, Hoffman, Tsai-Wu
    - since only one FI is available all moduli are reduced with the same factor according to ITYPE and A1
  - Puck and Hashin, additional control is available
    - A2 – Matrix compression factor. E_2 is reduced by:

      $$(1-\mathrm{A2})(1-\mathrm{e}^{1-\mathrm{F_{mc}}})$$

      with F_{mc} matrix compression failure. Default = 0.0.
    - A3 – Shear stiffness factor. G_{12} is reduced by:

      $$(1-\mathrm{A3})\left(1-\mathrm{e}^{1-\mathrm{F_m}}\right) \text{ or } (1-\mathrm{A2})(1-\mathrm{A3})(1-\mathrm{e}^{1-\mathrm{F_{mc}}})$$

      for matrix or matrix compression failure. Default=0.0.
    - A4 – E3 reduction. With F_f and F_m (fiber & matrix failure):

      $$(1-\mathrm{A4})\left(1-\mathrm{e}^{1-\mathrm{F_f}}\right)+\mathrm{A4}(1-\mathrm{e}^{1-\mathrm{F_m}})$$

PROGRESSIVE FAILURE (CONT.)

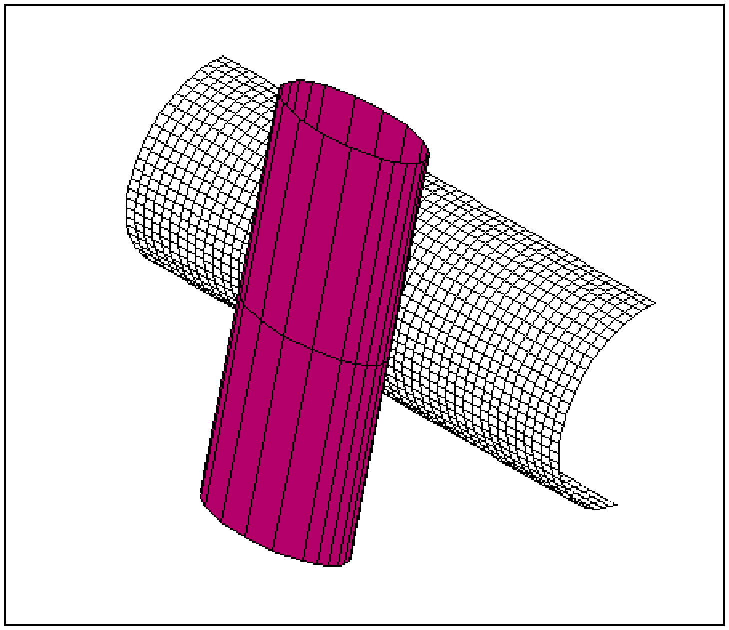
- **Stiffness Degradation (cont.)**
  - Puck and Hashin (cont.)
    - A5 – Shear reduction from fiber failure. With F_f and F_m (fiber & matrix failure) G12 will be reduced by:

$$(1 - A5)\left(1 - e^{1-F_f}\right) + A5\left(1 - e^{1-F_m}\right)$$

  - Hashin Fabric
    - A2 – A5 are not used
    - The E moduli are reduced separately according to the failure modes
    - The shear moduli are reduced using the worst of E_i

PROGRESSIVE FAILURE EXAMPLE

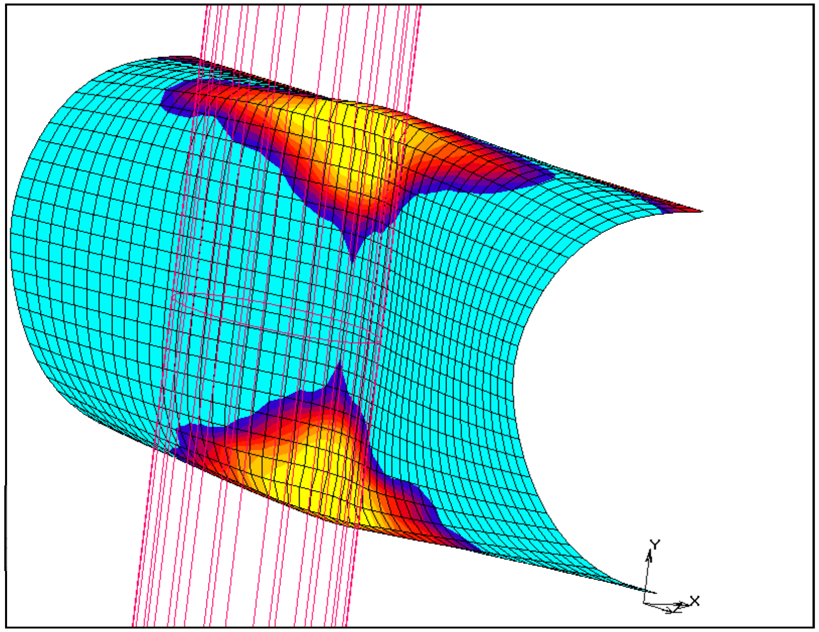
- **Rigid elliptical cylinder hitting composite shell**
  - Moving rigid body



Composite with 5 layers

[0 45 90 -45 0]

Yellow means outer ply fully damaged



GASKET MATERIAL

GASKET MATERIAL – MATG

- **Format:**

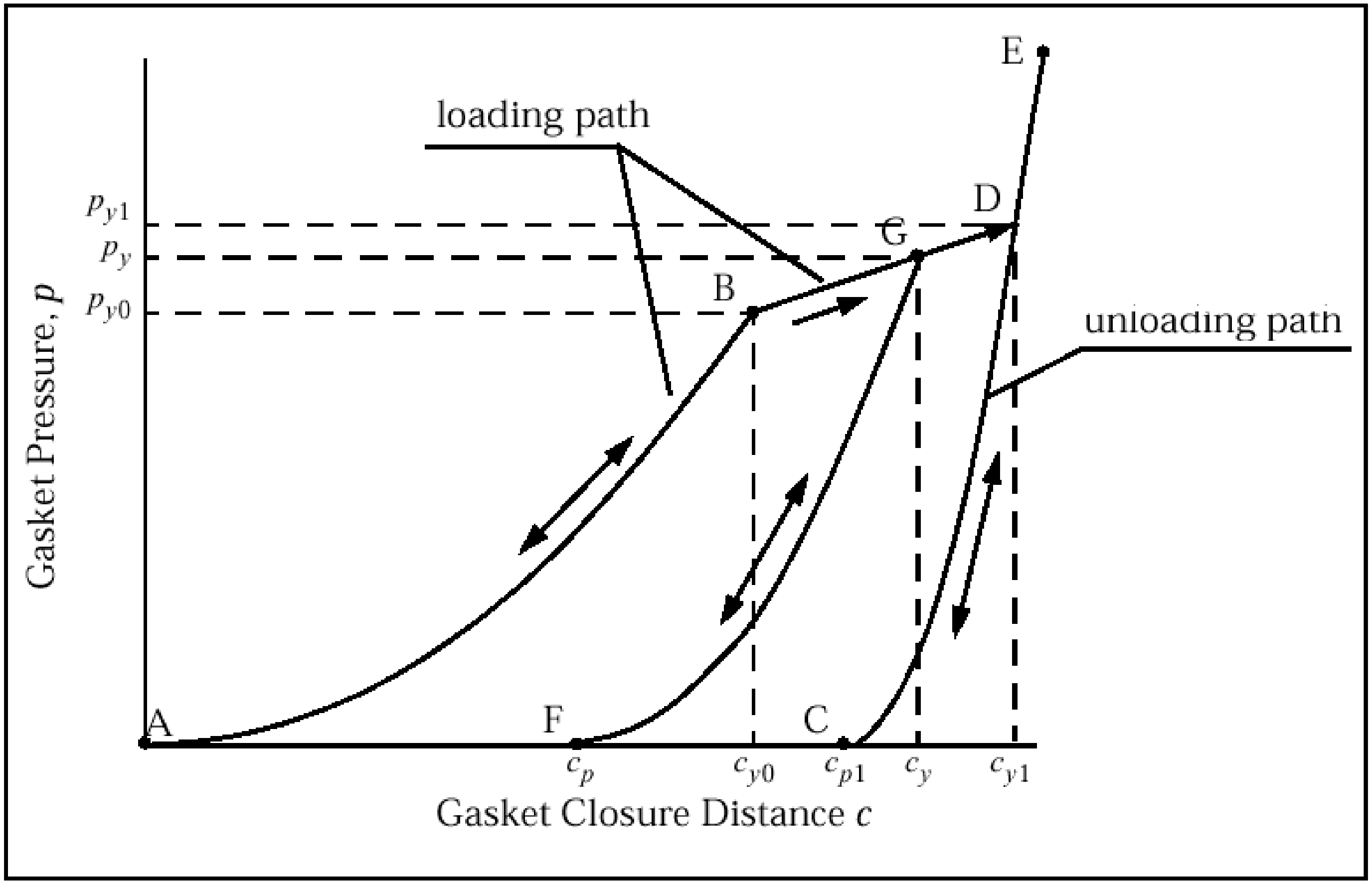
1	2	3	4	5	6	7	8	9	10
MATG	MID	IDMEM	BEHAV	TABLD	TABLU1	TABLU2	TABLU3	TABLU4	
	TABLU5	TABLU6	TABLU7	TABLU8	TABLU9	TABLU10	YPRS	EPL	
	GPL	GAP	TABYPRS	TABEPL	TABGPL	TABGAP	N/A	N/A	

- **Example:**

MATG	100	10	0	1001	1002	1003			
							100.	2500.	
	950.	0.0							

GASKET MATERIAL (CONT.)

- **MATG – Pressure vs. Distance Curves**



GASKET MATERIAL (CONT.)

- **MATG – Input Example**

transverse shear modulus

```
BEGIN BULK
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
MATG    1       2       0       1       2
+                                                       52.     72.
+       35.     .090909
TABLES1 1
+       0.      0.      .027    2.08    .054    8.32    .081    18.72
+       .108    33.28   .135    52.     .175    56.     ENDT
TABLES1 2
+       .1      0.      .1225   5.04    .1375   14.     .1525   27.44
+       .16     35.84   .1675   45.36   .175    56.     ENDT
```

yield stress

tensile modulus

initial opening

loading curve

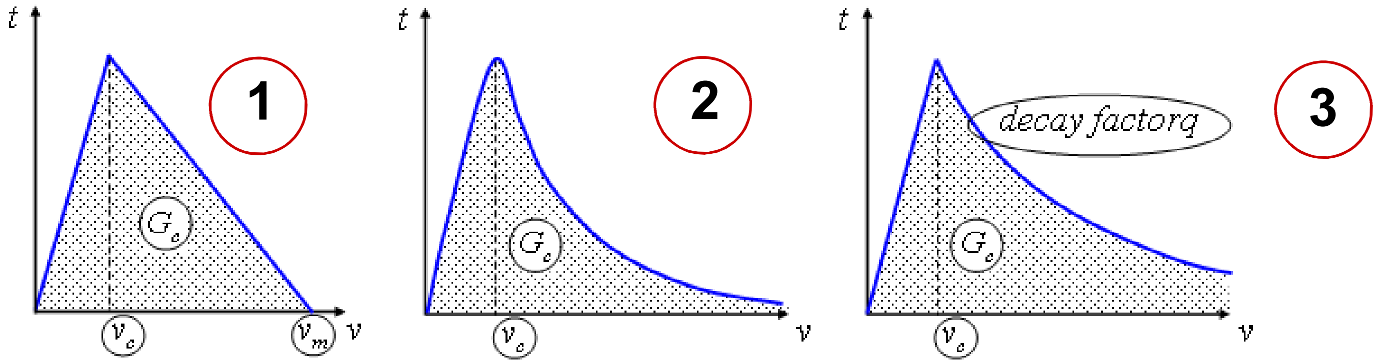
unloading curve

CRACK MATERIALS

COHESIVE ZONE MATERIAL – MCOHE

1	2	3	4	5	6	7	8	9	10
MCOHE	MID	MODEL		TID					
	COHE	CRTOD	MAXOD	SNSR	EXP	VED	RRRD	SFC	
	SNER								

Model

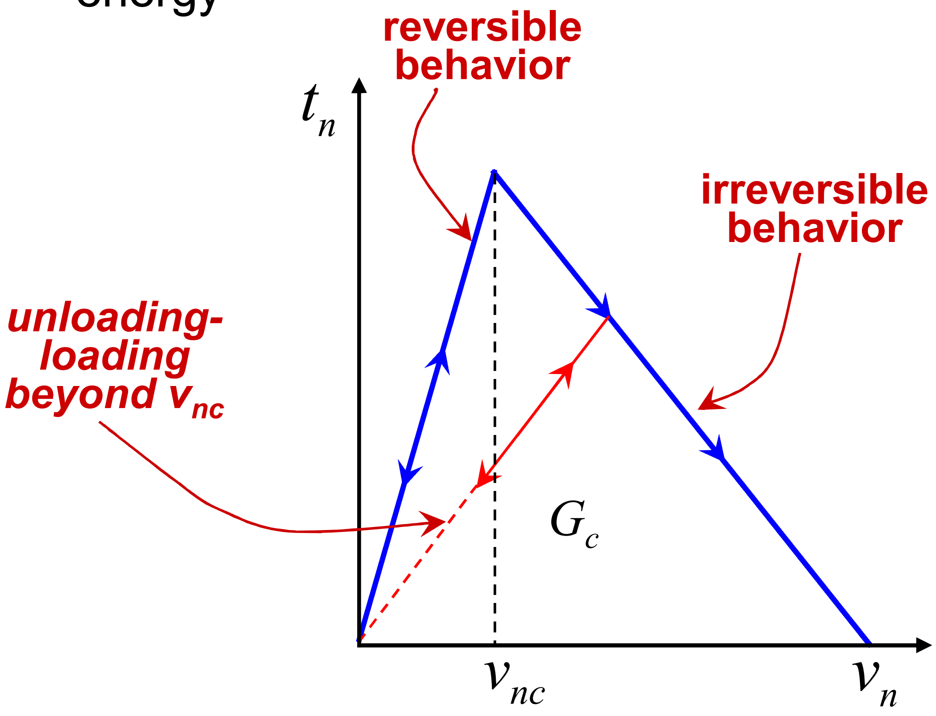


- **Referring Property Entry**

1	2	3	4	5	6	7	8	9	10
PCOHE	PID	MID	INT	T	OUTPUT	SECANT			

COHESIVE ZONE MATERIAL (CONT.)

- **Material behavior:**
  - Initially reversible behavior
  - Irreversible behavior as soon as a critical relative displacement has been reached
  - Area below the traction-relative displacement curve is called the cohesive energy



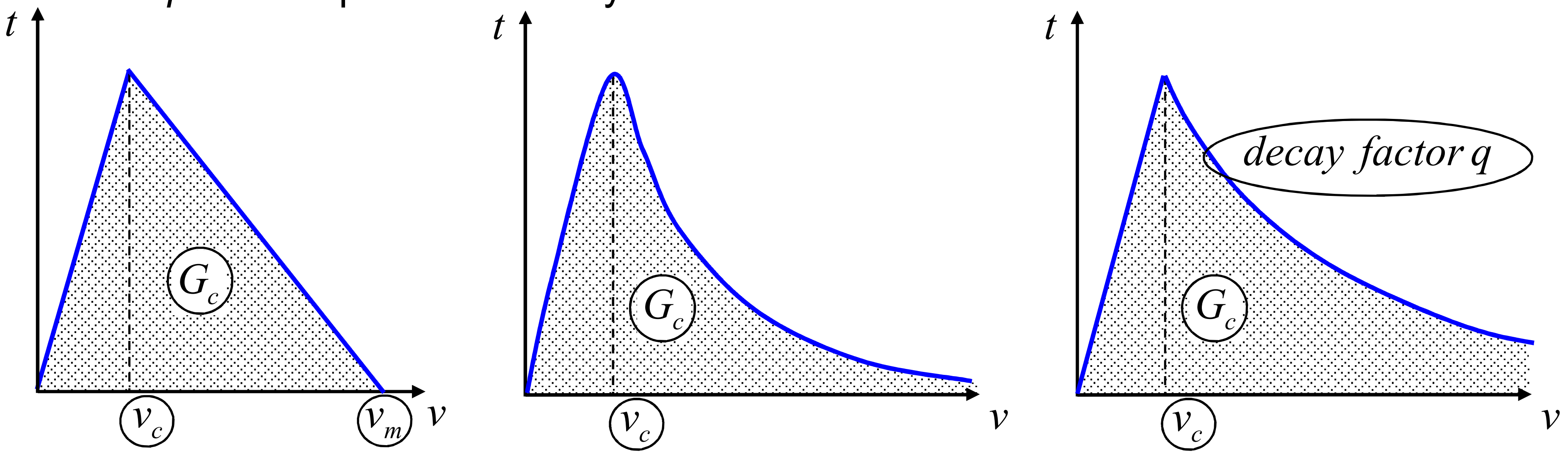
G_c units = Energy/Area

COHESIVE ZONE MATERIAL (CONT.)

- **Available material models**
  - Bilinear (3 input quantities: G_c V_c V_m)
  - Exponential (2 input quantities: G_c V_c) →
  - Linear-Exponential (3 input quantities: G_c V_c q)

$$\boxed{t = G_c \frac{v}{v_c^2} e^{-v/v_c}}$$

- **Input quantities**
  - G_c is the cohesive energy
  - V_c is the critical effective opening displacement
  - V_m is the maximum effective opening displacement
  - q is the exponential decay factor



COHESIVE ZONE MATERIAL (CONT.)

- **Available testing data could be the cohesive energy and the maximum traction, instead of the critical or maximum opening displacement**
- **The relationships to convert such data:**

$$\text{- exponential model: } v_c = \frac{G_c}{et_c} ;$$

$$\text{- linear model: } v_m = \frac{2G_c}{t_c} ;$$

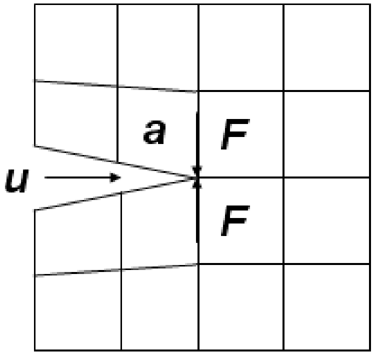
$$\text{- linear-exponential model: } v_c = \frac{2qG_c}{t_c(q+2)}$$

VIRTUAL CRACK CLOSURE TECHNIQUE

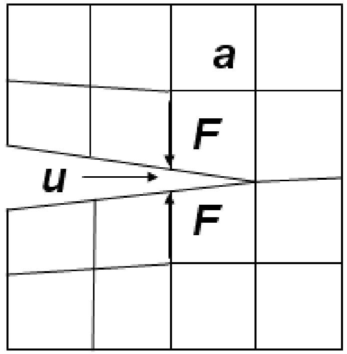
1	2	3	4	5	6	7	8	9	10
VCCT	ID	IDCR	ITYPE	IGROW	INCM	METHOD	TIME	IACT	
	CGI	GC	GTH	C	M	GMIN	GC-II	GC-III	
	TABCGI	TABGC	TABGTH	TABC	TABM	TABGMIN	TABGC-II	TABGC-III	
	G1	G2	G3	G4	G5	etc.			

- **Simple and Robust**
  - Force needed to keep the crack closed, F
  - Crack opening displacement, u
  - Area around the crack tip, a
  - The crack is initially closed
  - Energy release rate:

$$G = \frac{Fu}{2a}$$



①



②

VCCT – BASIC

- **ITYPE = 0 (no crack propagation)**
- **User only needs to define the crack front**
- **Basic VCCT definition gives you**
  - Calculation of the energy release rate G and the respective modes G_I, G_{II} and G_{III}
  - Estimated crack growth direction
  - Results are saved for post-processing and printed to the .f06 file
  - Results output format in the .f06 file:

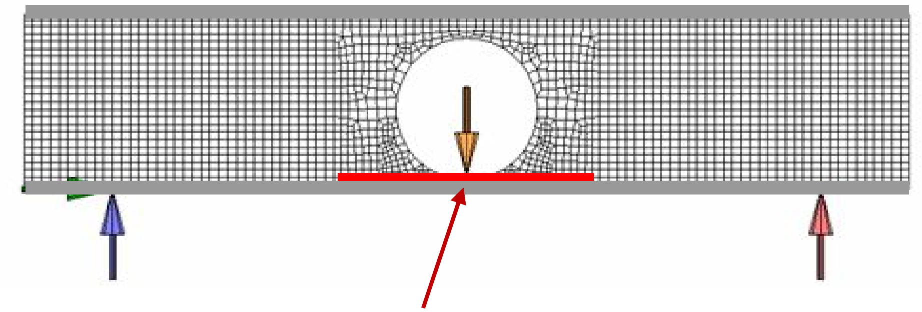
```
                               V C C T     C R A C K     R E S U L T S


        CRACK TIP    ------------- ENERGY RELEASE RATE ------------    ESTIMATED CRACK GROWTH DIRECTION
CRACK ID  GRID ID       TOTAL      MODE I       MODE II     MODE III          X           Y           Z
```

VCCT – CRACK PROPAGATION

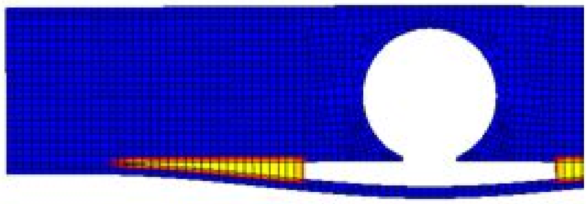
- **Crack growth mode**
  - Direct Growth (ITYPE=2)
- **Crack growth method**
  - By releasing glued contact
- **Crack growth criteria**
  - Total energy release rate $G>G_C$ (enter G_C only)
  - Individual modes $G_I>G_{c-I}$ or $G_{II}>G_{c-II}$ or $G_{III}>G_{c-III}$ (enter $G_c=G_{c-I}$, G_{c-II} and G_{c-III})
- **Crack growth direction method**
  - Maximum hoop stress direction

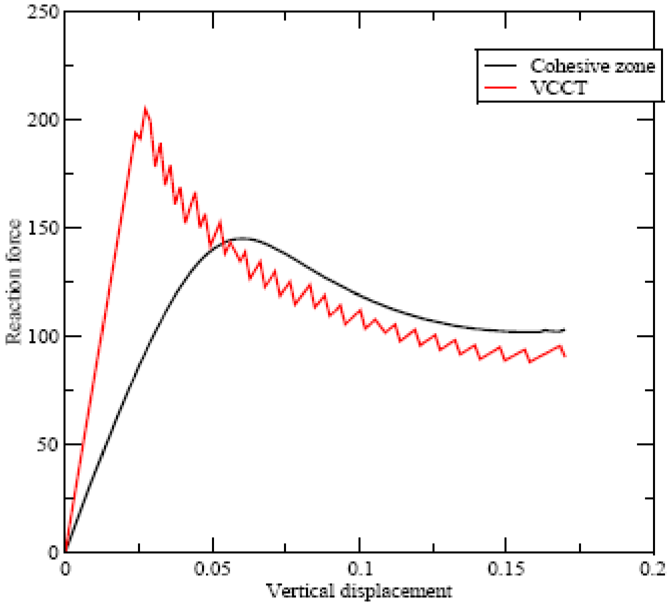
COHESIVE ZONE AND VCCT EXAMPLE

- **Core with facesheets**



initial defect



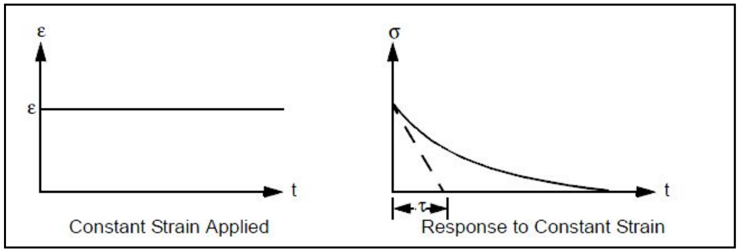


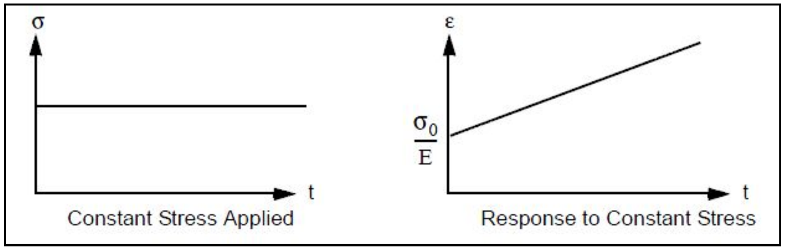
- **NASA Push Test**
  - UG #20

CREEP MATERIAL

BASICS OF CREEP MATERIAL

- **Creep is a time-dependent inelastic behavior**
  - Typically exhibited as stress relaxation of a component loaded with a prescribed constant displacement or changing deflection of a component loaded with a constant load (and, thereby, stress):

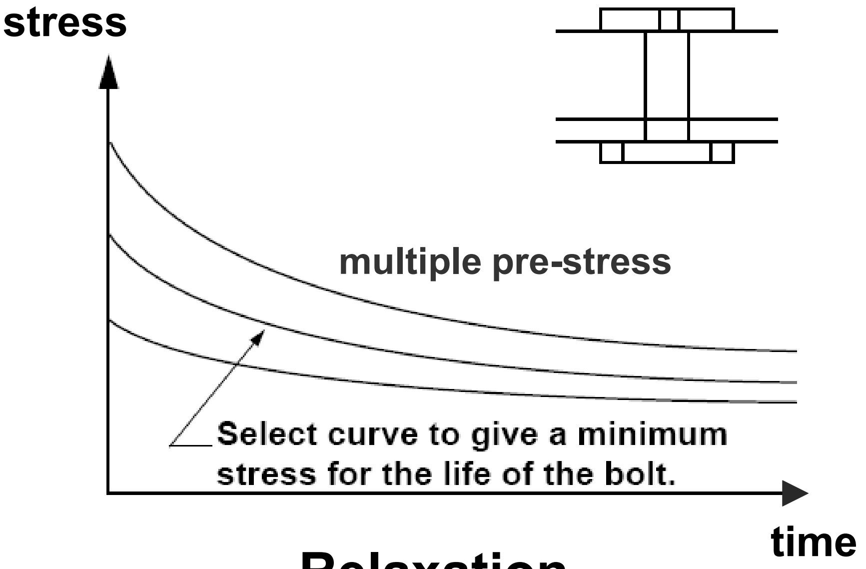




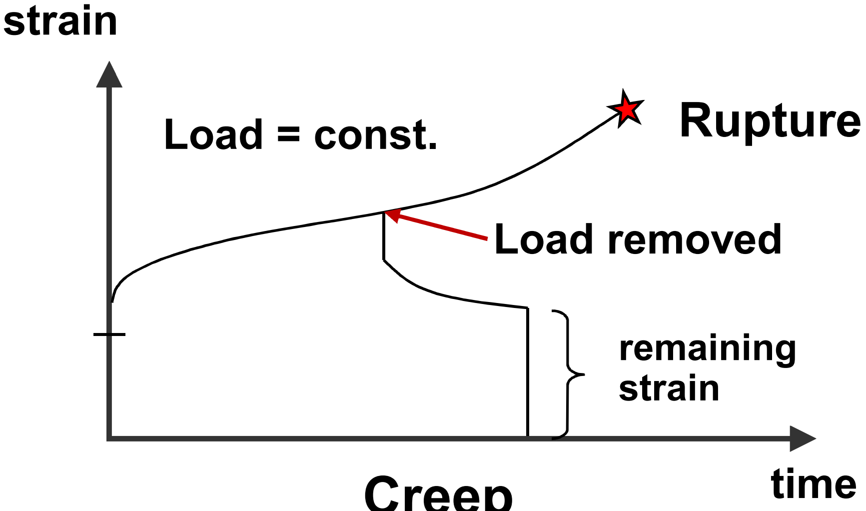
  - Can occur at stresses well below the yield stress
  - Typically more common at high temperatures
  - The minimum temperature required for creep deformation to occur is 30-40% of the melting point for metals
  - Creep can occur at relatively low temperatures in low melting-point metals such as solder and zinc, as well as plastics

BASICS OF CREEP MATERIAL

- **Applications**
  - Asphalt
  - Concrete
  - pre-stressed bolts
- **Loading & Unloading**
  - Constant load leads to creep strains over time in addition to elastic and plastic strains at time 0
  - Upon load removal the elastic strains recover immediately, part of the creep strains over time, the other part and ε^{pl} remain



Relaxation



Creep

INPUT OF CREEP MATERIAL

- **Two input types of creep strain rate**
  - Exponent input

temperature

$$\dot{\bar{\varepsilon}}^c = A\bar{\sigma}^m \cdot (\bar{\varepsilon}^c)^n \cdot T^p \cdot (qt^{q-1})$$

time

MATVP	MID	A		m	n	p	q		

  - Table input

$$\dot{\bar{\varepsilon}}^c = A \cdot f(\bar{\sigma}) \cdot g(\bar{\varepsilon}^c) \cdot h(T) \cdot \frac{dk(t)}{dt}$$

MATVP	MID	A	IT3D						

ID of a TABL3Di

can be coupled with MAT1, MAT2, MATORT, MAT9, MATEP

INPUT OF CREEP MATERIAL (CONT.)

- **When coupled with MATEP**

MATEP	MID	FORM	Y0	FID	IMPCREEP	WKHARD	METHOD	H	
	IMPCREEP	vMISES							

- **Choose CREEP parameters**

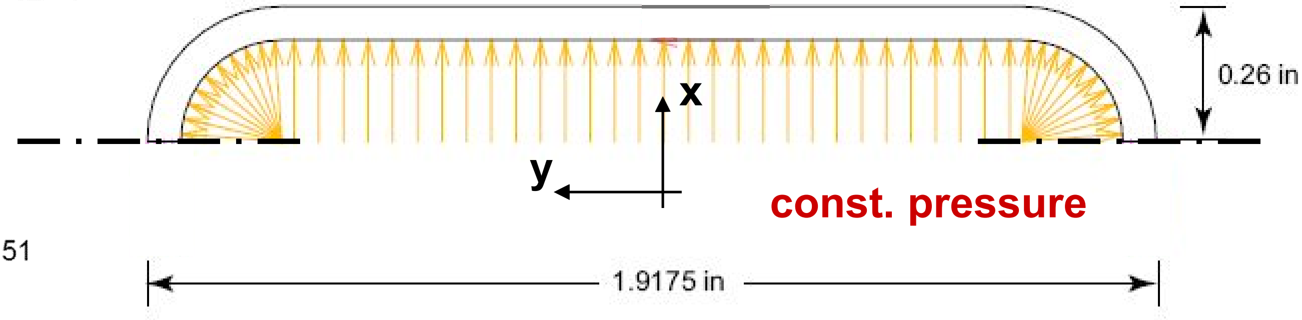
NLMOPTS	CREEP	VALC1	VALC2	VALC3	VALC4				

- **VALC1**
  - 0 – Maxwell model
  - 1 – visco-plastic creep
- **VALC2**
  - 1 – explicit Kelvin model
- **VALC3**
  - 1 – implicit Maxwell creep or implicit visco-plastic model
- **VALC4**
  - 0 – elastic tangent (Default)
  - 1 – secant tangent
  - 2 – radial return

EXAMPLE FOR CREEP MATERIAL

- **Creep of a Tube**
  - plane strain QUAD4
  - E = 21.46 Mpsi
  - $\nu = 0.3$
  - Norten Creep
    - $\dot{\varepsilon} = 4.E\text{-}24 \cdot \sigma^{4.51}$

Model



const. pressure

- **Two Steps (nonlinear statics followed by creep)**

```
SUBCASE 1
  STEP 1                    $ static step ( ANALYSIS=NLSTATIC is default )
        NLSTEP = 1
        SPC = 2
        LOAD = 3
  STEP 2                    $ creep step ( also NLSTATIC )
        NLSTEP = 2
        SPC = 2
        LOAD = 3
```

UG #51

EXAMPLE FOR CREEP (CONT.)

- **Iteration Strategy**

NLSTEP	ID	TOTTIME	CTRLDEF						
	"GENERAL"	MAXITER	MINITER	MAXBIS	CREEP				

```
BEGIN BULK
NLOUT   101     CCASTRSS        CCRPSTRN        EQCRSTRN
NLMOPTS,CREEP,
,LRGSTRN,1,
,ASSM,ASSUMED
PARAM   PRTMAXIM YES
NLSTEP      1   1.00E-09
        GENERAL       40       0      10       0
        FIXED          1       1
        MECH      PV    0.00    .100E-010.00        PFNT       0       3
NLSTEP      2   3.47e6
        GENERAL       40       0      10       1
        ADAPT   5.0E-06 1.0E-12 .500           6 1.50          0  999999
                      0 .200E-03       1       1       1 .100    10.0
        MECH      PV    0.00    .100    0.00        PFNT      -1      3
```

creep

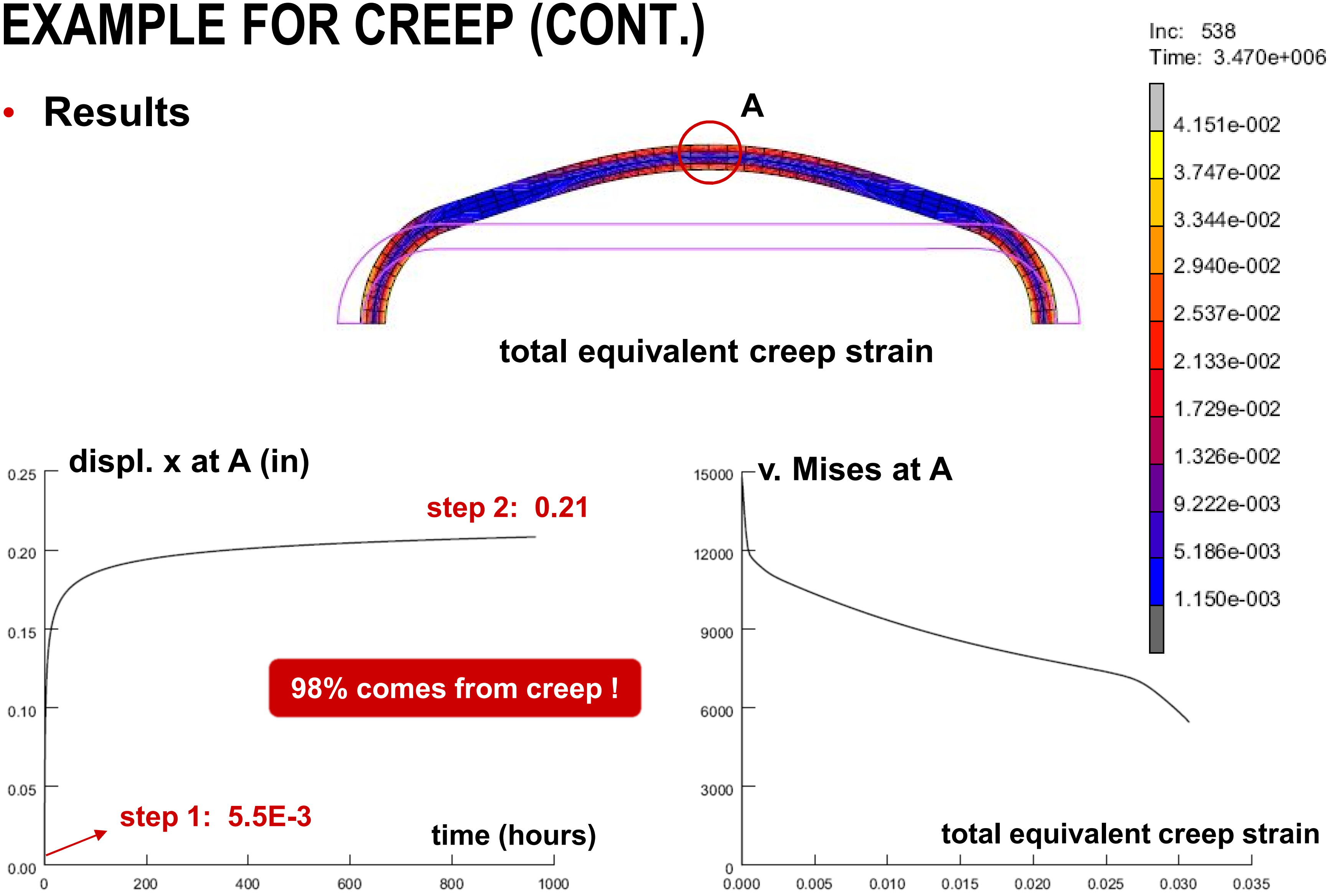
EXAMPLE FOR CREEP (CONT.)

- **Elements & Materials**

```
.
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
MAT1    1       2.14+7          .3
MATVP   1       4.E-24          4.51
PLPLANE 1       1
PSHLN2  1       1       1       1.0
+       C4      PLSTRN  L
CQUAD4  1       1       1       2       3       4
.
.
```

EXAMPLE FOR CREEP (CONT.)

- **Results**



total equivalent creep strain

step 2: 0.21

98% comes from creep !

step 1: 5.5E-3

TABLE INPUT FOR NONLINEAR MATERIALS

TABLES1 AND TABLEST

- **TABLES1 (MATS1, CREEP, MATEP, MATG, MATHE, MCOHE)**

1	2	3	4	5	6	7	8	9	10
TABLES1	TID	TYPE							
	x1	y1	x2	y2	x3	y3	-etc.-	“ENDT”	

  - For SOL 400, **TYPE** denotes the type of the stress-strain curve:
    - 1 – stress vs. total strain and
    - 2 – stress vs. plastic strain
  - For **MATS1** Bulk Data entry, only TYPE = 1 can be used. A user fatal error will be issued if TYPE = 2 is used. For MATEP Bulk Data entry both TYPE = 1 and 2 can be used.

- **TABLEST (MATS1, MATTEP, MATTHE, MCOHE)**

TABLEST	TID								
	T1	TID1	T2	TID2	T3	-etc.-			

TABL3Di

- **for MATEP, MATS3/8/ORT, MATHE/VE/VP, VCCT**

1	2	3	4	5	6	7	8	9	10
TABL3D0	ITID		KIND	EXTRP	ITIDR	ITIDS	ITIDB	SM	
	X1	Y1	X2	Y2	X3	Y3	X4	Y4	
	X5	Y5	-etc.-						

TABL3D1	ITID	NV	KIND1	KIND2	KIND3	KIND4	NW1	NW2	
	NW3	NW4	EXTRP1	EXTRP2	EXTRP3	EXTRP4			
	ITIDS1	ITIDB1	ITIDS2	ITIDB2	ITIDS3	ITIDB3	ITIDS4	ITIDB4	
	SM1	SM2	SM3	SM4					
	X11	X12	X13	X14	X15	X16	-etc.-		
	X21	X22	X23	X24	X25	X26	-etc.-		Enter if NW2>0
	X31	X32	X33	X34	X35	X36	-etc.-		Enter if NW3>0
	X41	X42	X43	X44	X45	X46	-etc.-		Enter if NW4>0
	Y1	Y2	Y3	Y4	Y5	Y6	-etc.-		See Remark 1

TABL3DI (CONT.)

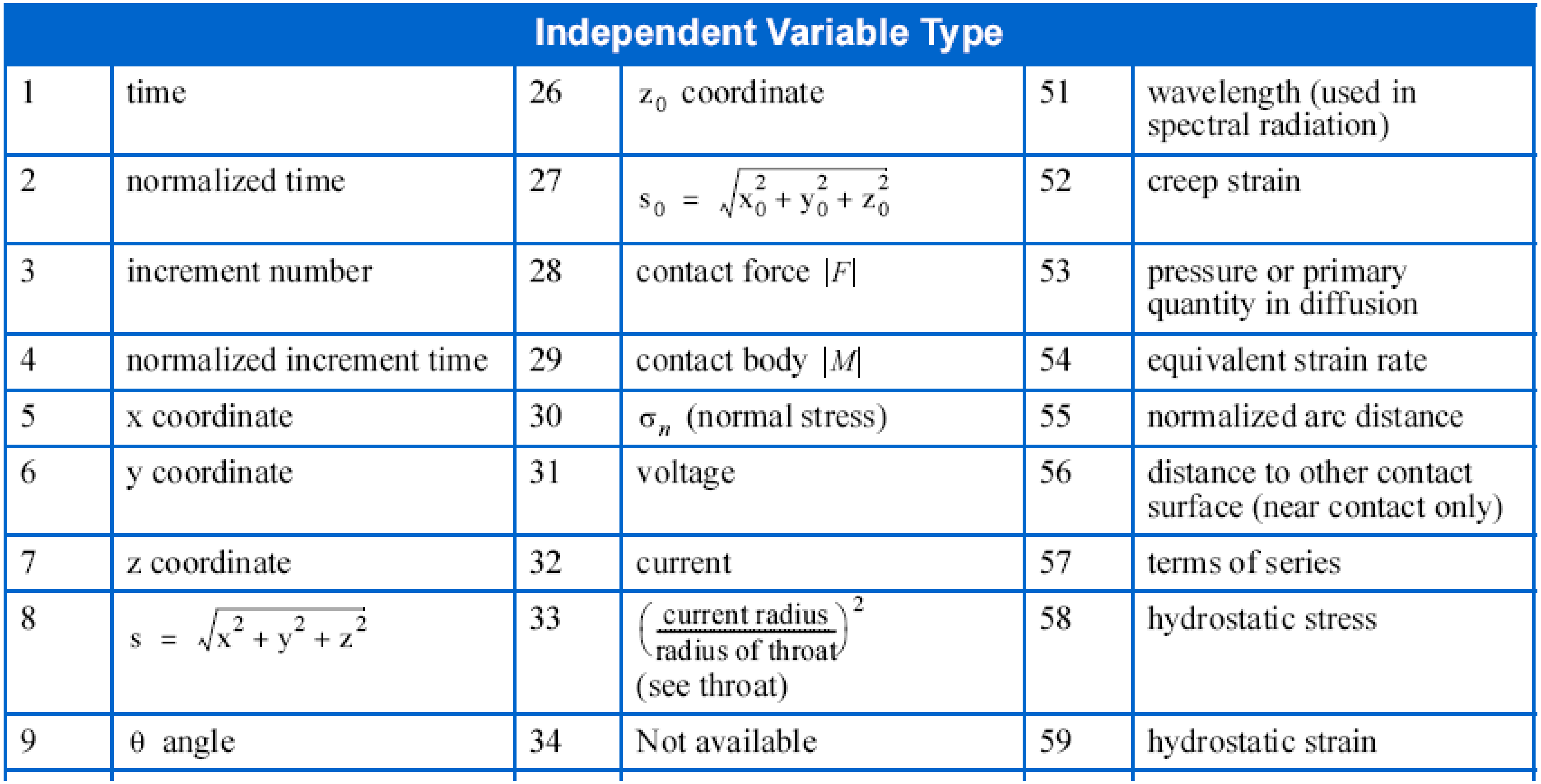
- **TABL3D2 has the same format. However the x data are entered differently:**
  - for TABL3D1 the x values are entered one row at a time
  - for TABL3D2 the x values are entered one point at a time
- **Function input**

1	2	3	4	5	6	7	8	9	10
TABL3D3	ITID	NV	KIND1	KIND2	KIND3	KIND4	NW1	NW2	
	NW3	NW4	EXTRP1	EXTRP2	EXTRP3	EXTRP4			
	ITIDS1	ITIDB1	ITIDS2	ITIDB2	ITIDS3	ITIDB3	ITIDS4	ITIDB4	
	SM1	SM2	SM3	SM4					
	Formula								See Remark 3

TABL3DI (CONT.)

- **KIND is the type of independent variable**

Table 8-49



Independent Variable Type					
1	time	26	z_0 coordinate	51	wavelength (used in spectral radiation)
2	normalized time	27	$s_0 = \sqrt{x_0^2 + y_0^2 + z_0^2}$	52	creep strain
3	increment number	28	contact force $\|F\|$	53	pressure or primary quantity in diffusion
4	normalized increment time	29	contact body $\|M\|$	54	equivalent strain rate
5	x coordinate	30	σ_n (normal stress)	55	normalized arc distance
6	y coordinate	31	voltage	56	distance to other contact surface (near contact only)
7	z coordinate	32	current	57	terms of series
8	$s = \sqrt{x^2 + y^2 + z^2}$	33	$\left(\frac{\text{current radius}}{\text{radius of throat}}\right)^2$ (see throat)	58	hydrostatic stress
9	θ angle	34	Not available	59	hydrostatic strain

See QRG for more

GUIDELINES & LIMITATIONS

GUIDELINES FOR NL MATERIALS

- **For elasto-plastic materials, Cauchy stress and log strain (true stress and strain) data is expected**
  - At large strains, there are significant differences in how the stresses and strains are defined
- **Beware of data extrapolation**
  - extend the work hardening data sufficiently to cover the entire strain range. Extrapolation will be used if the range is exceeded.

GUIDELINES FOR NL MATERIALS (CONT.)

- **If a perfect plasticity model experiences convergence difficulties, use a more realistic plasticity model with non-zero work hardening**
  - Any material model in which the tangent stiffness is zero or negative will often cause convergence problems

GUIDELINES FOR NL MATERIALS (CONT.)

- **Check the material stability for elastomer materials**
  - use Patran “Experimental Data Fitting” menu
- **When fitting experimental data, engineering stress/strain data is expected**
- **Check that the material data covers the entire strain range:**
  - This can cause “elements inside out” errors
  - The analysis may not converge if any part of the model experiences strains beyond the stability limits of the material

SECTION 6

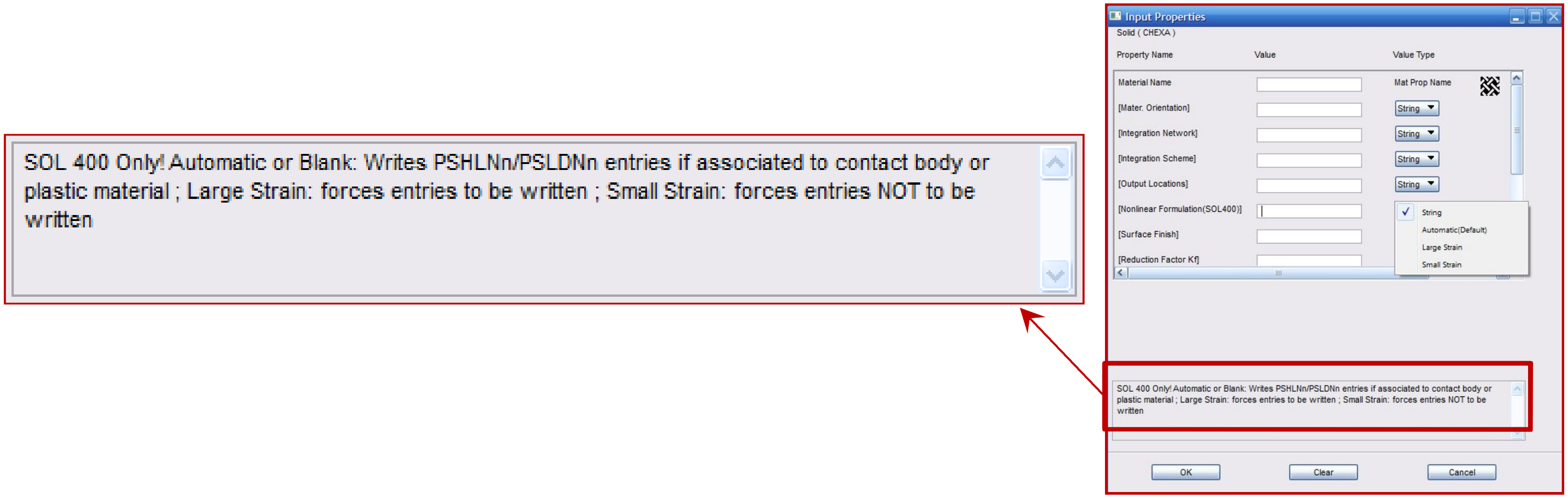
ADVANCED NONLINEAR ELEMENTS

OVERVIEW

- Advanced Conventional Elements
- Advanced Composite Elements
- Advanced Incompressible elements
- Nonlinear Connector Elements
- Nonlinear Kinematic Elements
- Additional Results Output

SOL 400 – ADVANCED ELEMENTS

- **SOL400 – What is it?**
  - – MSC Nastran solution sequence that implements MARC nonlinear ‘Advanced’ element and solver technology
  - – Patran provides ‘seamless’ interface to implementation of Marc elements
    - Uses standard Patran forms
    - Choosing appropriate choices automatically updates to advanced elements



SOL 400 Only! Automatic or Blank: Writes PSHLNn/PSLDNn entries if associated to contact body or plastic material ; Large Strain: forces entries to be written ; Small Strain: forces entries NOT to be written

Solid (CHEXA)

Property Name	Value	Value Type
Material Name		Mat Prop Name
[Mater. Orientation]		String
[Integration Network]		String
[Integration Scheme]		String
[Output Locations]		String
[Nonlinear Formulation(SOL400)]		✓ String Automatic(Default) Large Strain Small Strain
[Surface Finish]		
[Reduction Factor Kf]		

SOL 400 Only! Automatic or Blank: Writes PSHLNn/PSLDNn entries if associated to contact body or plastic material ; Large Strain: forces entries to be written ; Small Strain: forces entries NOT to be written

AVAILABLE ELEMENT TYPES

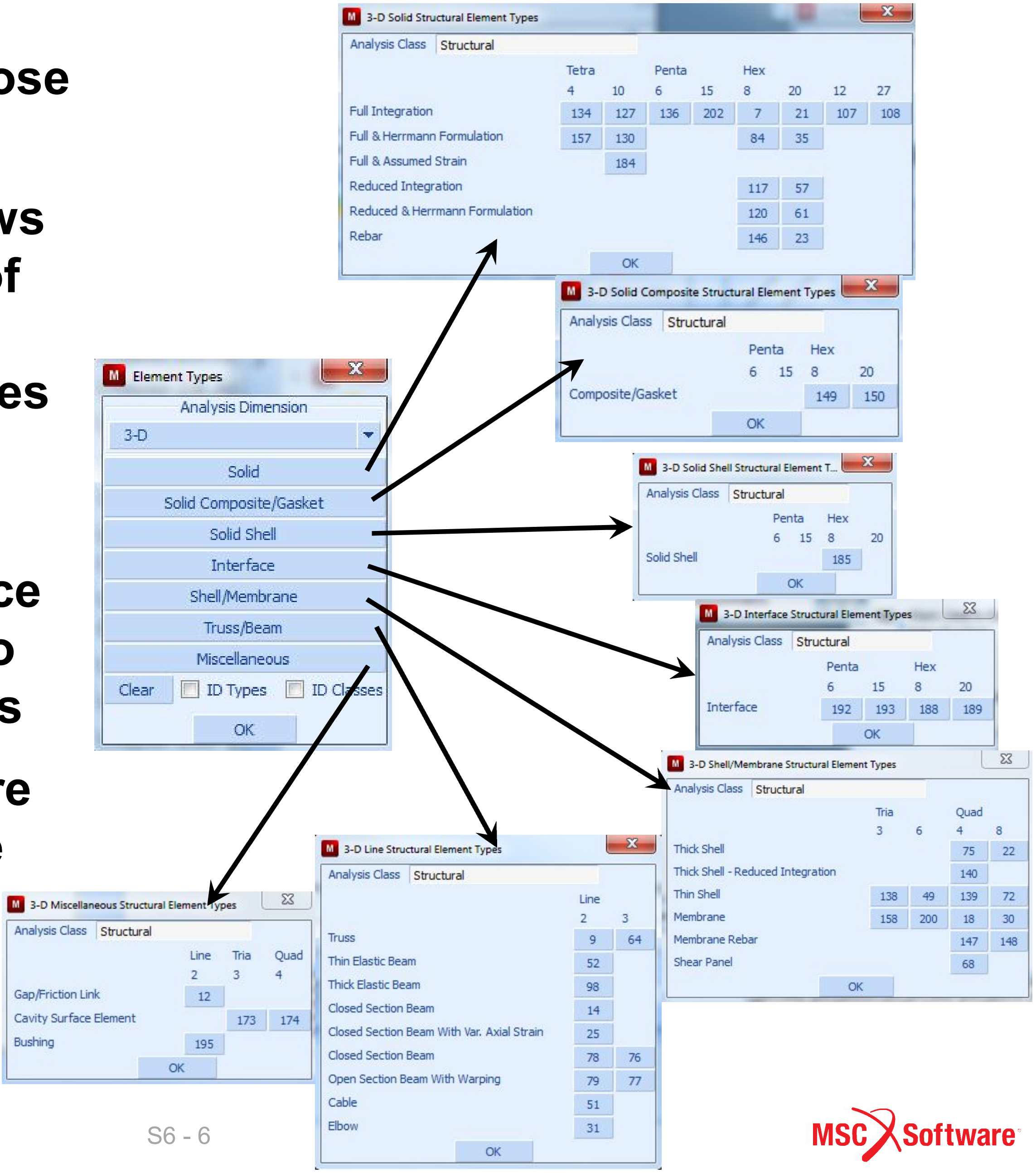
- **Advanced Conventional Elements**
  - 1D Elements: ROD, BAR, BEAM
  - 2D and 3D Solid Elements
  - Shell Element
- **Advanced Composite Elements**
  - 2D and 3D Solid Composites
  - Composite Shell Element
- **Nonlinear Connector Elements**
  - BUSH
  - WELD and FAST
- **Nonlinear Kinematic Elements**
  - RBE1, RBE2, RBE3
  - RBAR, RBAR1
  - RJOINT
- **Additional Results Output Requests for all Advanced Nonlinear Elements**

ADVANCED ELEMENT TYPES

- **The terminology 'Advanced Elements' indicates that SOL400 uses elements of greater capability than 'Standard' MSC Nastran elements in other solution sequences**
- **Typically this means large displacement and large strain**
  - Large displacement does not require large strain (think 'fishing pole'!)
  - Large strain, on the other hand , typically does require large displacement
  - Large strain typically requires more complicated material models
- **If large displacement/large strain capabilities are turned off these elements reduce to 'standard' element capabilities (i.e. large displacement /large strain are not required for the use of these elements)**
- **Remember, with large deformation/large strain activated the solutions will be nonlinear and require iterative solutions – longer run times!**

WHAT ELEMENT IS THE 'RIGHT' ELEMENT?

- **200+ element types to chose from**
- **Practical experience shows that a very small subset of the full element library accounts for most analyses**
- **In most cases the model element types - bar/shell/solid – will reduce the users options down to just a few element choices**
- **Images on these pages are from Mentat but these are selected in Patran as well by choosing appropriate options**



3-D Solid Structural Element Types

Analysis Class: Structural

	Tetra		Penta		Hex			
	4	10	6	15	8	20	12	27
Full Integration	134	127	136	202	7	21	107	108
Full & Herrmann Formulation	157	130			84	35		
Full & Assumed Strain		184						
Reduced Integration					117	57		
Reduced & Herrmann Formulation					120	61		
Rebar					146	23		

3-D Solid Composite Structural Element Types

Analysis Class: Structural

	Penta		Hex	
	6	15	8	20
Composite/Gasket			149	150

3-D Solid Shell Structural Element T...

Analysis Class: Structural

	Penta		Hex	
	6	15	8	20
Solid Shell			185	

3-D Interface Structural Element Types

Analysis Class: Structural

	Penta		Hex	
	6	15	8	20
Interface	192	193	188	189

3-D Shell/Membrane Structural Element Types

Analysis Class: Structural

	Tria		Quad	
	3	6	4	8
Thick Shell			75	22
Thick Shell - Reduced Integration			140	
Thin Shell	138	49	139	72
Membrane	158	200	18	30
Membrane Rebar			147	148
Shear Panel			68	

3-D Line Structural Element Types

Analysis Class: Structural

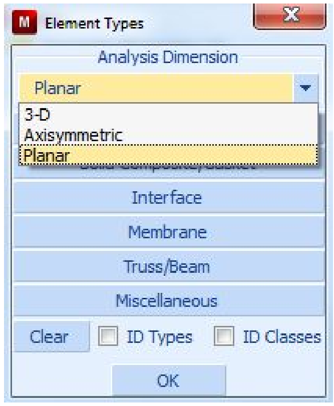
	Line	
	2	3
Truss	9	64
Thin Elastic Beam	52	
Thick Elastic Beam	98	
Closed Section Beam	14	
Closed Section Beam With Var. Axial Strain	25	
Closed Section Beam	78	76
Open Section Beam With Warping	79	77
Cable	51	
Elbow	31	

3-D Miscellaneous Structural Element Types

Analysis Class: Structural

	Line	Tria	Quad
	2	3	4
Gap/Friction Link	12		
Cavity Surface Element		173	174
Bushing	195		

A SIDE NOTE: ELEMENT DIMENSION



Analysis Dimension: Planar

- **Patran terminology**
  - – 0D/1D/2D/3D refer to element topology
    - 0D elements - point elements
    - 1D elements - line elements
    - 2D elements - shell or planar solids
      - – axisymmetric/plane strain/plane stress
    - 3D elements - solid elements
- **MARC terminology**
  - – 2D/3D refers to analysis ‘space‘
    - 3D are elements that have degrees of freedom (DOFs) that allow motion in x, y, and z directions (and sometimes rotX, rotY, and rotZ)
    - 2D elements are limited to motion in specific planes:
      - – Axisymmetric – all results limited to motion in global XY plan with the assumption that the models are solids of revolution about the global X axis and Y representing the radial direction
      - – Planar – plane stress/plane strain – all deformation limited to global XY plane

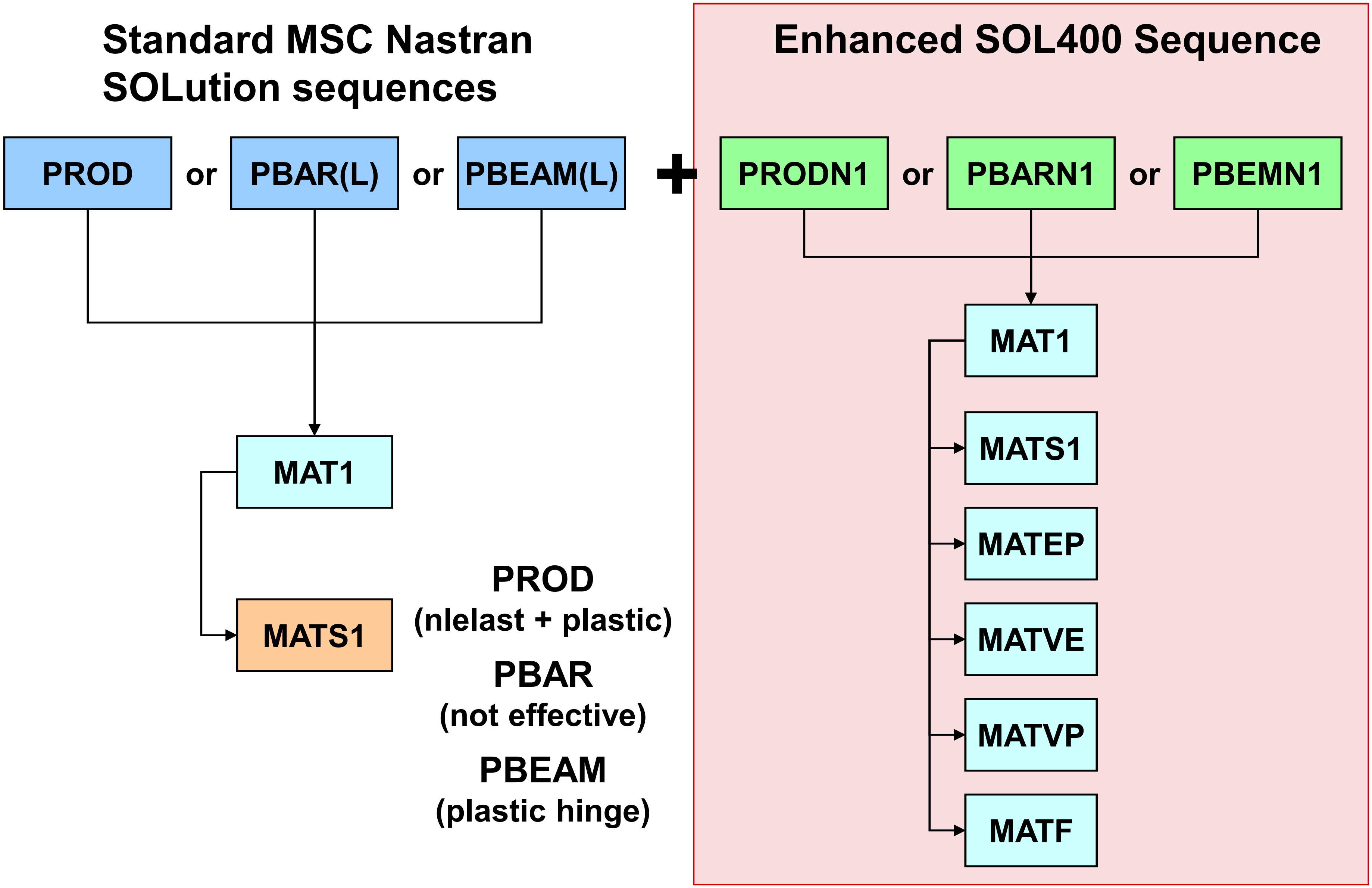
ADVANCED ELEMENT TYPES

- **The choice of element types is generally dictated by the application, i.e. bar, shell, solid, etc.**
- **For each element type, the unique combination of 'Behavior' and 'Integration' options dictates which element type will be used**
- **The most common Integration options include:**
  - Linear
  - Quadratic
  - Cubic
  - Linear/Reduced Int/Hourglass
  - Quadratic/Reduced Int
- **Behavior is dictated by the element type and includes:**
  - Bar
  - Rod
  - Beam
  - Membrane
  - Doubly Curved Thick or Thin Shell
  - Composite
  - Plane stress/strain
  - Solid

ADVANCED ELEMENT TYPES

- **Specification of the Integration and Behavior types is provided to SOL400 on the supplemental property entries:**
  - PRODN1 (1D elements)
  - PBARN1 (1D elements)
  - PBEMN1 (1D elements)
  - PSHLN1 (2D shell elements)
  - PSHLN2 (2D solid elements – plane strain, plane stress, axisymmetric)
  - PSLDN1 (3D solid elements)
  - More available but these are the most commonly used

ENHANCED INPUT FOR 1D



Standard MSC Nastran SOLution sequences

PROD or PBAR(L) or PBEAM(L)

MAT1

MATS1

PROD
(nlelast + plastic)

PBAR
(not effective)

PBEAM
(plastic hinge)

\+

Enhanced SOL400 Sequence

PRODN1 or PBARN1 or PBEMN1

MAT1

MATS1

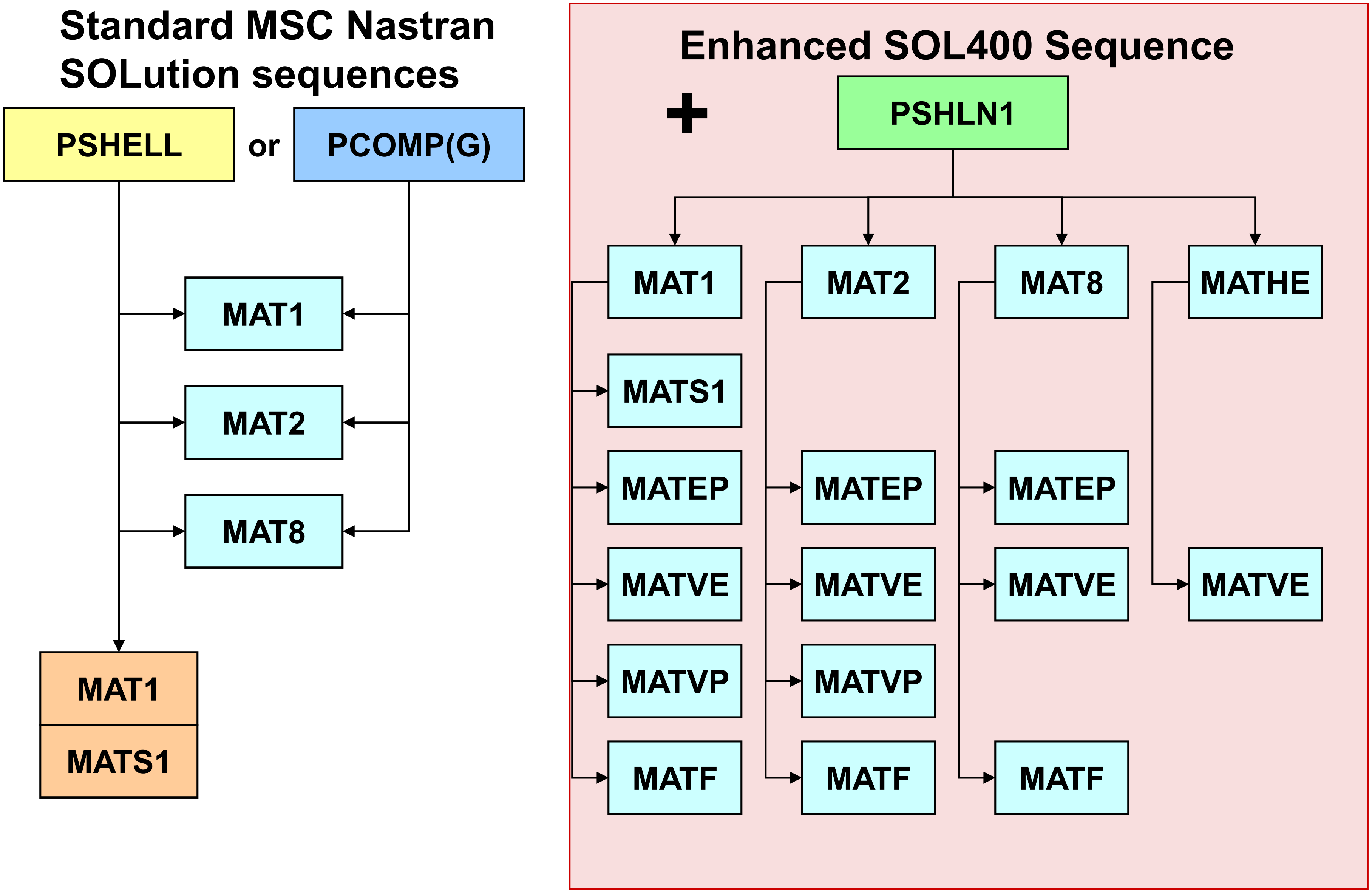
MATEP

MATVE

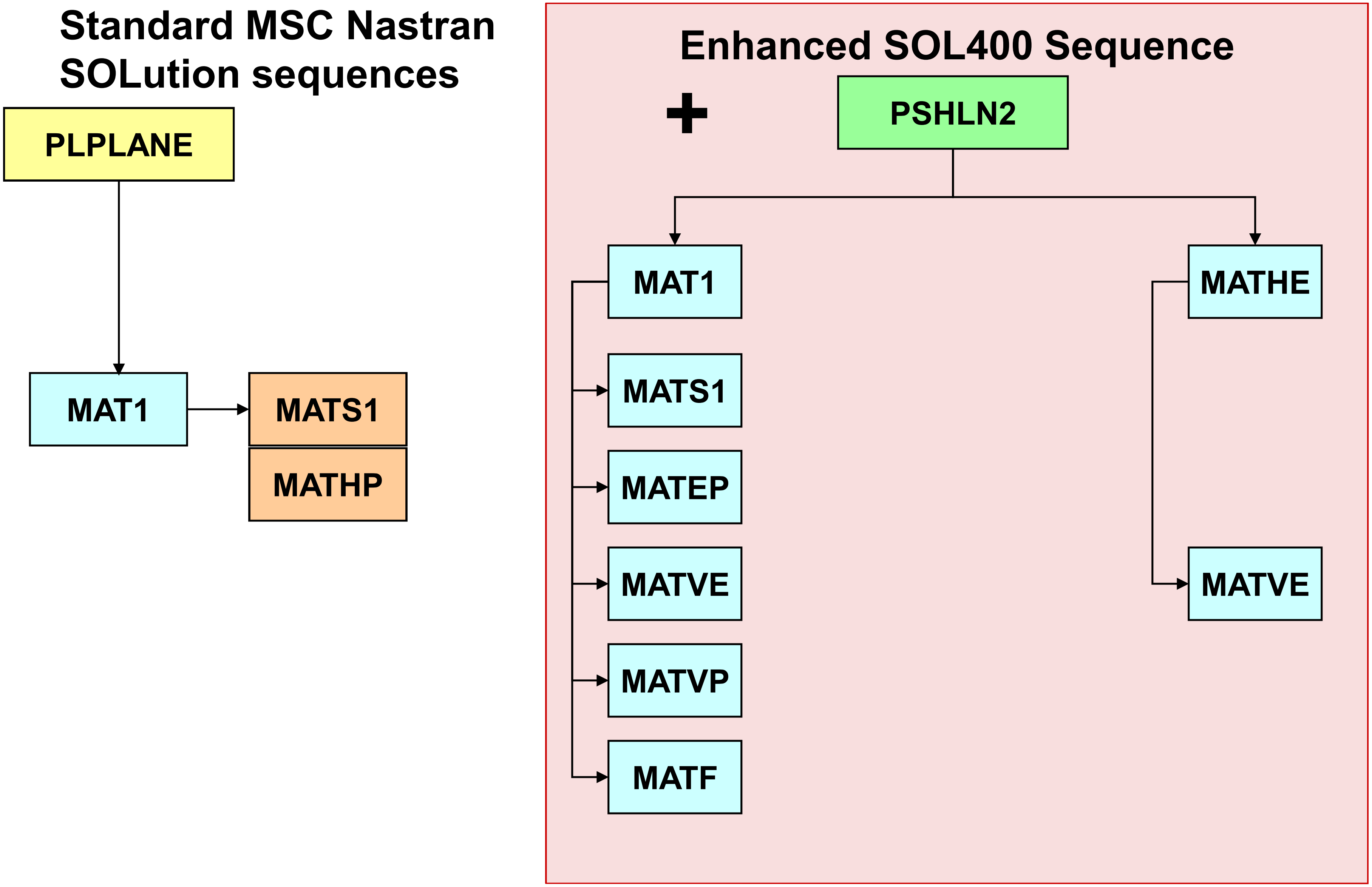
MATVP

MATF

ENHANCED INPUT FOR SHELLS



ENHANCED INPUT FOR 2D SOLIDS



Standard MSC Nastran SOLution sequences

PLPLANE

MAT1

MATS1

MATHP

Enhanced SOL400 Sequence

+

PSHLN2

MAT1

MATS1

MATEP

MATVE

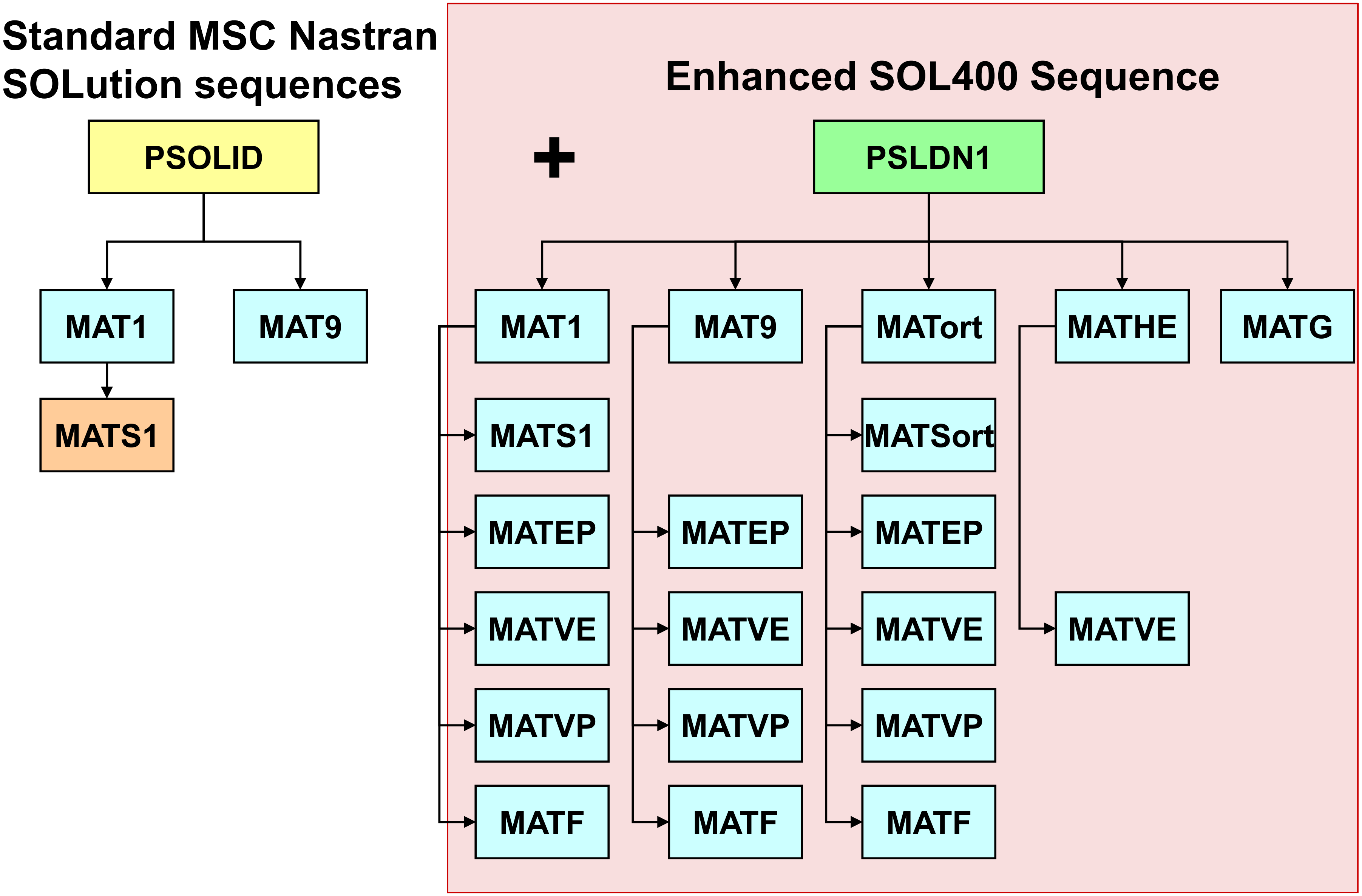
MATVP

MATF

MATHE

MATVE

ENHANCED INPUT FOR 3D SOLIDS



Standard MSC Nastran SOLution sequences

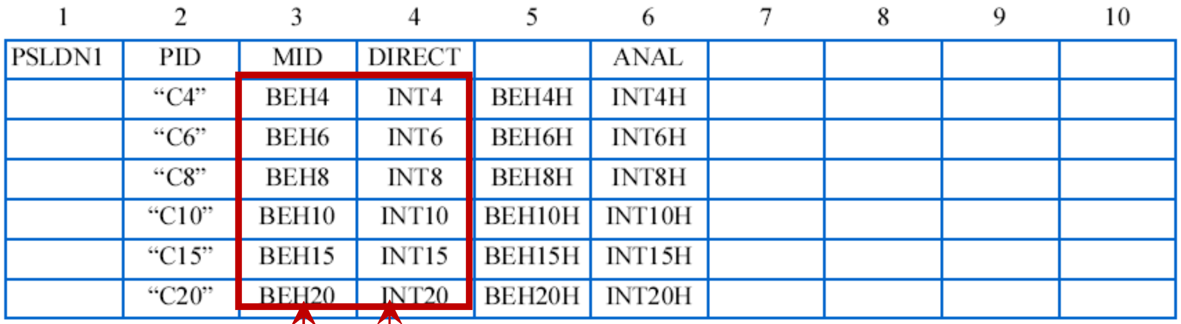
\+

Enhanced SOL400 Sequence

ADVANCED ELEMENT TYPES

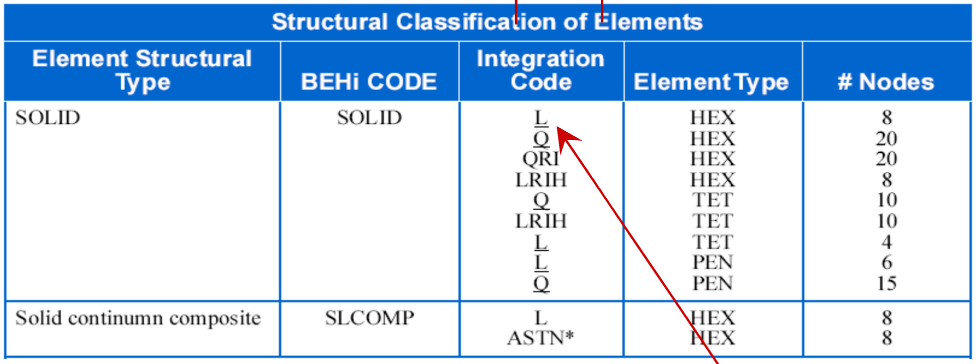
- **Typical supplemental property entry, this one for PSOLID, the PSLDN1 from the MSC Nastran QRG showing specification of Behavior and Integration**

Format:

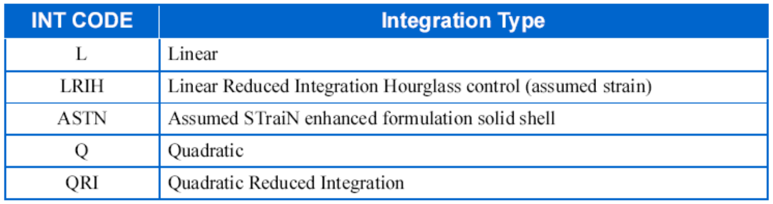


1	2	3	4	5	6	7	8	9	10
PSLDN1	PID	MID	DIRECT		ANAL				
	"C4"	BEH4	INT4	BEH4H	INT4H				
	"C6"	BEH6	INT6	BEH6H	INT6H				
	"C8"	BEH8	INT8	BEH8H	INT8H				
	"C10"	BEH10	INT10	BEH10H	INT10H				
	"C15"	BEH15	INT15	BEH15H	INT15H				
	"C20"	BEH20	INT20	BEH20H	INT20H				

- BEHi (behavior) and INTi (integration type)



Structural Classification of Elements				
Element Structural Type	**BEHi CODE**	**Integration Code**	**Element Type**	**# Nodes**
SOLID	SOLID	L Q QRI LRIH Q LRIH L L Q	HEX HEX HEX HEX TET TET TET PEN PEN	8 20 20 8 10 10 4 6 15
Solid continumn composite	SLCOMP	L ASTN*	HEX HEX	8 8



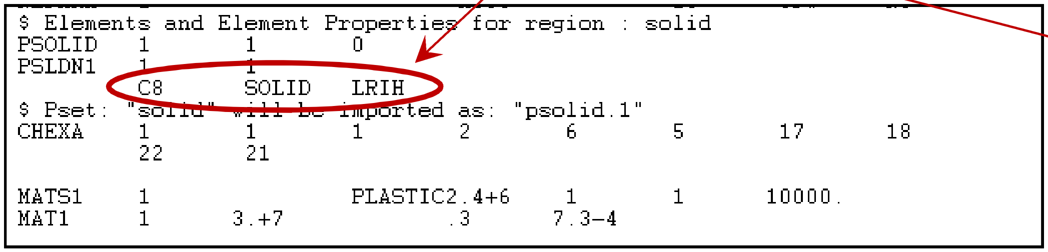
INT CODE	Integration Type
L	Linear
LRIH	Linear Reduced Integration Hourglass control (assumed strain)
ASTN	Assumed STraiN enhanced formulation solid shell
Q	Quadratic
QRI	Quadratic Reduced Integration

Underline indicates default – for 8 node hex the default behavior is Linear integration/Solid Behavior

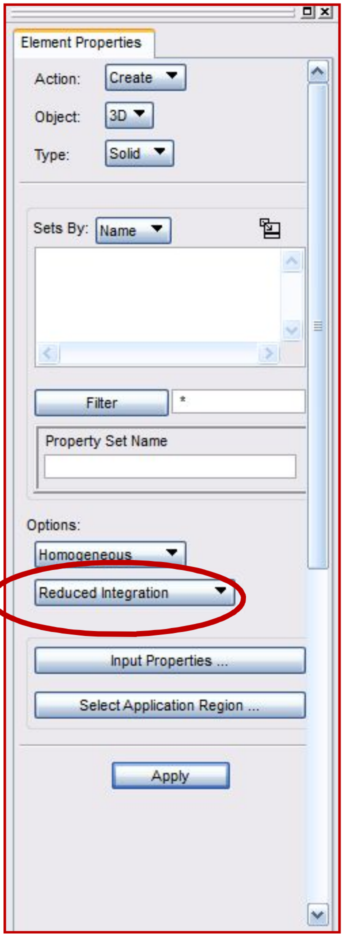
ADVANCED ELEMENT TYPES

- **Selection of the Integration and Behavior specifications can easily be controlled from within Patran, no manual editing of input file required**
- **Can also be specified via manual editing**

In this example the user has requested reduced integration/Solid Behavior



```
$ Elements and Element Properties for region : solid
PSOLID   1       1       0
PSLDN1   1       1
         C8      SOLID   LRIH
$ Pset: "solid" will be imported as: "psolid.1"
CHEXA    1       1       1       2       6       5       17      18
         22      21

MATS1    1               PLASTIC2.4+6    1       1      10000.
MAT1     1      3.+7            .3      7.3-4
```



Action: Create
Object: 3D
Type: Solid
Sets By: Name
Options:
Homogeneous
Reduced Integration

SPECIAL CASES OF NONLINEAR ELEMENTS

- **CQUADR/CTRIAR**
- **Advanced Composite elements**
- **Connectors**
- **Kinematic Elements**

NONLINEAR QUADR / TRIAR

- **CQUADR/CTRIAR elements are enhancements to the standard workhorse CQUAD4/CTRIA3 elements**
- **Add in-plane rotational stiffness to avoid the issue of zero stiffness in that direction for CQUAD4/CTRIA3 elements**
- **Generally considered more accurate than CQUAD4/CTRIA3**
- **QUADR/TRIAR elements have been extended to nonlinear static and transient analysis for SOL 400**
- **Benefits:**
  - No K6ROT necessary
  - MOMENTs and SPC normal to plane rotations can be applied
  - The membrane performance of QUADR/TRIAR is superior to that of QUAD4/TRIA3
  - For material nonlinear analysis, 4 (QUADR) or 3 (TRIAR) in-plane integration points are used. This will give better results.

NONLINEAR QUADR / TRIAR

- **Benefits (cont.):**
  - Green strains are implemented
  - The stress output location can be either corner or integration points
- **Type of analyses:**
  - Material nonlinear analysis – the material models are elastoplastic and nonlinear elastic
  - Geometric nonlinear analysis – PARAM, LGDISP,1
  - Geometric nonlinear analysis for temperature dependent composites
    - PARAM, COMPMATT, YES
    - PARAM, EPSILONT, SECANT or INTEGRAL

NONLINEAR QUADR / TRIAR

- **There are two methods to compute the out of plane shear stiffness:**
  - The stiffness method – new method implemented for QUADR
  - The flexibility method – the method used in QUAD4
- **These methods can be selected by the user with MDLPRM, QRSHEAR, n**
  - n=0, use the stiffness method if MID3≠0 on PSHELL entry and use the flexibility method if MID3=0. This is Default.
  - n=1, use the flexibility method
  - n=2, use the stiffness method

NONLINEAR QUADR / TRIAR

- **Green strain**
  - Green strain is implemented for QUADR/TRIAR. It can be requested by MDLPRM, GNLSTN, n
  - n=0, Small strain. Default
  - n=1, Green strain
- **Stress output locations**
  - For QUADR/TRIAR, the stress/force output locations can be either at the corner points or at the integration points. User can select the output location by MDLPRM, INTOUT, n
    - n=0, corner point output. Default
    - n=1, integration point output

NONLINEAR QUADR / TRIAR

- **An option is provided not to compute the differential stiffness by MDLPRM, NLDIFF, n**
  - n=0, compute the differential stiffness / follower force stiffness (default)
  - n=1, do not compute the differential stiffness
- **Please note, the differential stiffness / follower force stiffness can be deactivated and the solution will still be correct**
- **Limitations:**
  - No CREEP, creep material
  - No MATHP, hyperelastic material like rubber

ADVANCED COMPOSITE ELEMENTS

INPUT FOR 2D COMPOSITES

- **2D Composites used for cross-sectional analyses - plane strain, or axisymmetric - with layered properties**
- **PLCOMP – property for 2D composite**
  - Only applicable to SOL400
  - Unique in that it replaces (not supplements) a linear property entry

Format:

1	2	3	4	5	6	7	8	9	10
PLCOMP	PID	DIRECT	THICKOP	SB	ANAL				
	"C4"	BEH4	INT4	BEH4H	INT4H				
	"C8"	BEH8	INT8	BEH8H	INT8H				
	ID1	MID1	T1	THETA1					
	ID2	MID2	T2	THETA2					

-etc-

INPUT FOR 2D COMPOSITES

- **BEHi (behavior) and INTi (integration type)**

Implicit Structural Classification of Elements

Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Plane Strain composite	COMPS	L Q	QUAD QUAD	4 8
Axisymmetric composite	AXCOMP	L Q	QUAD QUAD	4 8

INT CODE	Integration Type
L	Linear
Q	Quadratic

INPUT FOR 2D COMPOSITES

- **MID (materials) and DIRECT (layer orientation)**

Implicit Structural Materials			
MAT1	**MAT3**	**MATORT**	**MATHE**
MATVE	<MATVE>	<MATVE>	MATVE
MATVP	MATVP	MATVP	
MATEP	MATEP	MATEP	
MATF	MATF	MATF	
MATS1		MATSORT	
MATM		MATM	

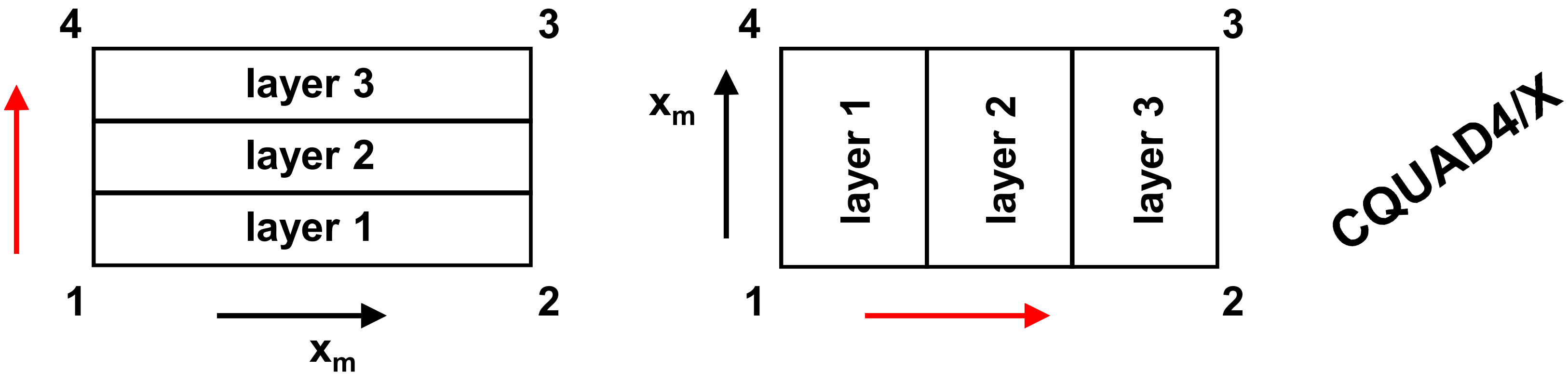
for AXCOMP

for COMPS

Layer Orientation			
DIRECT	**Normal to Layer edge**	**Layers run parallel from edge (ply numbering starts here)**	**to edge (ends here)**
1	Element Y direction	G1-G2	G4-G3
2	Element X direction	G1-G4	G2-G3

INPUT FOR 2D COMPOSITES

- **Meaning of DIRECT (layer orientation)**



DIRECT = +1 or -1 **DIRECT = +2 or -2**

DIRECT >0 – fractional layer thickness
<0 – absolute layer thickness

INPUT FOR 2D COMPOSITES

- **Former 2D Test Case also applied here**

```
BEGIN BULK
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
MATORT  1       100000. 100000. 100000. 0.3     0.3     0.3
+       38461.5 38461.5 38461.5
MATEP   1               200.                                    1000.
PLCOMP  10      1       10.
+       C4      COMPS
+       1       1       0.5     0.0
+       2       1       0.5     0.0
CQUAD4  101     10      1       2       3       4
```

- plane strain

- 2 plies for same number of integration points

- MATORT for demo

```
 S T R E S S E S   A N D   S T R A I N S   F O R   L A Y E R E D   C O M P O S I T E   E L E M E N T S

ELEMENT           INTEG.  --------------- S T R E S S E S  &  S T R A I N S --------------
     ID    PLY ID POINT ID       S11         S22         S33         S12         S23         S31
      1         1        1 -2.870E+02 -1.924E-03 -1.667E+02 -5.145E-14  0.000E+00  0.000E+00
                           -2.370E-03  1.361E-03 -8.065E-04 -1.338E-18  0.000E+00  0.000E+00
                         2 -2.870E+02 -1.924E-03 -1.667E+02  1.111E-13  0.000E+00  0.000E+00
                           -2.370E-03  1.361E-03 -8.065E-04  2.889E-18  0.000E+00  0.000E+00
                2        1 -2.870E+02 -1.924E-03 -1.667E+02 -8.686E-14  0.000E+00  0.000E+00
                           -2.370E-03  1.361E-03 -8.065E-04 -2.258E-18  0.000E+00  0.000E+00
                         2 -2.870E+02 -1.924E-03 -1.667E+02  7.393E-14  0.000E+00  0.000E+00
                           -2.370E-03  1.361E-03 -8.065E-04  1.922E-18  0.000E+00  0.000E+00
```

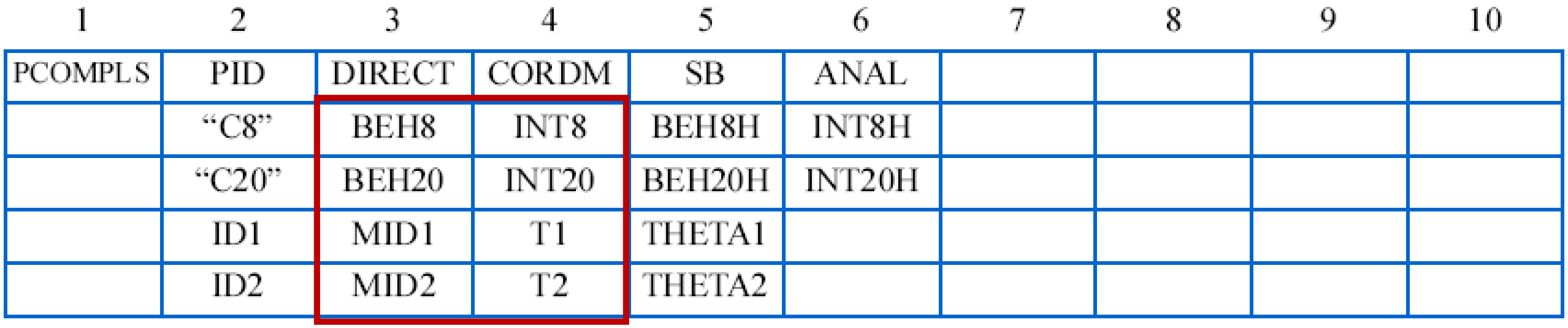
stresses the same, strains are the elastic strains.

to request the total strains NLOUT has to be used

INPUT FOR 3D COMPOSITES

- **3D Composites used for solid (i.e. HEX element) representation with layered properties**
- **PCOMPLS – property for 3D Composite**
  - Unique in that it replaces (not supplements) a linear property entry

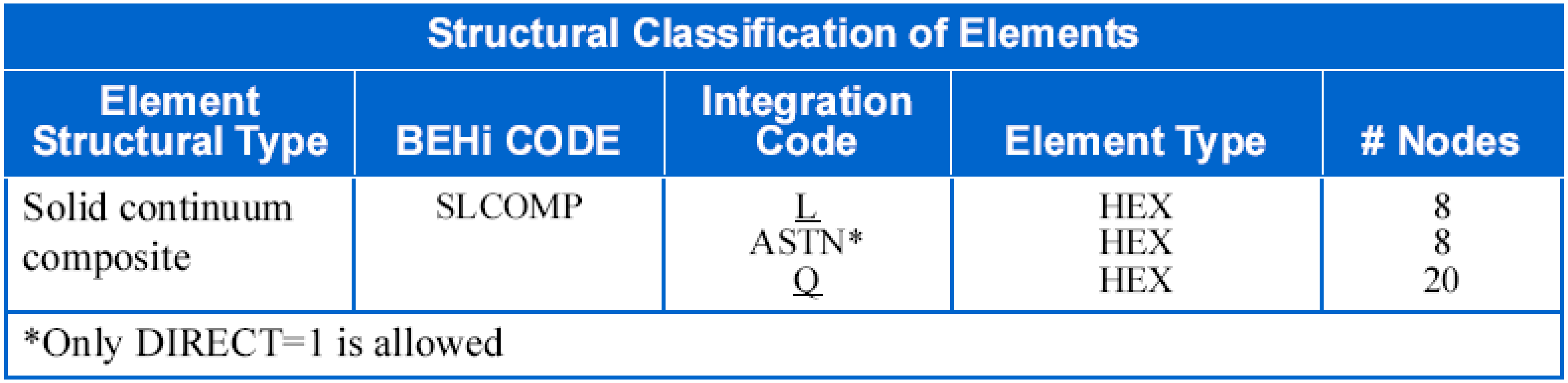
Format:



1	2	3	4	5	6	7	8	9	10
PCOMPLS	PID	DIRECT	CORDM	SB	ANAL				
	“C8”	BEH8	INT8	BEH8H	INT8H				
	“C20”	BEH20	INT20	BEH20H	INT20H				
	ID1	MID1	T1	THETA1					
	ID2	MID2	T2	THETA2					

INPUT FOR 3D COMPOSITES

- **BEHi (behavior) and INTi (integration type)**



Structural Classification of Elements

Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Solid continuum composite	SLCOMP	L	HEX	8
		ASTN*	HEX	8
		Q	HEX	20

*Only DIRECT=1 is allowed

INPUT FOR 3D COMPOSITES

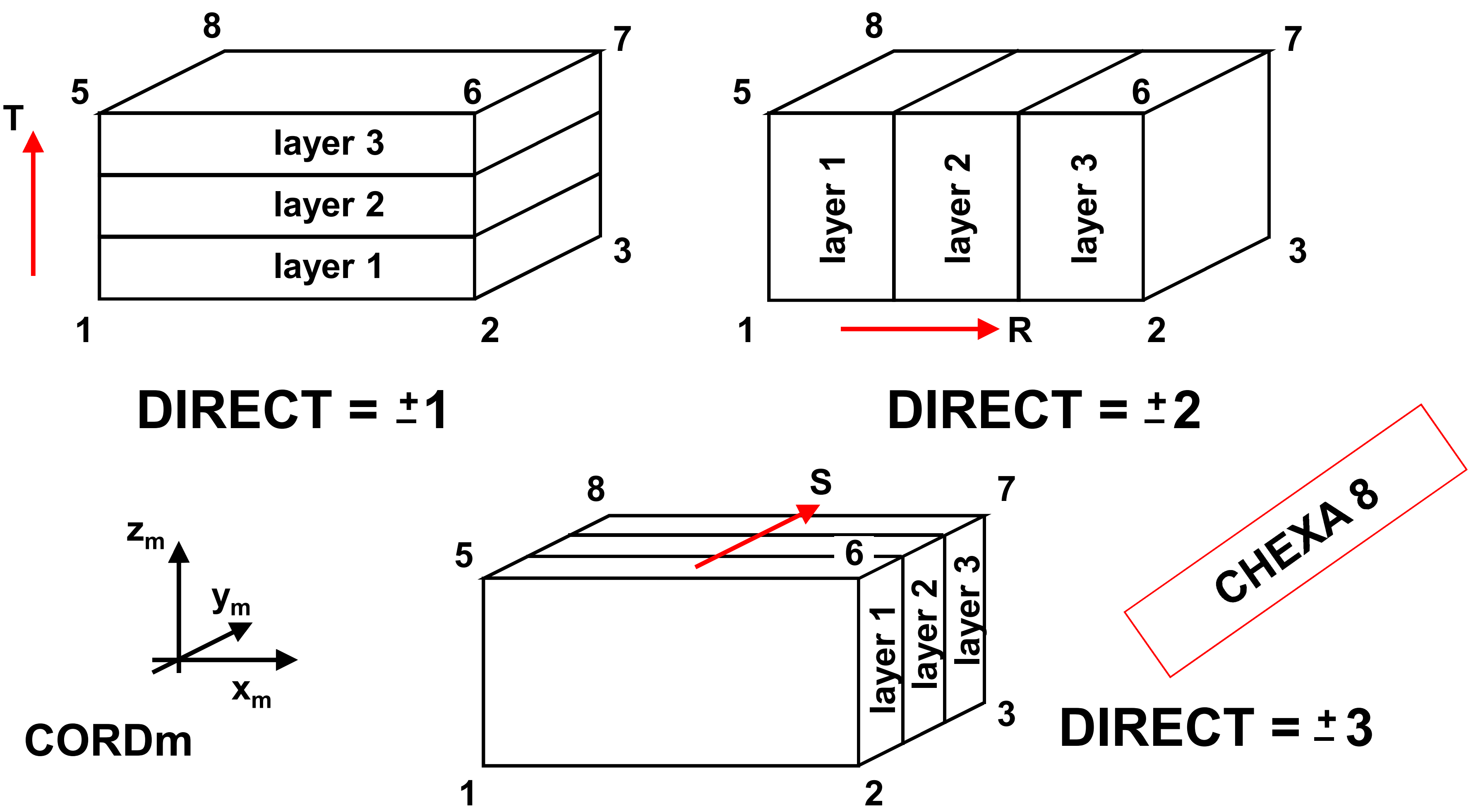
- **MID (materials) and DIRECT (layer orientation)**

Implicit Structural Materials			
MAT1	**MAT9**	**MATORT**	**MATHE**
MATVE	<MATVE>	<MATVE>	MATVE
MATVP	MATVP	MATVP	
MATEP	MATEP	MATEP	
MATF	MATF	MATF	
MATS1		MATSORT	
MATM		MATM	

Layer orientation			
DIRECT	**Normal to Layer Plane**	**Layers run parallel from face (ply numbering starts here)**	**to face (ends here)**
1	Element T direction	G1-G2-G3-G4	G5-G6-G7-G8
2	Element R direction	G1-G4-G8-G5	G2-G3-G7-G6
3	Element S direction	G1-G2-G6-G5	G4-G3-G7-G8

INPUT FOR 3D COMPOSITES

- **Meaning of DIRECT (layer orientation)**



DIRECT = ±1

DIRECT = ±2

DIRECT = ±3

INPUT FOR 3D COMPOSITES

- **Example for PCOMPLS – solid HEX8**

```
BEGIN BULK
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
MATORT  1       100000. 100000. 100000. 0.3     0.3     0.3
+       38461.5 38461.5 38461.5
MATEP   1               200.                                    1000.
PCOMPLS 1       1
+       C8      SLCOMP  L
+       1       1       0.5     0.0
+       2       1       0.5     0.0
CHEXA   101     1       1       2       3       4       5       6
+       7       8
```

same model as previous PSLDN1 example

```
 S T R E S S E S   A N D   S T R A I N S   F O R   L A Y E R E D   C O M P O S I T E   E L E M E N T S

ELEMENT            INTEG.  --------------- S T R E S S E S  &  S T R A I N S --------------
     ID    PLY ID POINT ID      S11         S22         S33         S12         S23         S31
      1         1        1 -2.772E+02 -3.792E-04 -3.792E-04  7.837E-14 -3.707E-14  7.723E-14
                           -2.772E-03  8.317E-04  8.317E-04  2.038E-18 -9.637E-19  2.008E-18
                         2 -2.772E+02 -3.792E-04 -3.792E-04 -9.877E-14 -3.631E-15 -3.535E-14
                           -2.772E-03  8.317E-04  8.317E-04 -2.568E-18 -9.440E-20 -9.190E-19
                         3 -2.772E+02 -3.792E-04 -3.792E-04  1.247E-13 -8.523E-14 -5.593E-14
                           -2.772E-03  8.317E-04  8.317E-04  3.242E-18 -2.216E-18 -1.454E-18
                         4 -2.772E+02 -3.792E-04 -3.792E-04 -5.422E-14 -4.472E-14 -9.106E-14
                           -2.772E-03  8.317E-04  8.317E-04 -1.410E-18 -1.163E-18 -2.367E-18
                2        1 -2.772E+02 -3.792E-04 -3.792E-04  6.552E-14  4.256E-14  8.577E-14
                           -2.772E-03  8.317E-04  8.317E-04  1.703E-18  1.107E-18  2.230E-18
                         2 -2.772E+02 -3.792E-04 -3.792E-04 -4.464E-14  1.240E-13 -1.435E-14
                           -2.772E-03  8.317E-04  8.317E-04 -1.161E-18  3.225E-18 -3.732E-19
                         3 -2.772E+02 -3.792E-04 -3.792E-04  9.989E-14  1.732E-14 -1.898E-14
                           -2.772E-03  8.317E-04  8.317E-04  2.597E-18  4.503E-19 -4.935E-19
                         4 -2.772E+02 -3.792E-04 -3.792E-04 -1.330E-14  5.600E-14 -4.757E-14
                           -2.772E-03  8.317E-04  8.317E-04 -3.457E-19  1.456E-18 -1.237E-18
```

**stresses the same,
strains are the elastic strains.**

INPUT FOR 3D COMPOSITES

- **Example for PCOMPLS – solid shell HEX8**

```
BEGIN BULK
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
MATORT  1       100000. 100000. 100000. 0.3     0.3     0.3
+       38461.5 38461.5 38461.5
MATEP   1               200.                                    1000.
PCOMPLS 1       1
+       C8      SLCOMP  ASTN
+       1       1       0.5     0.0
+       2       1       0.5     0.0
CHEXA   101     1       1       2       3       4       5       6
+       7       8
```

same model as for PSLDN1

```
 S T R E S S E S   A N D   S T R A I N S   F O R   L A Y E R E D   C O M P O S I T E   E L E M E N T S

 ELEMENT          INTEG.  --------------- S T R E S S E S  &  S T R A I N S --------------
      ID    PLY ID POINT ID      S11         S22         S33          S12         S23         S31
       1         1        1 -2.772E+02 -3.792E-04 -3.792E-04  1.828E-14 -1.426E-14 -8.839E-14
                            -2.772E-03  8.317E-04  8.317E-04  4.753E-19 -3.709E-19 -2.298E-18
                 2        1 -2.772E+02 -3.792E-04 -3.792E-04  2.960E-14 -2.105E-14 -9.214E-14
                            -2.772E-03  8.317E-04  8.317E-04  7.697E-19 -5.473E-19 -2.396E-18
```

stresses the same,
strains are the elastic strains.

COMPOSITE SHELL ELEMENTS

- **Composite shells - most commonly used FEA representation in aerospace type structures**
- **For nonlinearlinear must, in addition to standard PCOMP(G), must supplement data with PSHLN1 (behavior – same as in above discussion of PSHELL) and PCOMPF (integration control) entry:**

1	2	3	4	5	6	7	8	9	10
PCOMPF	INT	PID1	THRU	PID2	BY	N			

Alternate Formats:

1	2	3	4	5	6	7	8	9	10
PCOMPF	INT	PID	PID	PID1	THRU	PID2	PID3	THRU	
	PID4	PID5	TO	PID6	PID	PID	PID	PID7	
	THRU	PID8	BY	N					

1	2	3	4	5	6	7	8	9	10
PCOMPF	INT	ALL							

- INT=1 (Default), conventional through the thickness integration of each layer, allows all available material behavior through the thickness.
- INT=2, linear elastic material, fast-integrated through the thickness - thermal strains and temperature dependent material properties are not allowed.
- INT=3, linear elastic material, fast integrated through the thickness.

COMPOSITE SHELL ELEMENTS

- **Example for PCOMP(G) showing PSHLN1 and PCOMPF**

```
BEGIN BULK
PARAM   LGDISP  1
NLMOPTS LRGSTRN 1
+       TSHEAR  TSHEAR
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
MAT8    1       140000. 12000.  0.26    5800.   5400.   5400.   1.5-9
MATF    1       0       100.
+       CRI     1       2000.   1500.   70.     230.
+       90.
$
PCOMP   1                       100.    STRN                    SYM
+       1       0.5     90.     YES
+       1       0.5     0.      YES
+       1       0.5     90.     YES
+       1       0.5     0.      YES
+       1       0.5     90.     YES
+       1       0.5     0.      YES
+       1       0.5     90.     YES
+       1       0.5     0.      YES
PCOMPF  2       1
PSHLN1  1       1       1
```

for quadratic shear calculation

maximum stress criterion

Same model as for PSHELL:

u_{max} = 36.8 against 36.7 mm
weight = 12 against 13.5 kg

ADVANCED INCOMPRESSIBLE ELEMENTS

- **Incompressible elements are generally used to model incompressible materials such as rubber**
- **Three advanced incompressible elements are available in SOL400**
  - TRIA3 plane strain element
  - TRIA3 axisymmetric element
  - TET4 solid element
- **Those elements are specified by the Behavior types on the supplemental property entries**
  - IPS on PSHLN2 for TRIA3 incompressible plane strain element
  - IAX on PSHLN2 for TRIA3 incompressible axisymmetric element
  - ISOL on PSLDN1 for TET4 incompressible solid element

NONLINEAR CONNECTOR ELEMENTS

- **Types available in SOL400**
  - BUSH
  - WELD/FAST
  - Kinematic:
    - RBAR
    - RBAR1
    - RJOINT
    - RBE1
    - RBE2
    - RBE3
    - RTRPLT
    - RTRPL1

CONNECTOR ELEMENT – BUSH

- **BUSH element is a generalized spring-damper element used to model everything from special connectors to gap elements**
  - Large rotations are included
  - FUSE capability can be used to model bearing failure in rotor dynamics

CONNECTOR ELEMENT – BUSH

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH	EID	PID	GA	GB	GO/X1	X2	X3	CID	
	S	OCID	S1	S2	S3				

+

PBUSH	PID	“K”	K1	K2	K3	K4	K5	K6	
		“B”	B1	B2	B3	B4	B5	B6	
		“GE”	GE1	GE2	GE3	GE4	GE5	GE6	
		“RCV”	SA	ST	EA	ET			

+

PBUSHT	PID	“K”	TKID1	TKID2	TKID3	TKID4	TKID5	TKID6	
		“B”	TBID1	TBID2	TBID3	TBID4	TBID5	TBID6	
		“GE”	TGEID1	TGEID2	TGEID3	TGEID4	TGEID5	TGEID6	
		“KN”	TKNID1	TKIND2	TKNID3	TKIND4	TKIND5	TKIND6	
			FDC	FUSE	DIR	OPTION	LOWER	UPPER	
			FRATE	LRGR					

FUSE

CONNECTOR ELEMENT – BUSH

- **Large Rotation Option**
  - Controlled using LRGR flag on the PBUSHT entry
  - For the axial BUSH element(1) LRGR is ignored
  - For BUSH defined using a coordinate system CID or an orientation vector v
  - LRGR=0 (Default): the element coordinate system is rotated with the rotation of grid A for both the CID and the v vector
  - LRGR=1: will suppress large rotation at end A. The initial CID and the v vector will remain unchanged
  - LRGR=2: a mid-increment method is used to rotate the element system for the v vector

---

(1) grids A and B are non-coincident and vector v nor CID are specified

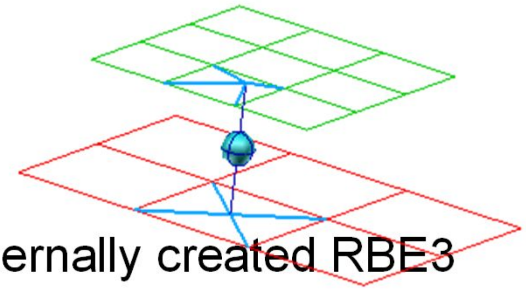
CONNECTOR ELEMENT – BUSH

- **Fuse option, with following entries on PBUSHT**
  - FDC, force-deflection curve rule
    - NR: all directions are independent (default)
    - TRXY, TRXZ or TRYZ: radial dependence, useful for radial bearings. For example, the force results from the length of ux and uy times K1.
    - TS: spherical dependence. The force results from the length of ux+ uy+ uz times K1.
  - FUSE
    - 0: no fuse (default)
    - 1: associated elements are deactivated if failure criterion is met, elements remain for post processing only
    - 2: associated elements are deactivated if failure criterion is met, elements are removed from post processing

CONNECTOR ELEMENT – BUSH

- **Fuse option (cont.)**
  - DIR, fuse direction
    - 0: any direction can fuse, same as 123456
    - 1-6: directions which can fuse (like SPC input)
  - OPTION, failure mode
    - ULTLD for load
    - RELDIS for displacement
  - UPPER/LOWER, failure limits
    - depending on OPTION a load or displacement limit
  - FRATE, fraction for stiffness scaling to avoid sudden zero (default=1.-5)

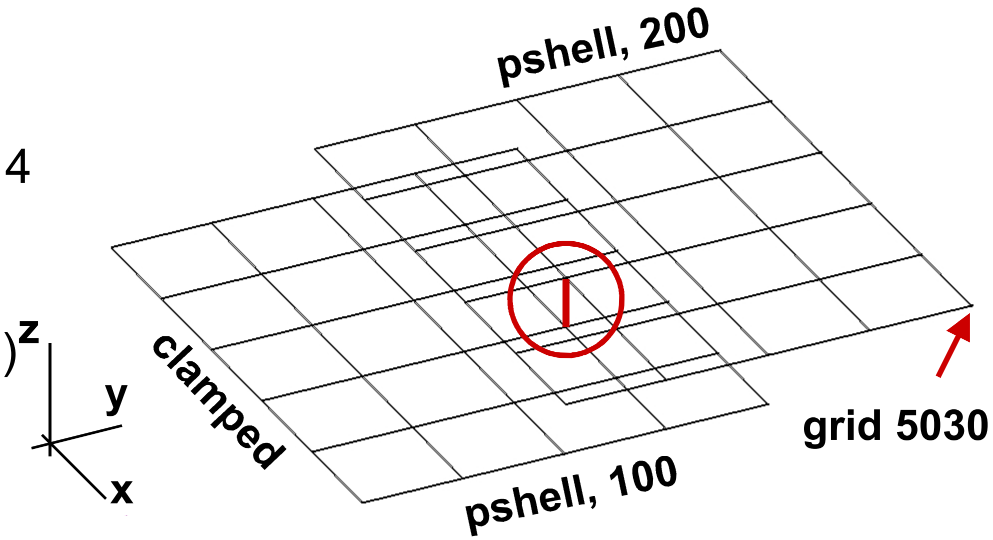
CONNECTOR ELEMENTS – WELD / FAST



- **WELDs and FASTs are elements designed for the connection of shells**
  - Applications are spot welds and fasteners
  - WELD is a BEAM element
  - FAST is a BUSH element
  - Both elements are connected to the shells by internally created RBE3 elements, i.e. do not require a node to node connection
  - Allows that the mesh does not have to have a node corresponding to the weld location on the components to be fastened
  - Both elements can undergo large rotations
  - The WELD element can have all nonlinear materials allowed for BEAM(PBEAM) elements

CONNECTOR ELEMENTS – WELD / FAST

- **Example for the WELD Element**
  - Weld element 1 is connecting 4 elements on each side in this case
  - Pressure on upper plate (1.+3)



2.5x2.0x0.1
E=3E7
ν=0.3

```
BEGIN BULK
PARAM   LGDISP  1
CWELD   1       11      100     PARTPAT
+       100     200
PWELD   11      10      0.8
$
PSHELL  100     10      .1      10      1.              .833333
CQUAD4  1000    100     1000    1001    2001    2000    0.      0.
.
PSHELL  200     10      .1      10      1.              .833333
CQUAD4  4005    200     5006    5007    5013    5012    0.      0.
.
MAT1    10      3.+7    1.153+7 0.3     .0074
```

connector diameter

CONNECTOR ELEMENTS – WELD / FAST

- **Results for WELD Example**

```
0                                                                                                      SUBCASE 1            STEP 1
     LOAD STEP =   1.00000E+00
                                                 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
          5030      G      1.445504E-02  -1.584998E+00  -2.640457E+00  -1.381116E+00  -1.278003E-01   9.589864E-02
```

without GNL WELD (MD R2.1) this value would be -2.44

```
0                                                                                                      SUBCASE 1            STEP 1
     LOAD STEP =   1.00000E+00
                                     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 )
                              STAT DIST/   - BENDING MOMENTS -             - WEB  SHEARS -           AXIAL          TOTAL          WARPING
   ELEMENT-ID   GRID   LENGTH     PLANE 1        PLANE 2        PLANE 1       PLANE 2         FORCE          TORQUE         TORQUE
0           1
            101000001   0.000   -8.839650E+02  2.412787E+03   3.064666E+02 -3.020579E+02  -4.963424E+03  -6.540462E+02   0.0
            101000002   1.000   -9.605817E+02  2.488301E+03   3.064666E+02 -3.020579E+02  -4.963424E+03  -6.540462E+02   0.0
```

these grid points are the grids A and B of an internally created BEAM for the WELD

This value meets top plate area (2.5x2.0) times pressure (1E3). Without GNL WELD it would be far off (-1502.2).

CONNECTOR ELEMENTS – WELD / FAST

- **Results for WELD Example (Cont.)**

```
0                                                                                                     SUBCASE 1           STEP 1
      LOAD STEP =   1.00000E+00
                                          S T R A I N S    I N   W E L D   E L E M E N T S       ( C W E L D )
                     STAT DIST/
    ELEMENT-ID   GRID   LENGTH    SXC           SXD           SXE           SXF           S-MAX         S-MIN         M.S.-T   M.S.-C
0          1
           101000001    0.000    -5.445103E-04 -1.928071E-03 -1.137843E-04  1.269777E-03  1.269777E-03 -1.928071E-03
           101000002    1.000     2.348792E-04 -1.978735E-03 -8.931737E-04  1.320440E-03  1.320440E-03 -1.978735E-03


0                                                                                                     SUBCASE 1           STEP 1
      LOAD STEP =   1.00000E+00
                                         S T R E S S E S    I N   W E L D   E L E M E N T S       ( C W E L D )
                     STAT DIST/
    ELEMENT-ID   GRID   LENGTH    SXC           SXD           SXE           SXF           S-MAX         S-MIN         M.S.-T   M.S.-C
0          1
           101000001    0.000     7.711506E+03 -5.787528E+04 -2.746034E+04  3.812645E+04  3.812645E+04 -5.787528E+04
           101000002    1.000     9.235746E+03 -5.937759E+04 -2.898458E+04  3.962876E+04  3.962876E+04 -5.937759E+04
0                                                                                                     SUBCASE 1           STEP 1
      LOAD STEP =   1.00000E+00
                                                  G R I D   P O I N T   F O R C E   B A L A N C E
        ●
        ●
0       3002          2001    QUAD4          1.084012E+03   1.374482E+03  -8.756651E+02   4.998597E+02  -1.588867E+02   1.458691E+02
        3002          2002    QUAD4         -9.773187E+02   1.014899E+03  -1.177605E+02   4.968433E+02   1.249129E+02  -1.175040E+02
        3002          3001    QUAD4          1.340506E+03  -1.484548E+03   6.786337E+02  -4.930043E+02  -1.708182E+01   7.086414E+01
        3002          3002    QUAD4         -1.472552E+03  -7.252704E+02   3.445563E+02  -5.036438E+02   5.055989E+01  -9.921779E+01
        3002     100001004    RBE3           2.550272E+01  -1.790153E+02  -3.152349E+01   0.0            0.0            0.0
        3002                  *TOTALS*       1.497325E-01   5.466715E-01  -1.759053E+00   5.487696E-02  -4.957646E-01   1.150136E-02
```

this is one of 8 newly created grid points which not directly belong to WELD

NONLINEAR KINEMATIC ELEMENTS

- **Nonlinear Kinematic Elements are the rigid elements with Lagrange formulation used in geometric nonlinear analysis of large rotation**
  - These include RBAR, RBAR1, RJOINT, RBE1, RBE2, RBE3, RTRPLT, and RTRPL1
- **Type of elements that can be combined with the kinematic elements:**
  - CBEAM, CQUAD4, CQUADR, CTRIA3, and CTRIAR
  - CHEXA, CPENTA, CTETRA (4 and 10 nodes)
  - CBAR and CSHEAR, which are converted into CBEAM and CQUAD4 by the Bulk Data entry:
    - MDLPRM, BRTOBM, 1, SHRTOQ4, 1

NONLINEAR KINEMATIC ELEMENTS

- **Implemented in SOL 400 only. User fatal error issued if used in SOL 106 or 129**
  - Implemented in both nonlinear static and transient analyses
- **This type of element becomes linkage if any dof is released**
  - Appropriate constraints must be supplied, otherwise the structure model will be singular and the solution diverges

NONLINEAR KINEMATIC ELEMENTS

- **Benefits:**
  - The kinematic elements can be combined with shell, beam, and solid elements
  - Differential stiffness are computed to facilitate convergence of the solution
  - Appropriate scale factors for Lagrange multipliers and penalty functions are computed automatically
  - Allow thermal load to be used with the kinematic elements
  - Both force and GPFORCE outputs available for output

NONLINEAR KINEMATIC ELEMENTS

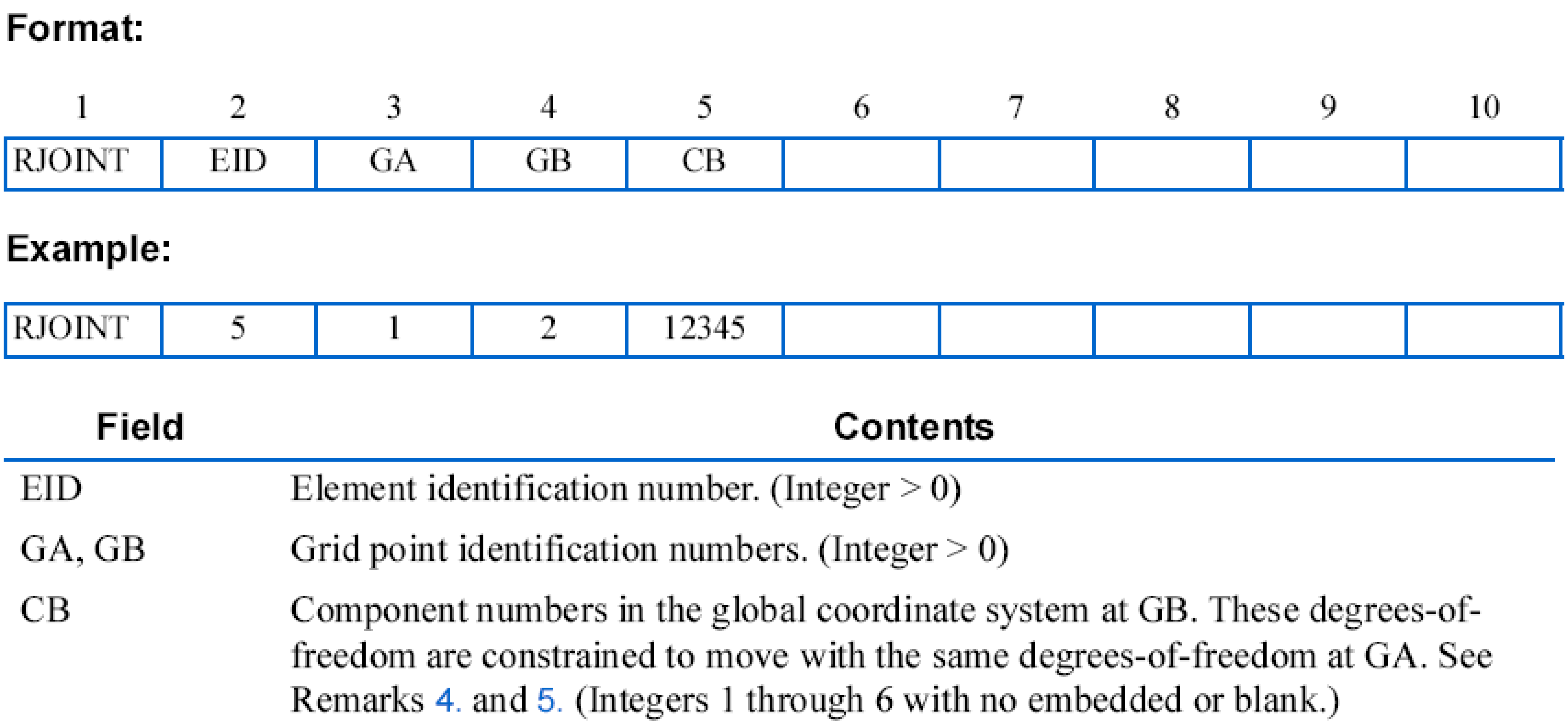
- **Thermal effects are computed if thermal load is requested**
  - ALPHA field on the Bulk Data entry must be supplied
- **Outputs:**
  - Element force is requested by the Case Control command MPCFO
  - The GPFORCE output is requested by the GPFO Case Control command

NONLINEAR KINEMATIC ELEMENTS

- **The scale factors for Lagrange multipliers and penalty functions are computed automatically based on:**
  - the geometry of the kinematic elements
  - the average magnitude of the stiffness matrix
- **PARAM, LMFACT and PENFN will overwrite the computed default values for the stiffness portion**
- **Case Control command RIGID=LINEAR can be used to request the corresponding linear rigid elements instead of the NL kinematic elements**

NONLINEAR KINEMATIC ELEMENTS

- **Example for RJOINT**



Format:

1	2	3	4	5	6	7	8	9	10
RJOINT	EID	GA	GB	CB					

Example:

RJOINT	5	1	2	12345					

Field	Contents
EID	Element identification number. (Integer > 0)
GA, GB	Grid point identification numbers. (Integer > 0)
CB	Component numbers in the global coordinate system at GB. These degrees-of-freedom are constrained to move with the same degrees-of-freedom at GA. See Remarks 4. and 5. (Integers 1 through 6 with no embedded or blank.)

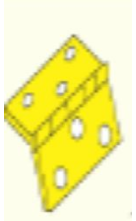
NONLINEAR KINEMATIC ELEMENTS

- **An RJOINT can be:**
  - Spherical joint - A spherical joint is a mechanical joint that rotates freely about all three axes. All rotational degrees of freedom released,

    i.e. CB = 123.

  - Hinge - A hinge is a mechanical joint that rotates freely about one axis of the local coordinate system. One rotational degree of freedom released,

    i.e., CB = 12356, 12346, or 12345.



  - Universal joint - A universal joint is a mechanical joint that rotates freely in two axes. Two rotational degrees for freedom released,

    i.e., CB = 1234, 1235, or 1236.

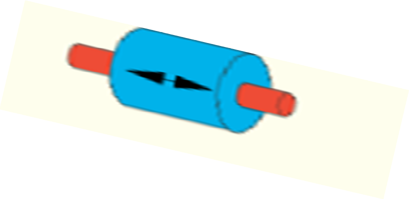
NONLINEAR KINEMATIC ELEMENTS

- **An RJOINT can be (cont.):**
  - Prismatic joint - A prismatic joint is a mechanical system with two blocks that are constrained to have the same rotations, but translate relatively with each other along a local axis,

    i.e., CB = 23456, 13456, 12456.

  - Cylindrical joint - A cylindrical joint is a mechanical system that allows two grid points have relative translation along a moving axis and relative rotation about the same axis,

    i.e. CB = 2356, 1346, 1245.



GENERAL GUIDELINES FOR SELECTING ELEMENTS

- **Choose quads over tris**
- **Choose bricks over wedges**
- **Avoid low order tetrahedral elements wherever possible**
  - The element exhibits slow convergence with mesh refinement
  - This element provides accurate results only with very fine meshing
  - This element is recommended only for filling in regions of low stress gradient in meshes of Hex8 elements, when the geometry precludes the use of Hex8 elements throughout the model
- **For tetrahedral element meshes the second-order element should be used**
- **Tet10s perform similar to hex8s in accuracy**
- **Full integration, first order elements in conjunction with the assumed strain procedure work well for most applications**
- **Use low order quads and hexes with assumed strain with reduced integration if the mesh is refined, of high quality and will not deform very badly**

GENERAL GUIDELINES FOR SELECTING ELEMENTS

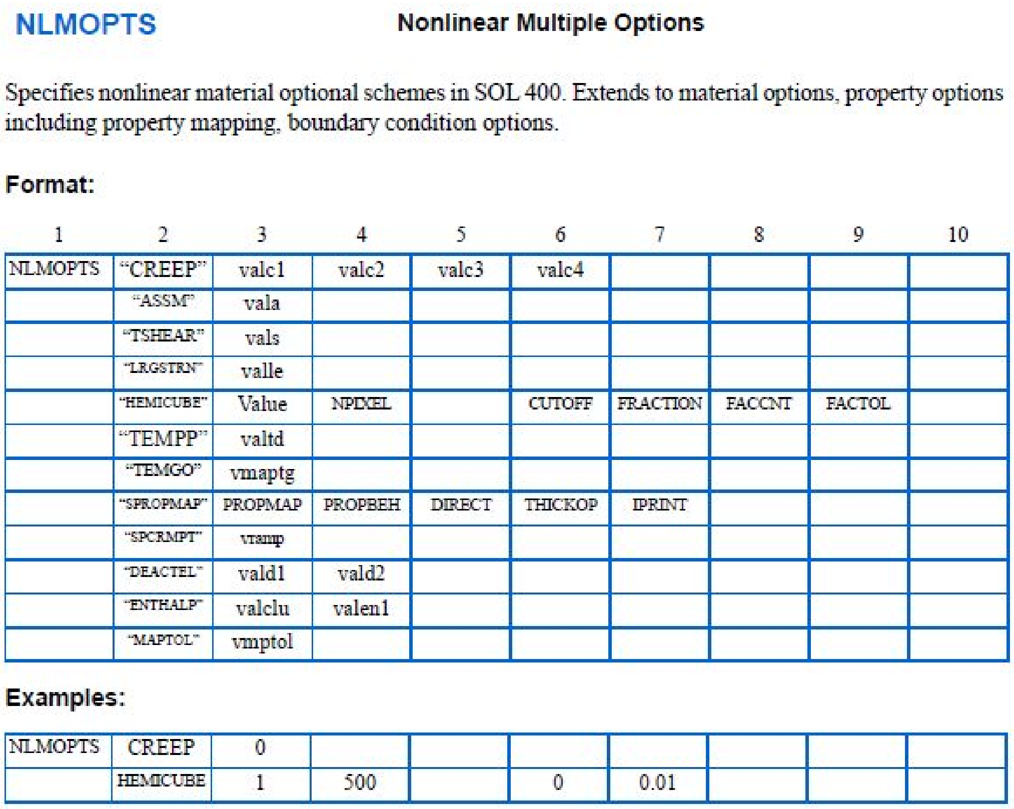
- **Use 2nd order reduced integration quads and solids if the mesh is coarse**
- **Don’t use 2nd order elements if there are gaps in the simulation – the sign reversal between corner and mid-side nodes will cause failure**
- **Contact fully supports lower and higher order elements**
- **During plastic deformation, metals exhibit incompressible behavior**
  - The incompressible behavior can lead to certain types of elements being over-constrained
  - This leads to an overly stiff behavior (volumetric locking)
  - Turn ON the “Constant Dilatation” option to correct for this

GENERAL GUIDELINES FOR SELECTING ELEMENTS

- **During hyper elastic deformation, rubbers exhibit incompressible behavior**
  - The incompressible behavior can lead to certain types of elements being over-constrained
  - This leads to an overly stiff behavior (volumetric locking)
  - Use the corresponding Hermann element to correct for this
- **Second order elements are susceptible to volumetric locking when modeling incompressible materials**
  - In general, avoid using them when modeling hyper-elasticity and plasticity
- **For large strain analyses, lower-order elements are recommended**
- **Second order elements should be used with caution above 20% strain**

ADDITIONAL RESULTS OUTPUT

- **NLMOPTS – Nonlinear Multiple Options**



NLMOPTS Nonlinear Multiple Options

Specifies nonlinear material optional schemes in SOL 400. Extends to material options, property options including property mapping, boundary condition options.

Format:

1	2	3	4	5	6	7	8	9	10
NLMOPTS	“CREEP”	valc1	valc2	valc3	valc4				
	“ASSM”	vala							
	“TSHEAR”	vals							
	“LRGSTRN”	valle							
	“HEMICUBE”	Value	NPIXEL		CUTOFF	FRACTION	FACCNT	FACTOL	
	“TEMPP”	valtd							
	“TEMGO”	vmaptg							
	“SPROPMAP”	PROPMAP	PROPBEH	DIRECT	THICKOP	IPRINT			
	“SPCRMPT”	vramp							
	“DEACTEL”	vald1	vald2						
	“ENTHALP”	valclu	valen1						
	“MAPTOL”	vmptol							

Examples:

NLMOPTS	CREEP	0							
	HEMICUBE	1	500		0	0.01			

ADDITIONAL RESULTS OUTPUT

- **NLMOPTS – Nonlinear Multiple Options**
  - NLMOPTS, SPROPMAP controls the conversion of element properties to the advanced formulations and can be used to dump the actual properties that are used. For example:

```
$ PROPMAP -- default = map to full integration elems
$ IPRINT  -- 1 = print generated elem props
$NLMOPTS, SPROPMAP, PROPMAP,,,, IPRINT
NLMOPTS, SPROPMAP,        ,,,, 1
```

  - Will dump out information that looks like this for each shell property element set:

```
PSHLN1  1        1       1        NO        ISH
        C3       DCTN    LDK      DCT       L
        C4       DCT     L        DCT       L
        C8       DCT     QRI      DCT       Q
        C6       MB      Q        MB        Q
```

  - So that you know exactly what element formulation is used. It does this for the other element types too (PBARN1**, etc).**

ADDITIONAL RESULTS OUTPUT

- **Extended NLSTRESS Command** [1)]
  which refers to an NLOUT Bulk Data entry

$$\text{NLSTRESS}\left[\,(\begin{bmatrix}\text{SORT1}\\\text{SORT2}\end{bmatrix},\begin{bmatrix}\text{PRINT, PUNCH}\\\text{PLOT}\end{bmatrix},\boxed{[\text{NLOUT}=\text{m}])}\,\right]=\left\{\begin{array}{c}\text{ALL}\\\text{n}\\\text{NONE}\end{array}\right\}$$

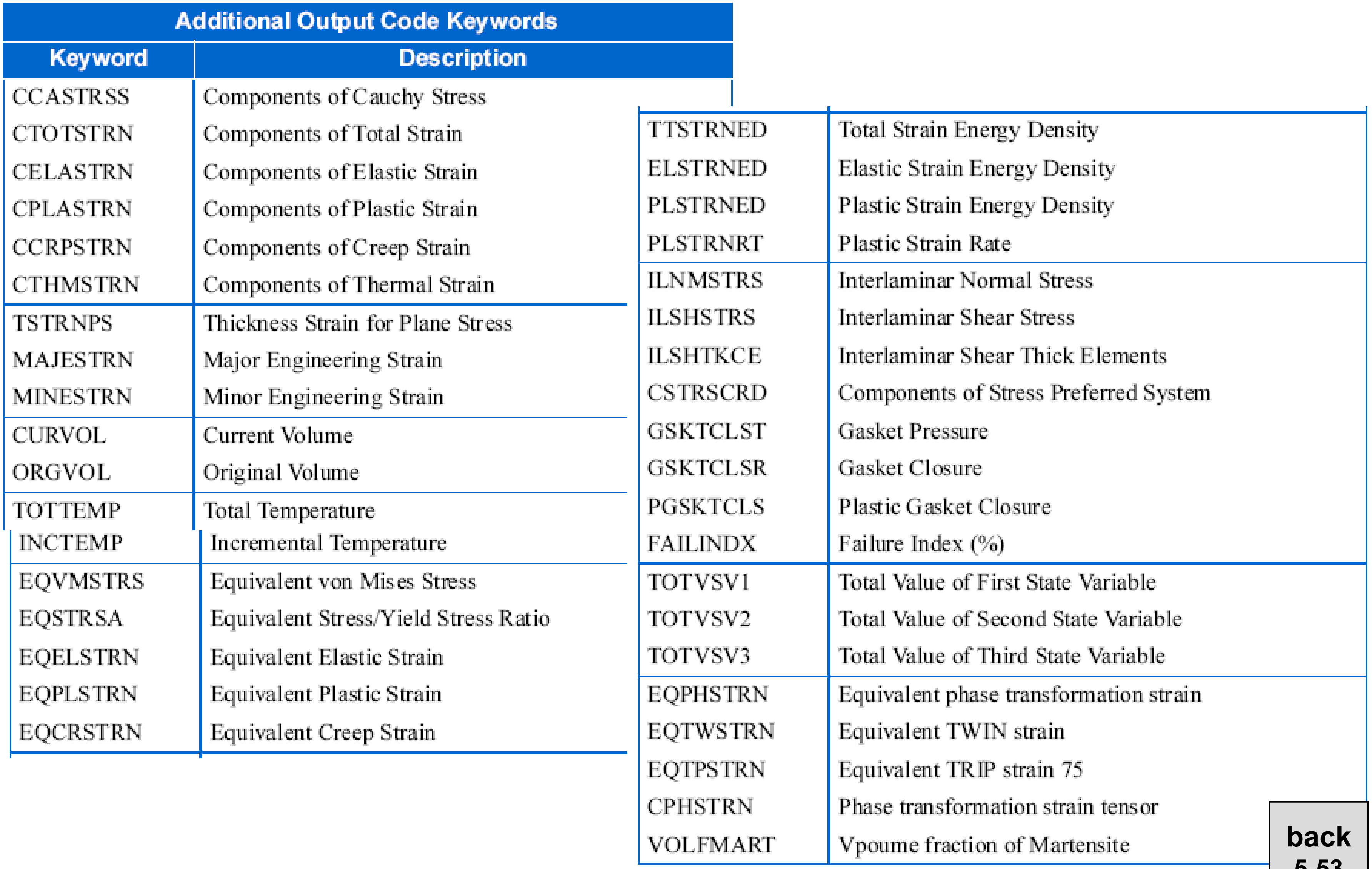
with the following results quantities

1	2	3	4	5	6	7	8	9	10
NLOUT	ID	AIO1	AIO2	AIO3	AIO4	AIO5	AIO6	AIO7	
	AIO8	AIO9	etc.						

---

1) applies only to the nonlinear property extension elements (PSHLN1, PSLDN1, …)

ADDITIONAL RESULTS OUTPUT



Additional Output Code Keywords

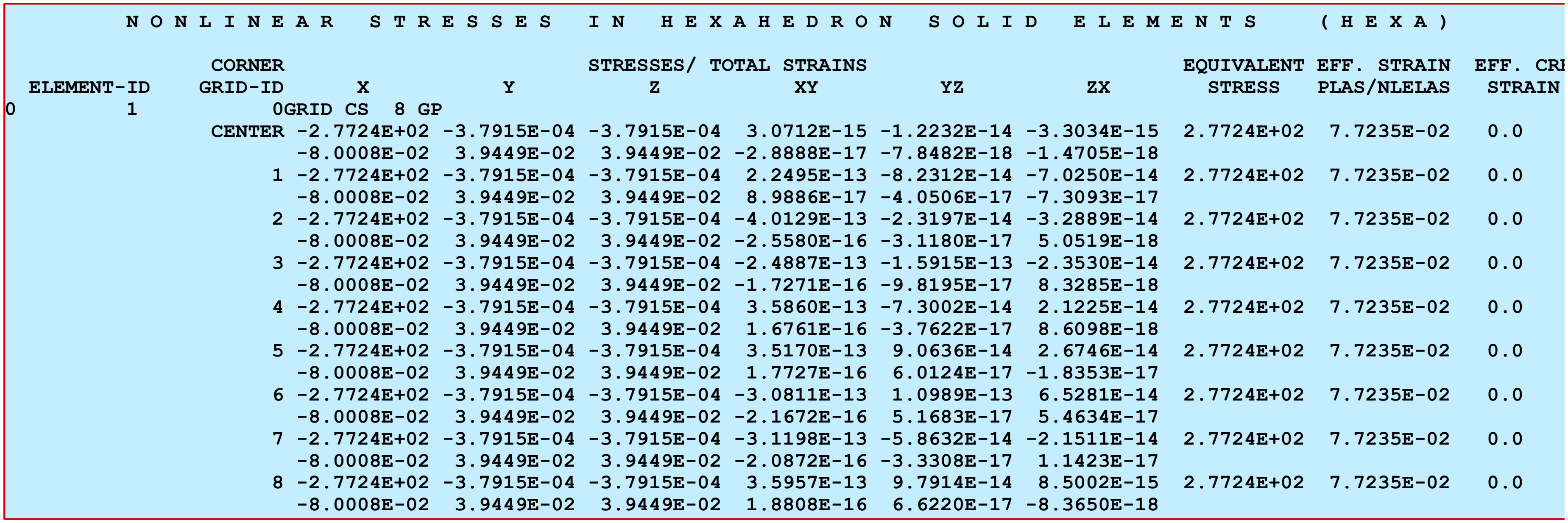
Keyword	Description
CCASTRSS	Components of Cauchy Stress
CTOTSTRN	Components of Total Strain
CELASTRN	Components of Elastic Strain
CPLASTRN	Components of Plastic Strain
CCRPSTRN	Components of Creep Strain
CTHMSTRN	Components of Thermal Strain
TSTRNPS	Thickness Strain for Plane Stress
MAJESTRN	Major Engineering Strain
MINESTRN	Minor Engineering Strain
CURVOL	Current Volume
ORGVOL	Original Volume
TOTTEMP	Total Temperature
INCTEMP	Incremental Temperature
EQVMSTRS	Equivalent von Mises Stress
EQSTRSA	Equivalent Stress/Yield Stress Ratio
EQELSTRN	Equivalent Elastic Strain
EQPLSTRN	Equivalent Plastic Strain
EQCRSTRN	Equivalent Creep Strain
TTSTRNED	Total Strain Energy Density
ELSTRNED	Elastic Strain Energy Density
PLSTRNED	Plastic Strain Energy Density
PLSTRNRT	Plastic Strain Rate
ILNMSTRS	Interlaminar Normal Stress
ILSHSTRS	Interlaminar Shear Stress
ILSHTKCE	Interlaminar Shear Thick Elements
CSTRSCRD	Components of Stress Preferred System
GSKTCLST	Gasket Pressure
GSKTCLSR	Gasket Closure
PGSKTCLS	Plastic Gasket Closure
FAILINDX	Failure Index (%)
TOTVSV1	Total Value of First State Variable
TOTVSV2	Total Value of Second State Variable
TOTVSV3	Total Value of Third State Variable
EQPHSTRN	Equivalent phase transformation strain
EQTWSTRN	Equivalent TWIN strain
EQTPSTRN	Equivalent TRIP strain 75
CPHSTRN	Phase transformation strain tensor
VOLFMART	Vpoume fraction of Martensite

back
5-53

ADDITIONAL RESULTS OUTPUT

- **Enhanced Input for 3D Solids (cont.)**
  - Results for PSLDN1

NLSTRESS = ALL



```
           N O N L I N E A R   S T R E S S E S   I N   H E X A H E D R O N   S O L I D   E L E M E N T S    ( H E X A )

                  CORNER                        STRESSES/ TOTAL STRAINS                          EQUIVALENT EFF. STRAIN  EFF. CRE
  ELEMENT-ID    GRID-ID      X           Y           Z           XY          YZ          ZX        STRESS    PLAS/NLELAS   STRAIN
0         1          0GRID CS  8 GP
                  CENTER -2.7724E+02 -3.7915E-04 -3.7915E-04  3.0712E-15 -1.2232E-14 -3.3034E-15  2.7724E+02  7.7235E-02   0.0
                         -8.0008E-02  3.9449E-02  3.9449E-02 -2.8888E-17 -7.8482E-18 -1.4705E-18
                       1 -2.7724E+02 -3.7915E-04 -3.7915E-04  2.2495E-13 -8.2312E-14 -7.0250E-14  2.7724E+02  7.7235E-02   0.0
                         -8.0008E-02  3.9449E-02  3.9449E-02  8.9886E-17 -4.0506E-17 -7.3093E-17
                       2 -2.7724E+02 -3.7915E-04 -3.7915E-04 -4.0129E-13 -2.3197E-14 -3.2889E-14  2.7724E+02  7.7235E-02   0.0
                         -8.0008E-02  3.9449E-02  3.9449E-02 -2.5580E-16 -3.1180E-17  5.0519E-18
                       3 -2.7724E+02 -3.7915E-04 -3.7915E-04 -2.4887E-13 -1.5915E-13 -2.3530E-14  2.7724E+02  7.7235E-02   0.0
                         -8.0008E-02  3.9449E-02  3.9449E-02 -1.7271E-16 -9.8195E-17  8.3285E-18
                       4 -2.7724E+02 -3.7915E-04 -3.7915E-04  3.5860E-13 -7.3002E-14  2.1225E-14  2.7724E+02  7.7235E-02   0.0
                         -8.0008E-02  3.9449E-02  3.9449E-02  1.6761E-16 -3.7622E-17  8.6098E-18
                       5 -2.7724E+02 -3.7915E-04 -3.7915E-04  3.5170E-13  9.0636E-14  2.6746E-14  2.7724E+02  7.7235E-02   0.0
                         -8.0008E-02  3.9449E-02  3.9449E-02  1.7727E-16  6.0124E-17 -1.8353E-17
                       6 -2.7724E+02 -3.7915E-04 -3.7915E-04 -3.0811E-13  1.0989E-13  6.5281E-14  2.7724E+02  7.7235E-02   0.0
                         -8.0008E-02  3.9449E-02  3.9449E-02 -2.1672E-16  5.1683E-17  5.4634E-17
                       7 -2.7724E+02 -3.7915E-04 -3.7915E-04 -3.1198E-13 -5.8632E-14 -2.1511E-14  2.7724E+02  7.7235E-02   0.0
                         -8.0008E-02  3.9449E-02  3.9449E-02 -2.0872E-16 -3.3308E-17  1.1423E-17
                       8 -2.7724E+02 -3.7915E-04 -3.7915E-04  3.5957E-13  9.7914E-14  8.5002E-15  2.7724E+02  7.7235E-02   0.0
                         -8.0008E-02  3.9449E-02  3.9449E-02  1.8808E-16  6.6220E-17 -8.3650E-18
```

ADDITIONAL RESULTS OUTPUT

- **Input for 2D Composites**
  - Additional Results Output

```
NLSTRESS(NLOUT=1)  = ALL
.
BEGIN BULK
NLOUT    1         CTOTSTRN
```

```
LOAD STEP =  1.00000E+00

        TENSOR OUTPUT QUANTITIES
 element  output    ply-id     CID   gaus    tensor components
 id       quantity                    point   X           Y           Z           XY          YZ          XZ
       1  CTOTSTRN       1       0       1  -0.461E-01   0.443E-01   0.000E+00   0.342E-16   0.000E+00   0.000E+00
                                         2  -0.461E-01   0.443E-01   0.000E+00  -0.490E-17   0.000E+00   0.000E+00
                         2       0       1  -0.461E-01   0.443E-01   0.000E+00   0.459E-17   0.000E+00   0.000E+00
                                         2  -0.461E-01   0.443E-01   0.000E+00  -0.310E-16   0.000E+00   0.000E+00
```

ADDITIONAL RESULTS OUTPUT

- **Input for 3D Composites**
  - Example for PCOMPLS – solid shell HEX8 (cont.)
  - total strains (NLOUT request)

```
                         N O N L I N E A R   O U T P U T   I N   H E X A H E D R O N   E L E M E N T S     ( H E X A )

       TENSOR OUTPUT QUANTITIES
element  output    ply-id     CID    gaus    tensor components
id       quantity                    point   X           Y           Z           XY          YZ          XZ
      1  CTOTSTRN        1       0       1  -0.800E-01   0.394E-01   0.394E-01   0.774E-17  -0.105E-16  -0.595E-15
                         2       0       1  -0.800E-01   0.394E-01   0.394E-01   0.393E-18  -0.157E-16  -0.590E-15
```

  - interlaminar shear stresses

```
 I N T E R L A M I N A R    S T R E S S E S    F O R    L A Y E R E D    C O M P O S I T E    E L E M E N T S

  ELEMENT                   INTEG.       N O R M A L   S T R E S S           S H E A R   S T R E S S             BOND
       ID      PLY ID  POINT ID        1          2          3          1          2          3            INDEX
        1           1         1   6.384E-19  3.237E-19 -3.792E-04 -1.006E-13 -1.730E-14  1.841E-28    0.000E+00
                    2         1   0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00    0.000E+00
```

ADDITIONAL RESULTS OUTPUT

- **Input for 3D Composites**
  - Example for PCOMPLS – solid HEX8 (cont.)
  - total strains (NLOUT request)

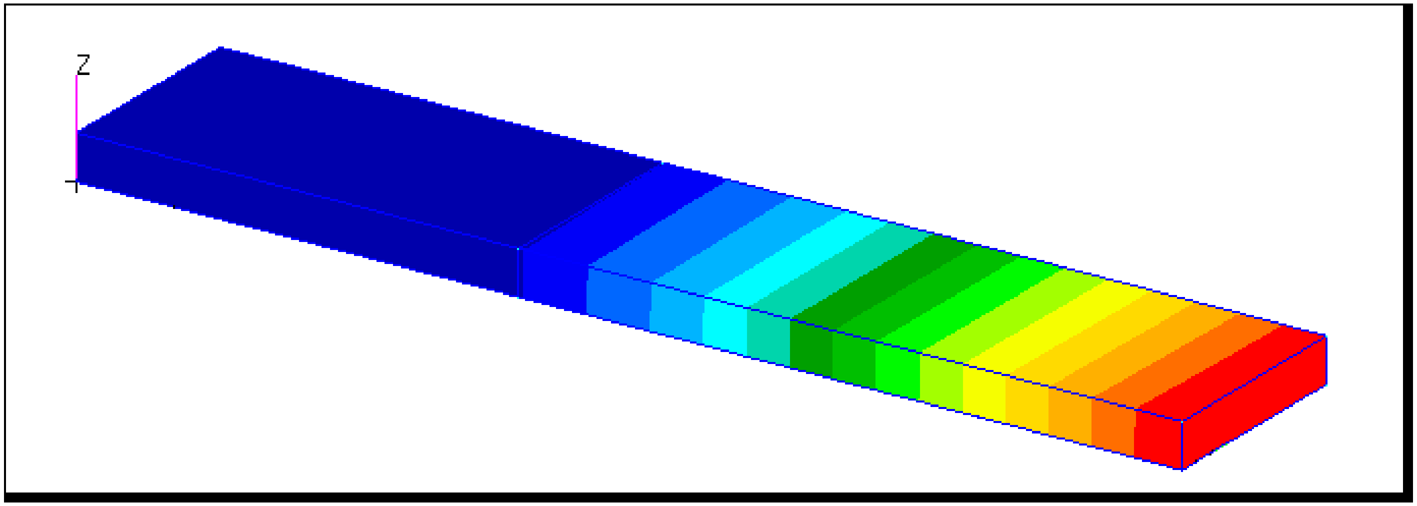
```
       TENSOR OUTPUT QUANTITIES
element   output     ply-id     CID    gaus    tensor components
id        quantity                     point    X            Y           Z            XY           YZ           XZ
      1   CTOTSTRN        1        0       1  -0.800E-01   0.394E-01   0.394E-01  -0.314E-16  -0.322E-16   0.161E-16
                                           2  -0.800E-01   0.394E-01   0.394E-01  -0.147E-17   0.118E-15  -0.636E-16
                                           3  -0.800E-01   0.394E-01   0.394E-01   0.212E-16  -0.514E-16  -0.621E-17
                                           4  -0.800E-01   0.394E-01   0.394E-01   0.474E-16  -0.283E-16  -0.487E-16
                          2        0       1  -0.800E-01   0.394E-01   0.394E-01   0.428E-16   0.278E-16   0.496E-16
                                           2  -0.800E-01   0.394E-01   0.394E-01   0.102E-15   0.127E-15  -0.469E-16
                                           3  -0.800E-01   0.394E-01   0.394E-01  -0.222E-16   0.688E-17  -0.324E-16
                                           4  -0.800E-01   0.394E-01   0.394E-01   0.370E-16  -0.225E-16  -0.730E-16
```

  - interlaminar shear stresses

```
 I N T E R L A M I N A R    S T R E S S E S    F O R    L A Y E R E D    C O M P O S I T E    E L E M E N T S

  ELEMENT              INTEG.       N O R M A L    S T R E S S           S H E A R    S T R E S S            BOND
       ID     PLY ID   POINT ID         1           2           3            1           2           3            INDEX
        1          1          1   -5.436E-28  9.836E-20 -3.792E-04   8.703E-14  2.748E-15  0.000E+00   0.000E+00
                              2   -5.436E-28  6.475E-20 -3.792E-04  -1.932E-14  6.021E-14  0.000E+00   0.000E+00
                              3   -7.296E-20  9.836E-20 -3.792E-04  -8.528E-14 -3.396E-14  0.000E+00   0.000E+00
                              4   -7.296E-20  6.475E-20 -3.792E-04  -1.171E-13  5.637E-15  0.000E+00   0.000E+00
                   2          1    0.000E+00  0.000E+00  0.000E+00   0.000E+00  0.000E+00  0.000E+00   0.000E+00
                              2    0.000E+00  0.000E+00  0.000E+00   0.000E+00  0.000E+00  0.000E+00   0.000E+00
                              3    0.000E+00  0.000E+00  0.000E+00   0.000E+00  0.000E+00  0.000E+00   0.000E+00
                              4    0.000E+00  0.000E+00  0.000E+00   0.000E+00  0.000E+00  0.000E+00   0.000E+00
```

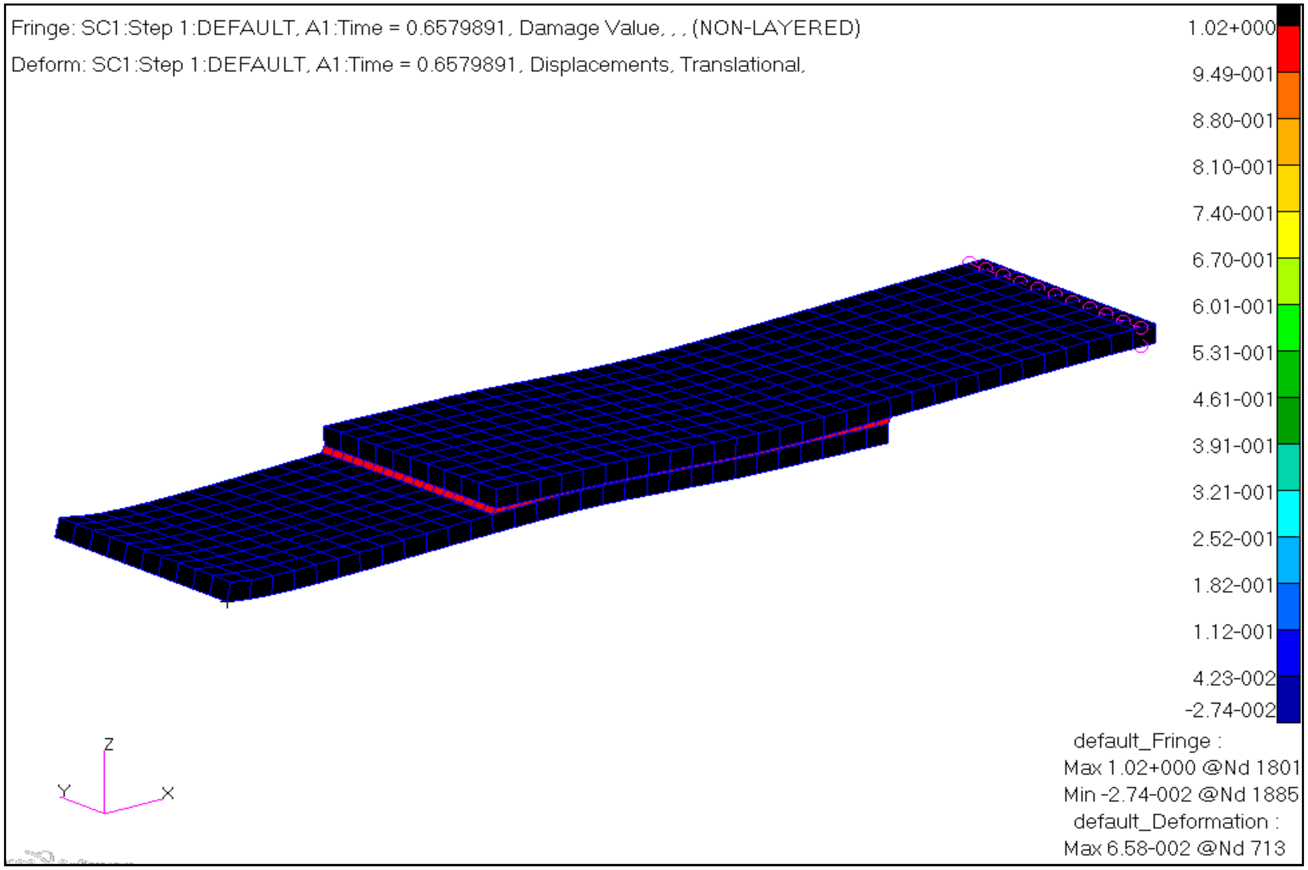
EXERCISE

- **Perform Workshop 7: Solid Shell Composites Modeling**



EXERCISE

- **Perform Workshop 8: Delamination of a Composite Solid Shell Beam**



Fringe: SC1:Step 1:DEFAULT, A1:Time = 0.6579891, Damage Value, , , (NON-LAYERED)

Deform: SC1:Step 1:DEFAULT, A1:Time = 0.6579891, Displacements, Translational,

default_Fringe :
Max 1.02+000 @Nd 1801
Min -2.74-002 @Nd 1885
default_Deformation :
Max 6.58-002 @Nd 713

SECTION 7

ADVANCED TOPICS

OVERVIEW

- Analysis Chaining
- SPC and MPC Changes
- Restart

ANALYSIS CHAINING

- **In SOL 400, all SUBCASEs are independent from each other**
  - Like in SOL 101, many independent subcases can be performed in one single run
  - To change back to the old SOL 106/129 behavior, use system cell 366 (STPFLG)=1
- **For analysis chaining, STEPs have to be defined**
  - SOL400 allows complete flexibility in allowing several static, dynamic and linear perturbation Steps to be performed in a single Subcase. This is called analysis chaining.
  - A Step is any convenient phase of the history – a nonlinear static loading phase, a thermal transient, a creep period, a nonlinear dynamic transient, etc.
  - In each “Step”, the analysis type to be performed is chosen as one of the “ANALYSIS” commands shown in the next slide.
  - The sequence of Steps in a Subcase should be chosen meaningfully.

GUIDELINES FOR ANALYSIS CHAINING

- For nonlinear structures, an ANALYSIS = NLSTAT must come before an ANALYSIS = NLTRAN.
- All coupled multi-physics steps have to come before the single-physics steps.
- Single-physics steps can follow the multi-physics steps.
- All linear perturbation steps need to be at the end after definition of all possible coupled multi physics steps and single-physics steps.
- The NLIC Case Control Command used for Nonlinear Initial Condition should be referenced for further requirements between the analysis types allowed in SOL 400.

STEP TIME

- Nonlinear analysis step the loads must be defined as total values.
- Each step also has its own step time, which begins at zero in each step.
- In a dynamic analysis, “step time” corresponds to that physical time.
- Otherwise, step time is any convenient time scale, typically 0.0-1.0, for the step.

ANALYSIS CHAINING

- **For the ANALYSIS command the following disciplines can be used**

NLSTATIC	Nonlinear static analysis
NLTRAN	Nonlinear transient analysis
STATICS	Linear static analysis
BUCK	Linear buckling
MODES	Normal modes analysis
MFREQ	Modal frequency response
DFREQ	Direct frequency response
MTRAN	Modal transient response
DCEIG	Direct complex eigenvalue analysis
MCEIG	Modal complex eigenvalue analysis
BSQUEAL	Break squeal analysis
HSTAT	Steady state heat transfer analysis
HTRAN	Transient heat transfer analysis

ANALYSIS CHAINING

- **Scenario 1 – mixing statics and transient**

```
SUBCASE 1
   STEP 1
           LOAD = 1
           NLPARM = 110
   STEP 2
           ANALYSIS = NLTRAN
           DLOAD = 3
           TSTEPNL = 130
SUBCASE 2
   STEP 1
           ANALYSIS = NLSTAT           $ This line can be omitted
           LOAD = 10
           NLPARM = 110
   STEP 2
           NLIC SUBCASE 2 STEP 1 LOADFAC 0.5
           ANALYSIS = NLTRAN
           DLOAD = 30
           TSTEPNL = 130
```

ANALYSIS CHAINING

- **Scenario 2 – normal modes perturbation**

```
SUBCASE 3
  STEP 1
          ANALYSIS = NLSTAT           $ This line can be omitted
          LOAD = 10
          NLPARM = 110
  STEP 2
          NLIC STEP 1 LOADFAC 0.25
          ANALYSIS = MODES
          METHOD = 1
  STEP 3
          NLIC STEP 1 LOADFAC 0.75
          ANALYSIS = MODES
          METHOD = 2
```

ANALYSIS CHAINING

- **Scenario 3 – brake squeal analysis**

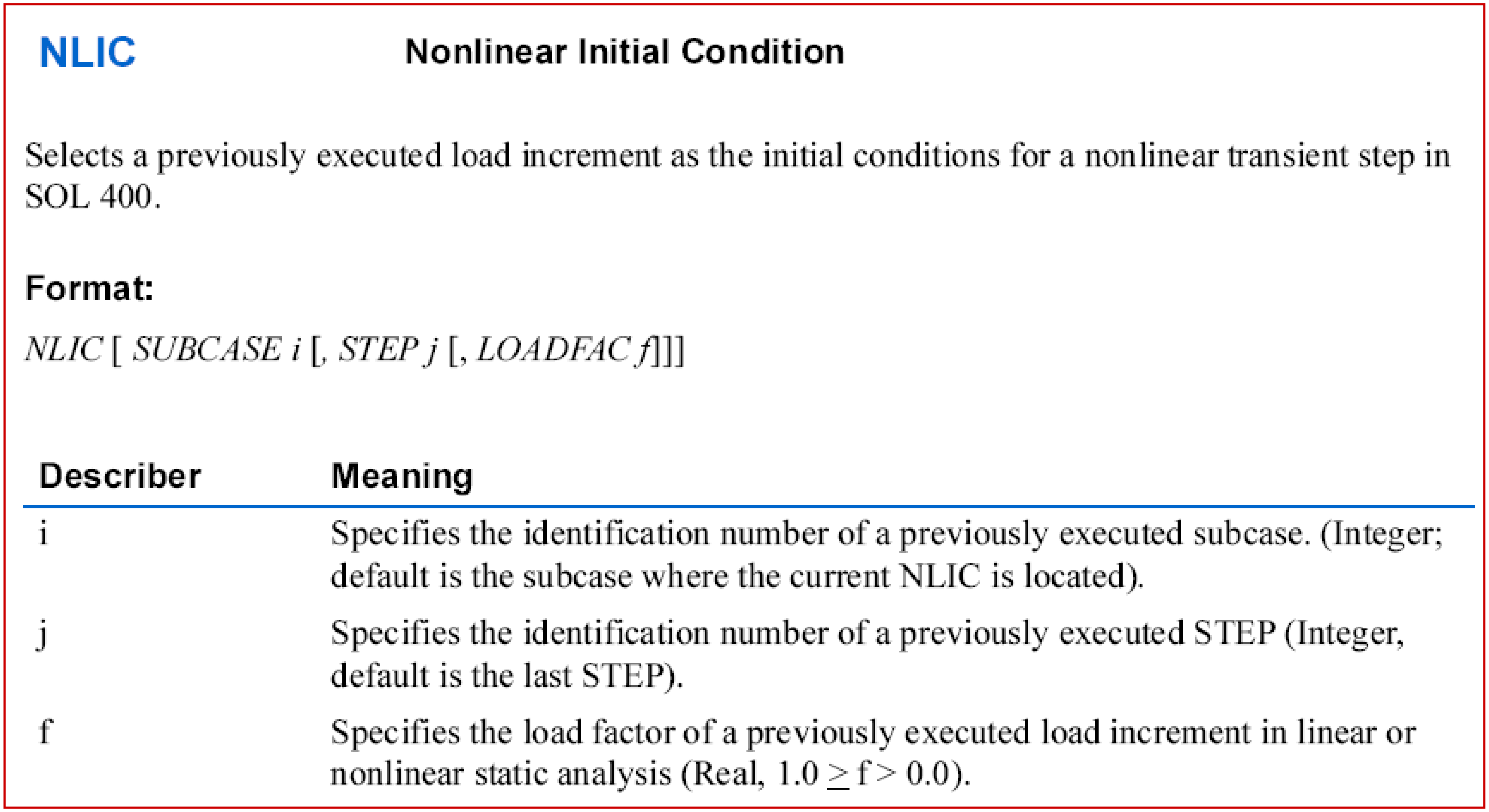
```
SUBCASE 4
  STEP 1
          LABEL = Nonlinear Static Step
          NLPARM = 3
          BCONTACT = 1
          SPC = 2
          LOAD = 4
  STEP 2
          LABEL = Modal Brake Squeal with NLIC at 0.5
          ANALYSIS = MCEIG
          BSQUEAL = 989
          NLIC STEP 1 LOADFAC 0.5
          SPC = 2
          CMETHOD = 1
          METHOD = 2
          AUTOSPC(noprint)= yes
          RESVEC = NO
```

ANALYSIS CHAINING

- **Scenario 4 – mixing structural and thermal**

```
SUBCASE 5
   STEP 1
          ANALYSIS = HSTAT             $ static heat transfer
          NLPARM = 1
          SPC = 1
          LOAD = 2
          THERMAL = ALL
          FLUX = ALL
          TSTRU = 200
   STEP 2
          ANALYSIS = NLSTAT            $ nonlinear statics
          NLPARM = 3
          SPC = 5
          TEMP(load)= 200
          LOAD = 13
```

ANALYSIS CHAINING



NLIC **Nonlinear Initial Condition**

Selects a previously executed load increment as the initial conditions for a nonlinear transient step in SOL 400.

Format:

NLIC [*SUBCASE i* [, *STEP j* [, *LOADFAC f*]]]

Describer	Meaning
i	Specifies the identification number of a previously executed subcase. (Integer; default is the subcase where the current NLIC is located).
j	Specifies the identification number of a previously executed STEP (Integer, default is the last STEP).
f	Specifies the load factor of a previously executed load increment in linear or nonlinear static analysis (Real, $1.0 \geq f > 0.0$).

SPC AND MPC CHANGES

SPC AND MPC CHANGES

- **The SPCs and MPCs are allowed to change from one step to the next for both static and transient analysis**

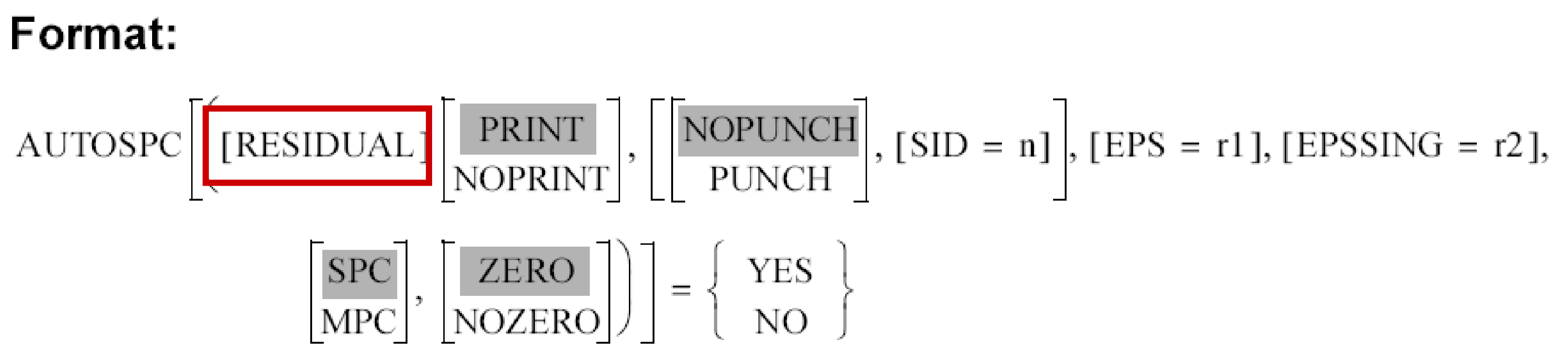
```
   .
  NLPARM = 1
STEP 1
  LOAD = 1
  SPC    = 1  $ grid 1 fixed, grid 2 free to move
STEP 2
  LOAD = 2
  SPC    = 2  $ grid 1 fixed, grid 2 in y is frozen at the location
$             it was at the end of step 1
STEP 3
  ANALYSIS = NLTRAN
  TSTEPNL = 3
  DLOAD = 3
  SPC    = 1  $ disp of grid 2 in y is free again
BEGIN BULK  $ and starts from its current value.
.
SPC1,1,123456,1  $ no SPC requested on grid 2.
SPC1,2,123456,1  $ same as original constraint for grid 1,
SPC1,2,2,2       $ extra SPC is requested on y dof of grid 2
```

SPC AND MPC CHANGES

- **SPC1 is used when a displacement is to be frozen (see page before)**
- **Before MD R3, for SPC's, only absolute boundary changes were allowed using SPCD or SPC**
  - For example, when the displacement is 0.1 at the end of step 1 it will get the value of SPCD or SPC in step 2
- **Starting with MD R3 incremental boundary changes can be done using SPCR (not NLTRAN)**
  - For example, if the displacement is 0.1 at the end of step 1, and there is a 0.3 on an SPCR called in step 2, the actual position will be 0.4
- **MPC changes are only incremental**
  - the relative deformation described with an MPC which is reached at the beginning of a step will be frozen
    - MPC activated in STEP 2: the MPC dof move together in STEP 2
    - STEP 1, stretch/preload a rubber band. STEP 2 use MPC to attach to structure

AUTOMATIC SPC AND MPC

- **AUTOSPC is available for nonlinear (MD R3+)**



Format:

$$\text{AUTOSPC}\left[\left(\boxed{[\text{RESIDUAL}]}\begin{bmatrix}\text{PRINT}\\ \text{NOPRINT}\end{bmatrix}, \left[\begin{bmatrix}\text{NOPUNCH}\\ \text{PUNCH}\end{bmatrix}, [\text{SID} = \text{n}]\right], [\text{EPS} = \text{r1}], [\text{EPSSING} = \text{r2}], \begin{bmatrix}\text{SPC}\\ \text{MPC}\end{bmatrix}, \begin{bmatrix}\text{ZERO}\\ \text{NOZERO}\end{bmatrix}\right)\right] = \begin{Bmatrix}\text{YES}\\ \text{NO}\end{Bmatrix}$$

- **RESIDUAL stands for the residual structure**
  - normally only superelements are effected
  - AUTOSPC(RESIDUAL) can be used for ANALYSIS=
    - NLSTAT
    - STATIC
    - NLTRAN

AUTOMATIC SPC AND MPC

- **AUTOSPC will be checked for the beginning of a step only**
  - it can be put above or within the step
- **Example**

```
STEP 1
        AUTOSPC(RESIDUAL,SPC) = YES
        LOAD = 10
STEP 2
        AUTOSPC(RESIDUAL,MPC) = YES
        LOAD = 20
STEP 3
        LOAD = 30
```

  - the singularity is constrained by the SPC option in step 1, by the MPC option in step 2, and there is no singularity checking in step 3.

AUTOMATIC SPC AND MPC

- **Care should be taken when the directions of singular DOF's undergo large rotations, like K6ROT when set to zero**
- **Also, under large deformation, DOF that start out singular might develop stiffness… in this case SPC'ing those DOF would be inappropriate**

RESTART

RESTARTS IN SOL400

- **What is a restart?**
  - The ability to continue a previously run simulation
- **Why restart?**
  - Eliminate duplication of calculation – i.e. initial steps of different simulations may be the same so a single run of those common steps can be run and then restarts performed from the end of the common steps
  - Unconverged solutions – allow ‘tweaking’ of parameters to gain convergence
    - This may be as simple as adding additional steps to allow a converging, but unconverged, solution to reach convergence
    - Or as complicated as respecifying contact parameters, convergence criteria, boundary conditions, etc.
    - Additional output requests
- **How to restart?**
  - Restart files from the Initial runs must be retained in the working directory. (MASTER/DBALL)
  - Subsequent runs specify use of the restart file and continue from there

RESTARTS IN SOL400

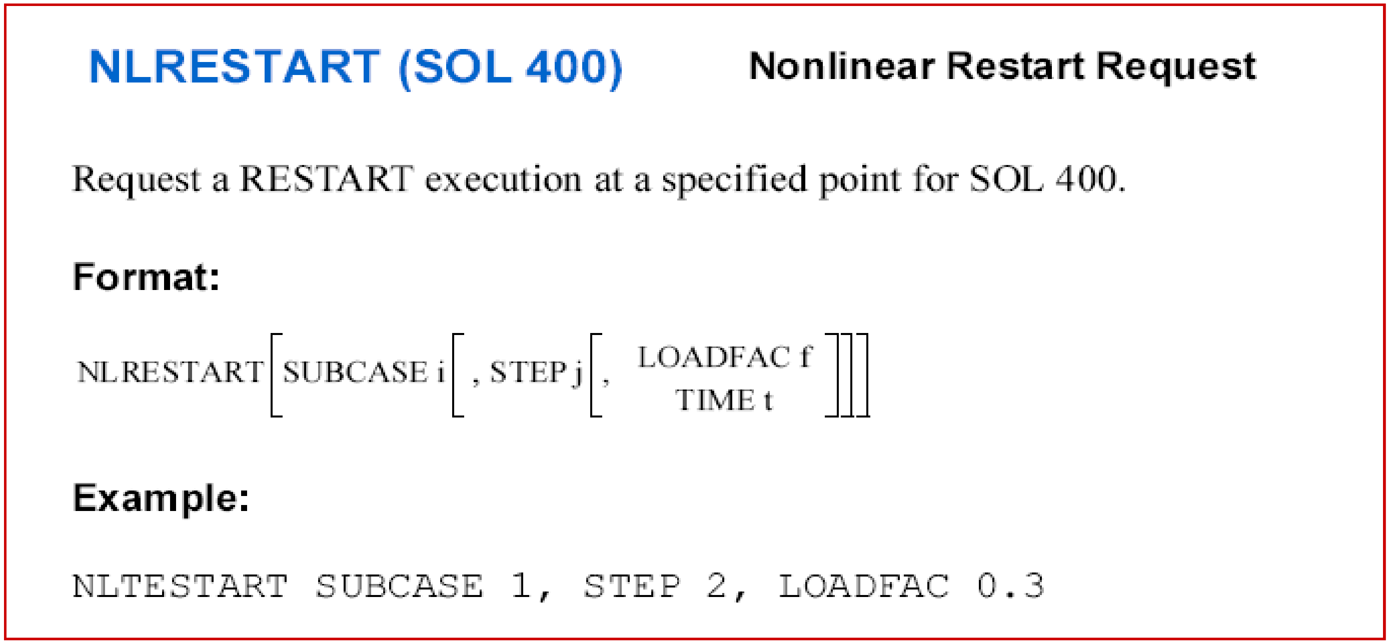
- **When utilizing the restart capabilities of SOL400 the user has the following options:**
  - Write restart data/files
    - The term ‘restart’ accurately implies that a simulation will be a continuation of a previous simulation hence the MASTER/DBALL file from previous simulation must exist.
    - These are automatically written in MASTER/DBALL file.
  - Read restart data/files
    - Adjust parameters/loading/convergence appropriately
    - Specify which restart point (restart file and increment, if appropriate) to start from
    - In the event of an unfinished loadcase the analyst must specify how to proceed (i.e. finish, modify parameters, remesh etc.)
  - Write and Read restart data
    - Read and Write capabilities can be activated simultaneously, no special consideration is necessary

RESTART

- **Restart via Case Control NLRESTART**

```
ASSIGN MASTER=‘cold.MASTER’
RESTART
SOL 400
CEND
NLRESTART SUBCASE 1, STEP 2, TIME 0.3    $ before any SUBCASE command
$
SUBCASE 1
        TSTEPNL = 10
        STEP 1              $ Case control of cold start with STEP 1 and 2
             LOAD = 10
        STEP 2              $ Restart begins at time 0.3 of this step
             TSTEPNL = 30
             LOAD = 20
        STEP 3
             LOAD = 30
BEGIN BULK
TSTEPNL,30, ,0.01,5
ENDDATA
```

RESTART ON QRG



NLRESTART (SOL 400) Nonlinear Restart Request

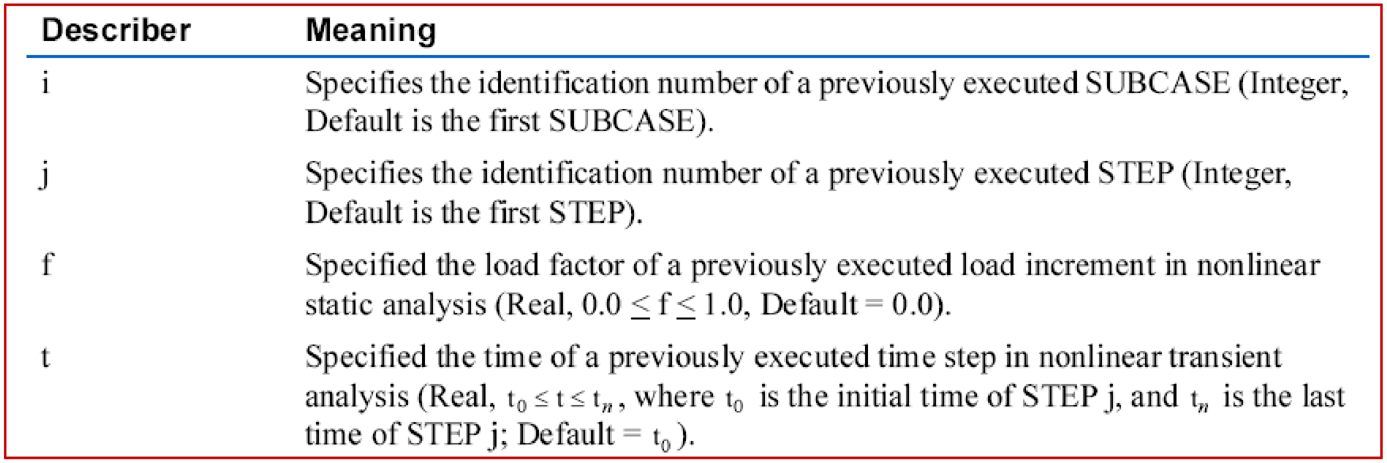
Request a RESTART execution at a specified point for SOL 400.

Format:

$$\text{NLRESTART}\left[\text{SUBCASE i}\left[\text{, STEP j}\left[,\ \begin{matrix}\text{LOADFAC f}\\ \text{TIME t}\end{matrix}\right]\right]\right]$$

Example:

```
NLTESTART SUBCASE 1, STEP 2, LOADFAC 0.3
```



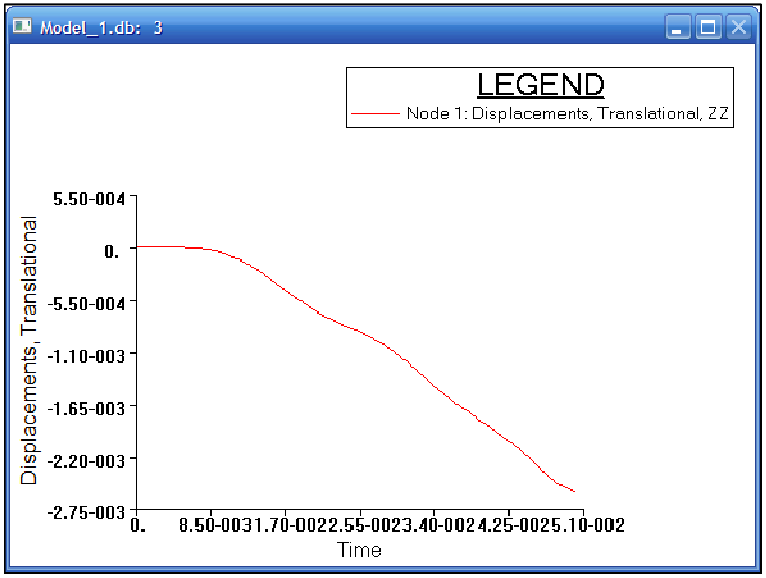
Describer	Meaning
i	Specifies the identification number of a previously executed SUBCASE (Integer, Default is the first SUBCASE).
j	Specifies the identification number of a previously executed STEP (Integer, Default is the first STEP).
f	Specified the load factor of a previously executed load increment in nonlinear static analysis (Real, $0.0 \leq f \leq 1.0$, Default = 0.0).
t	Specified the time of a previously executed time step in nonlinear transient analysis (Real, $t_0 \leq t \leq t_n$, where t_0 is the initial time of STEP j, and t_n is the last time of STEP j; Default = t_0).

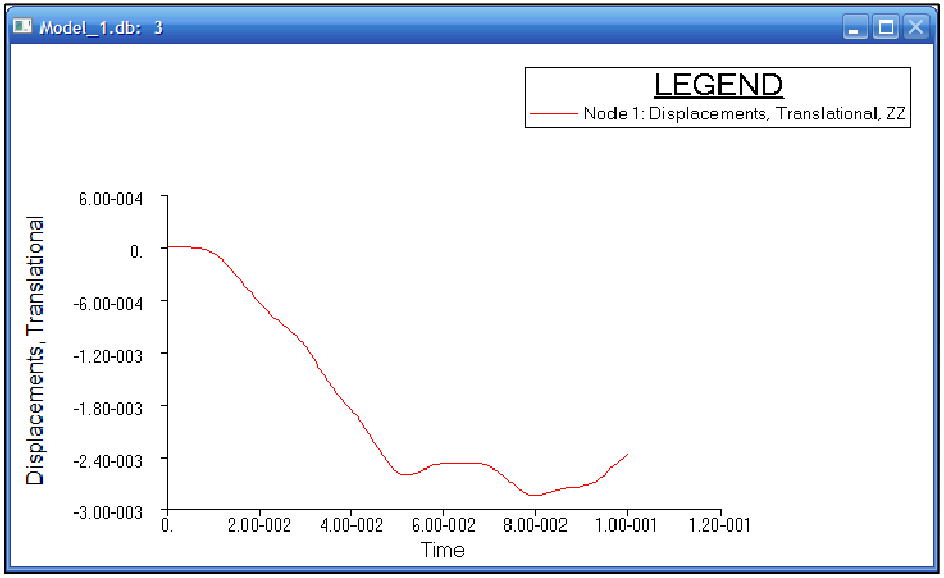
REMARKS

- **Nonlinear Restart is not supported in SOL 400 if advanced elements are used.**
- **Patran does not support Restart capability, manual editing of the input file is required.**
- **The MASTER/DBALL has all the restart information which needs to be assigned as shown in the example in previous slides.**
- **This means in the original run the user cannot use SCR=YES**.

EXERCISE

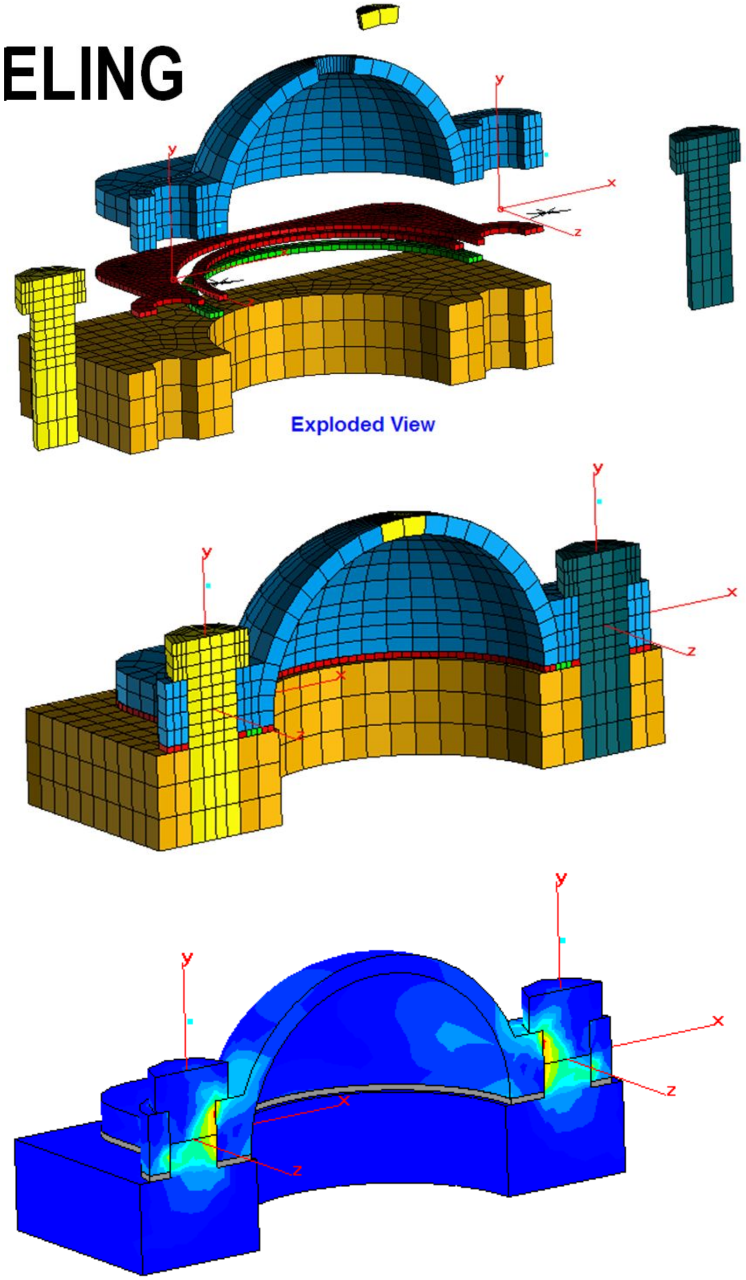
- **Perform Workshop 9: Restart**



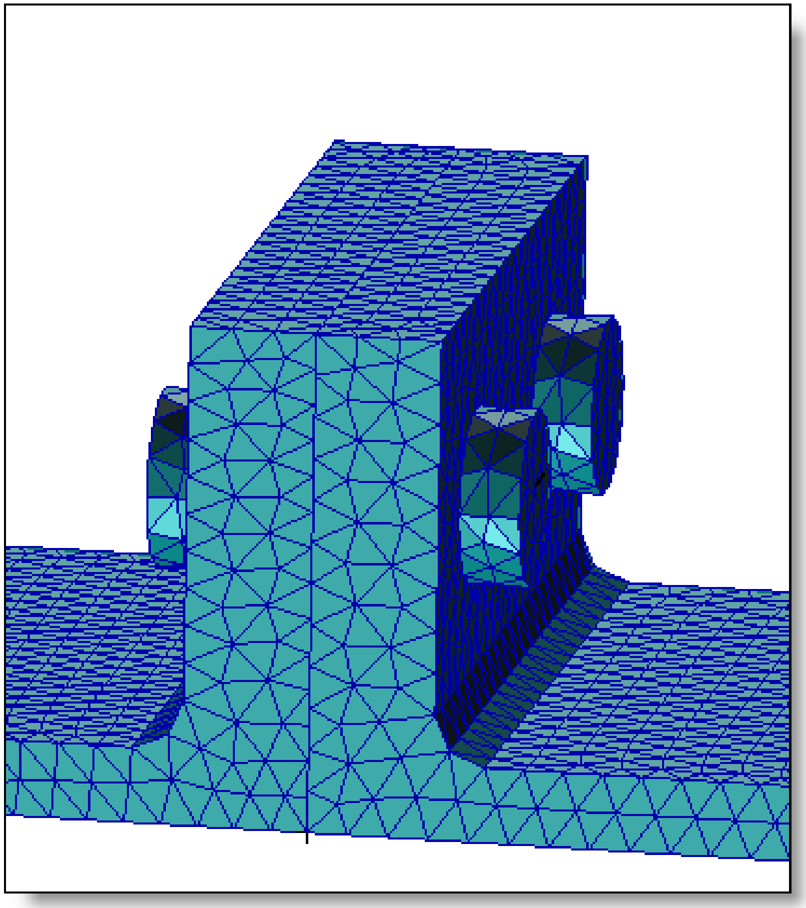


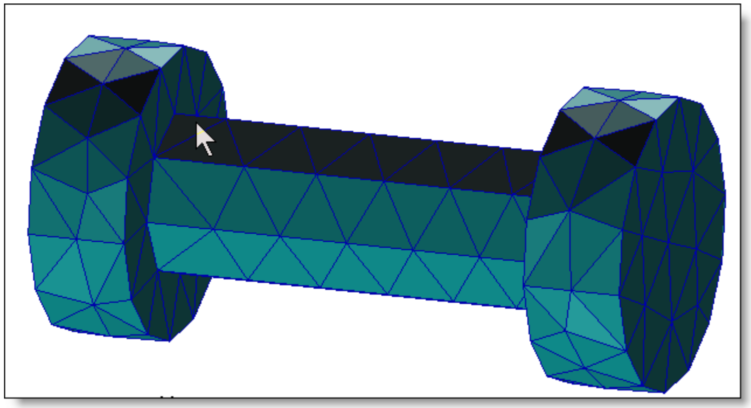
AUTOMATED BOLT PRELOAD MODELING

- **Description**
  - Automatic calculation of preload
- **How did you do it before?**
  - Split mesh
  - Write MPC's
  - Remove MPC Grids from Contact region
  - Applied thermal loads
- **Why is it better?**
  - New "BOLT" entry - Automatic MPCs
  - Continuous contact on bolt shaft
  - General method for pre-stressing, available for all element types (solids, shells, beams)



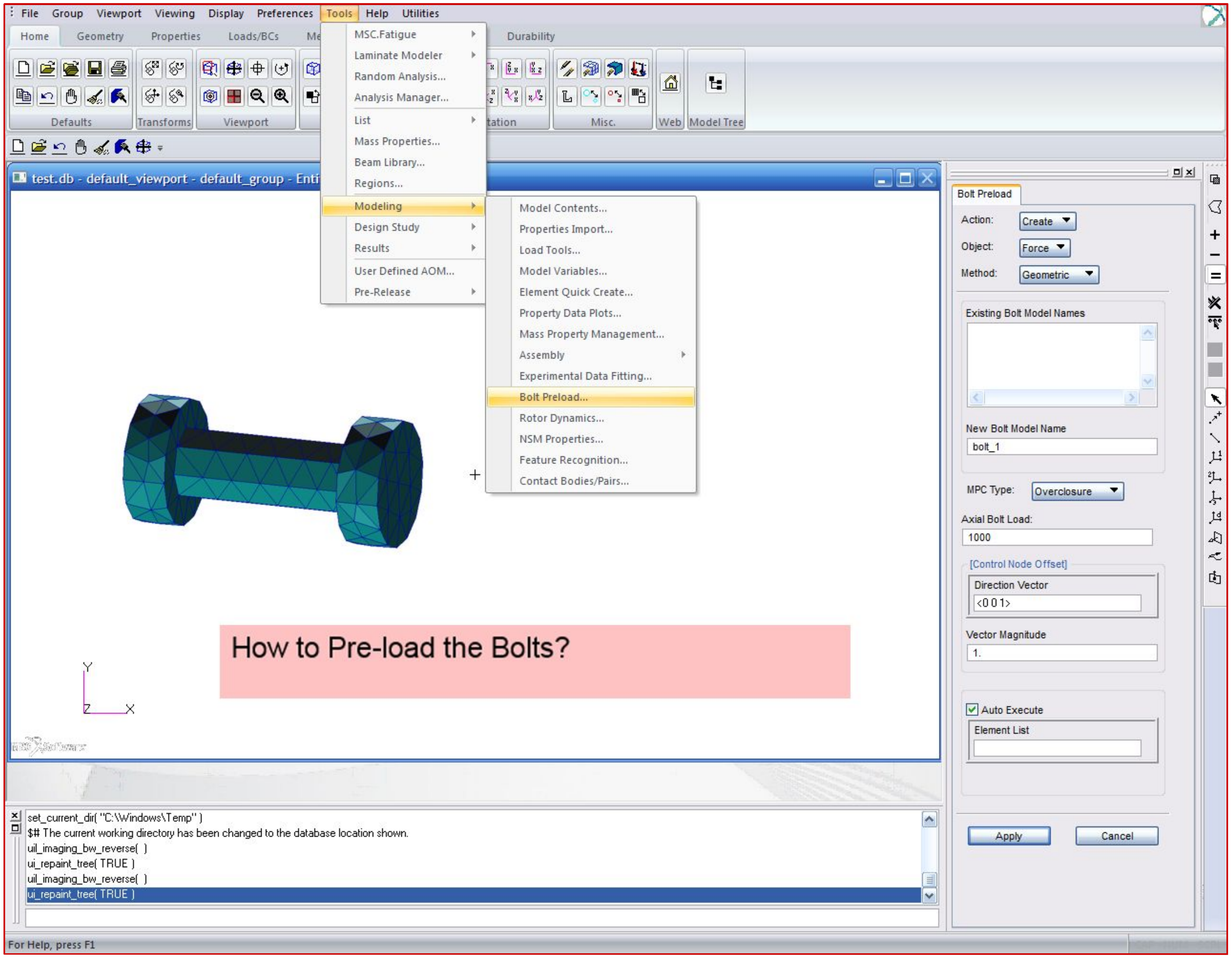
AUTOMATED BOLT PRELOAD MODELING: HOW?





How to Pre-load the Bolts?

AUTOMATED BOLT PRELOAD MODELING: HOW? (PATRAN SETUP)



How to Pre-load the Bolts?

Action: Create

Object: Force

Method: Geometric

Existing Bolt Model Names

New Bolt Model Name: bolt_1

MPC Type: Overclosure

Axial Bolt Load: 1000

[Control Node Offset]

Direction Vector: <0 0 1>

Vector Magnitude: 1.

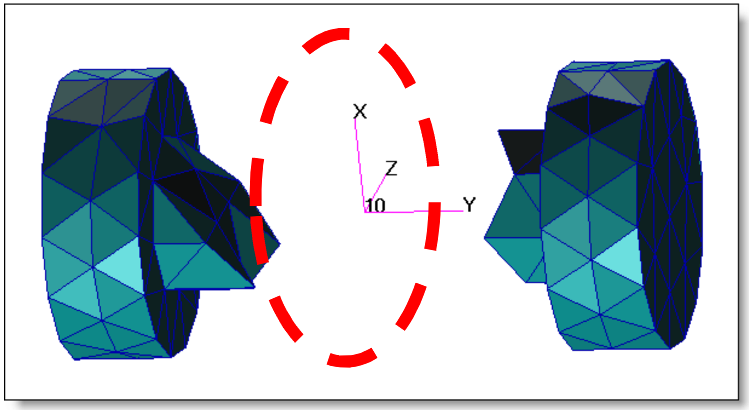
☑ Auto Execute

Element List

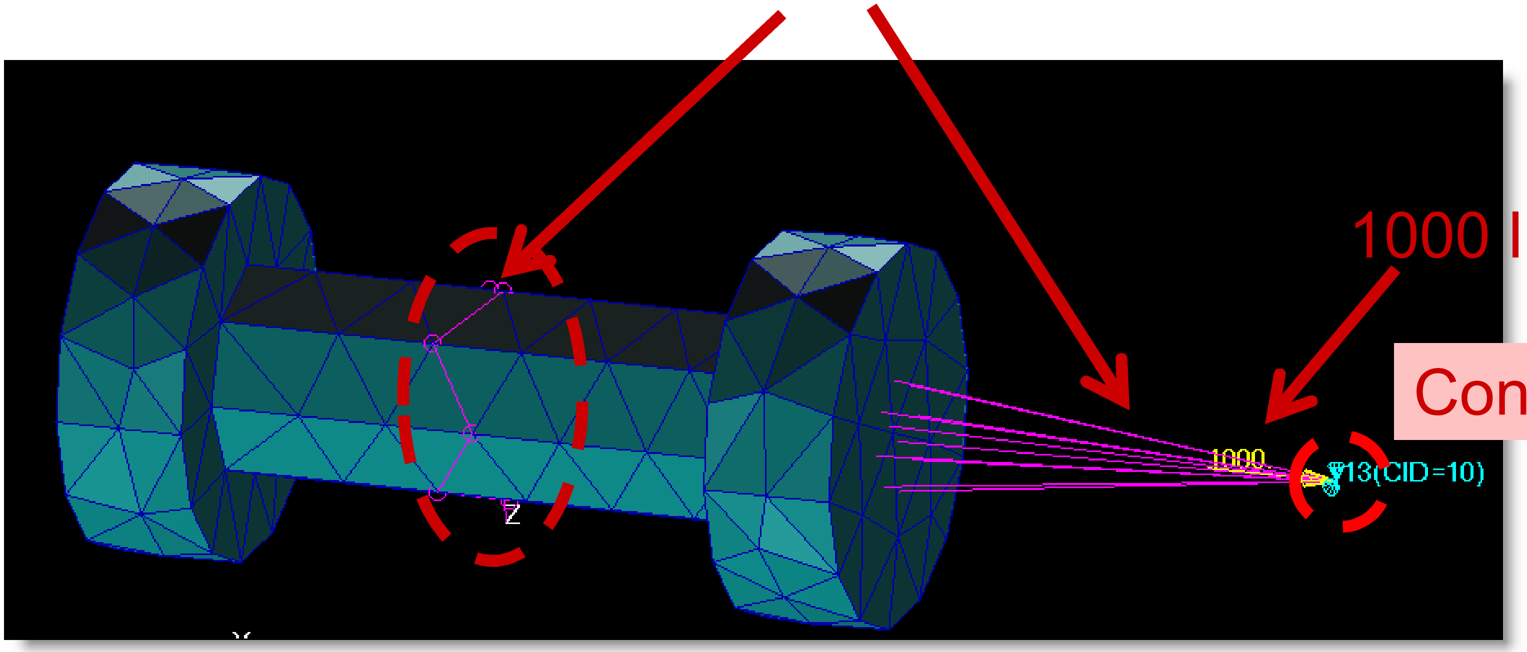
```
set_current_dir( "C:\Windows\Temp" )
$# The current working directory has been changed to the database location shown.
uil_imaging_bw_reverse( )
ui_repaint_tree( TRUE )
uil_imaging_bw_reverse( )
ui_repaint_tree( TRUE )
```

AUTOMATED BOLT PRELOAD MODELING: HOW?

- **Automatically Splits the Mesh and determines Bolt “long axis”**
- **Automatically creates necessary coordinate system and control node**
- **Automatically creates the BOLT Entry**



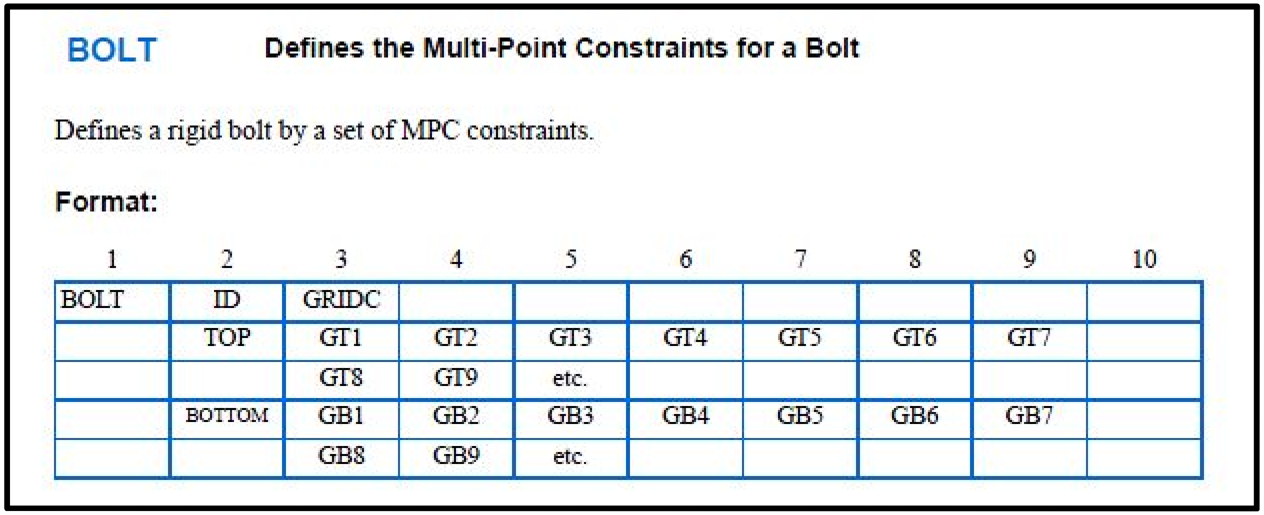
- **Automatically creates user specified pre-load or shrinkage**



1000 lb preload

Control Node

AUTOMATED BOLT PRELOAD MODELING: HOW?

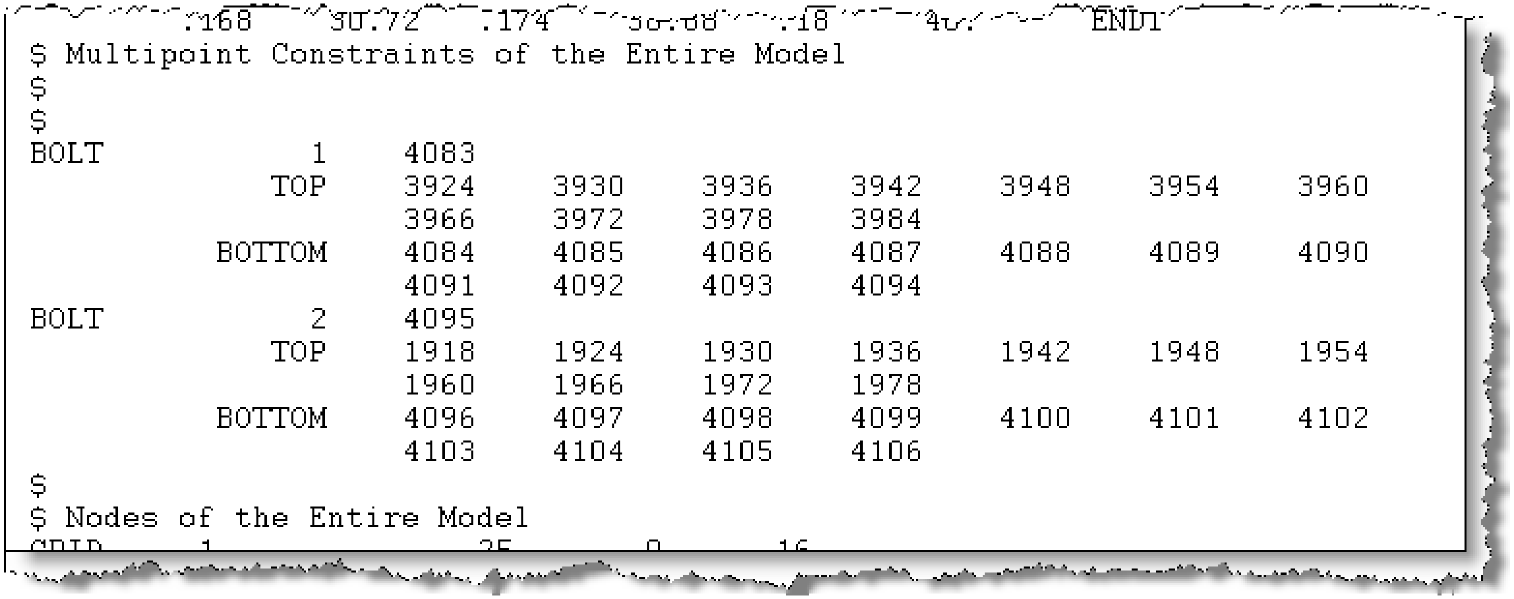


BOLT **Defines the Multi-Point Constraints for a Bolt**

Defines a rigid bolt by a set of MPC constraints.

Format:

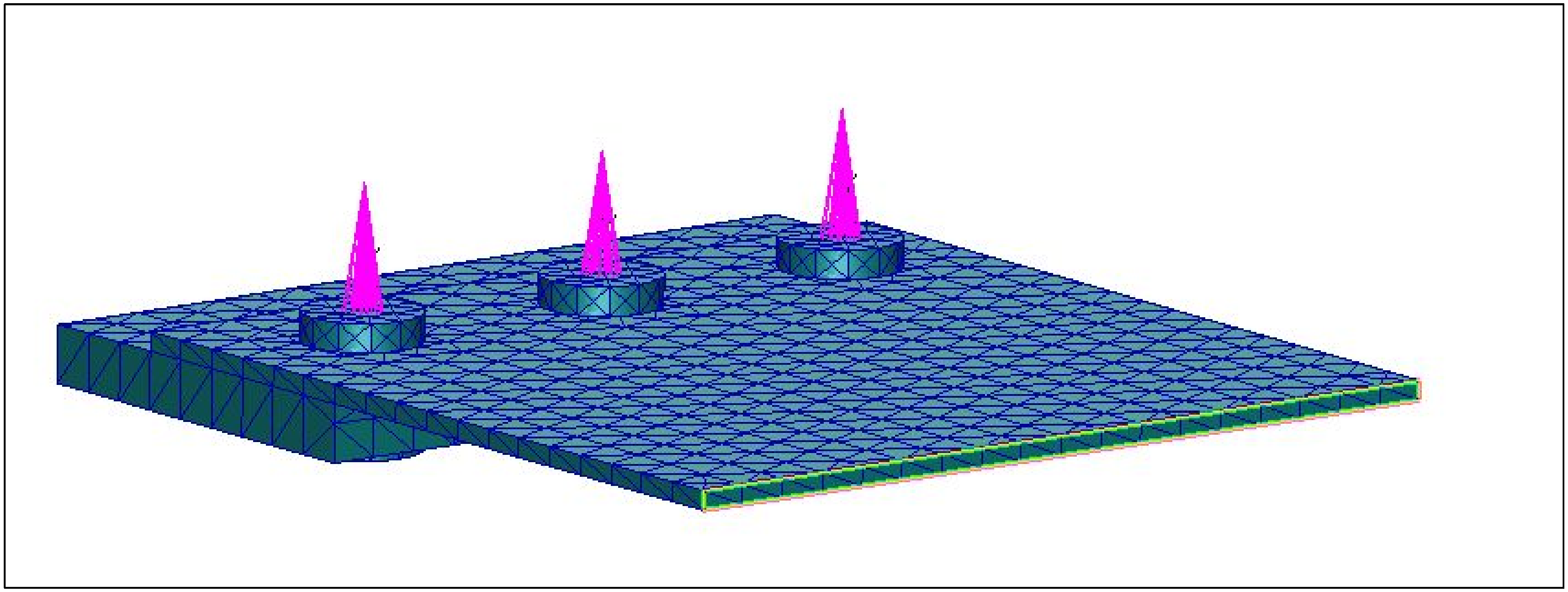
1	2	3	4	5	6	7	8	9	10
BOLT	ID	GRIDC							
	TOP	GT1	GT2	GT3	GT4	GT5	GT6	GT7	
		GT8	GT9	etc.					
	BOTTOM	GB1	GB2	GB3	GB4	GB5	GB6	GB7	
		GB8	GB9	etc.					



```
$ Multipoint Constraints of the Entire Model
$
$
BOLT           1    4083
            TOP     3924    3930    3936    3942    3948    3954    3960
                    3966    3972    3978    3984
         BOTTOM     4084    4085    4086    4087    4088    4089    4090
                    4091    4092    4093    4094
BOLT           2    4095
            TOP     1918    1924    1930    1936    1942    1948    1954
                    1960    1966    1972    1978
         BOTTOM     4096    4097    4098    4099    4100    4101    4102
                    4103    4104    4105    4106
$
$ Nodes of the Entire Model
```

EXERCISE

- **Perform Workshop 10: Bolt Modeling**



SECTION 8

CONTACT POST-PROCESSING

OVERVIEW

- **Contact Results Output**
- **Plot the Contact Status**
- **Glued Contact**
- **NLOPRM**

CONTACT RESULTS OUTPUT

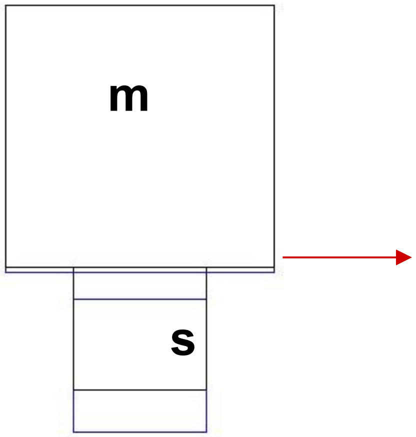
- **BOUTPUT**
  - Output of contact results
    - contact status, forces, and stresses
- **param,ph2out**
  - phase 2 output (default=0)
    - 0
      - regular phase III output (performed when all steps and subcases have been completed)
    - 1
      - phase II output only. This is useful when the run is terminated abnormally before the phase III outputs are formatted and printed. Prints in SORT1 format. There will be no output for the upstream superelements.
    - 3
      - both phase II and phase III outputs. In this case, some of the outputs for the residual structure may be redundant.

CONTACT RESULTS OUTPUT

```
N O D A L   F O R C E S   A N D   S T R E S S E S   I N   3 D   C O N T A C T   A N A L Y S I S

   LOAD STEP =  1.00000E+00                                                                          SUBCASE        1    STEP        1

   GRID    STATUS          CONTACT FORCES - RESIDUALS                       FRICTION FORCES                   NORMAL      FRICTION     FRICTION
                              EXCLUDING FRICTION                                                              STRESS      STRESS 1     STRESS 2
                        -       IN GLOBAL SYSTEM     -            -     IN GLOBAL SYSTEM     -             -     IN CONTACT LOCAL SYSTEM     -

      1         0     0.0000E+00  0.0000E+00  4.5417E-01     0.0000E+00  0.0000E+00  0.0000E+00     1.8167E-02  0.0000E+00  0.0000E+00
      2         0     0.0000E+00  0.0000E+00  4.5417E-01     0.0000E+00  0.0000E+00  0.0000E+00     1.8167E-02  0.0000E+00  0.0000E+00
      3         0     0.0000E+00  0.0000E+00  4.5417E-01     0.0000E+00  0.0000E+00  0.0000E+00     1.8167E-02  0.0000E+00  0.0000E+00
      4         0     0.0000E+00  0.0000E+00  4.5417E-01     0.0000E+00  0.0000E+00  0.0000E+00     1.8167E-02  0.0000E+00  0.0000E+00
     15         1    -3.1514E-18  1.3866E-17 -4.5417E-01     0.0000E+00  0.0000E+00  0.0000E+00     7.2667E-02  0.0000E+00  0.0000E+00
     16         1    -3.1514E-18  1.7648E-17 -4.5417E-01     0.0000E+00  0.0000E+00  0.0000E+00     7.2667E-02  0.0000E+00  0.0000E+00
     17         1     1.2606E-18  1.7648E-17 -4.5417E-01     0.0000E+00  0.0000E+00  0.0000E+00     7.2667E-02  0.0000E+00  0.0000E+00
     18         1     1.2606E-18  1.3866E-17 -4.5417E-01     0.0000E+00  0.0000E+00  0.0000E+00     7.2667E-02  0.0000E+00  0.0000E+00

           STATUS DEFINITION -----

           = 0  node is a retained node
           = 1  node is a tied node (touching node)
```

f06 file

Model



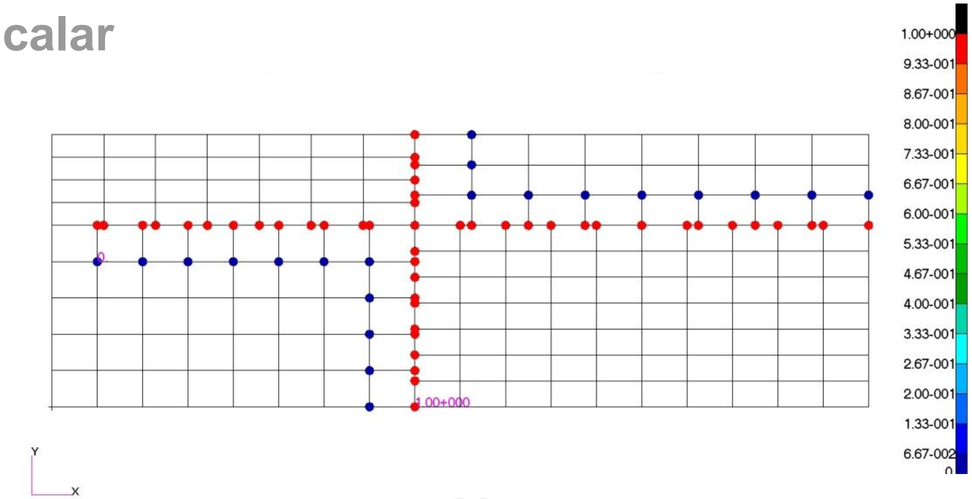
1, 2, 3, 4

15, 16, 17, 18

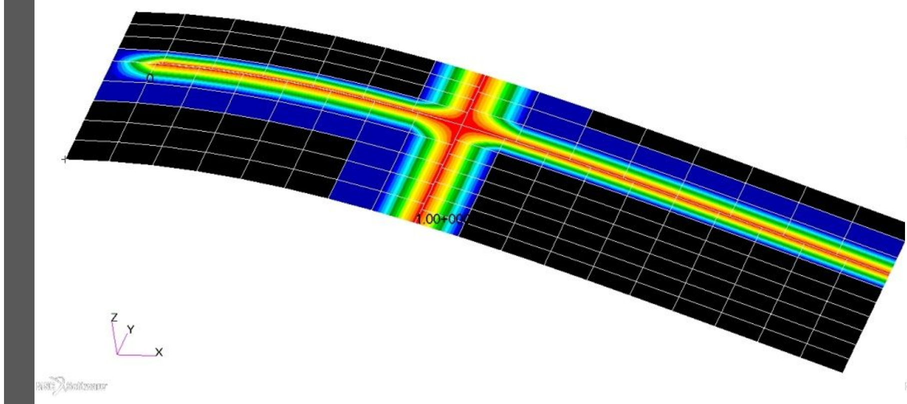
CONTACT RESULTS OUTPUT

- **Patran Contact Status Plots:**

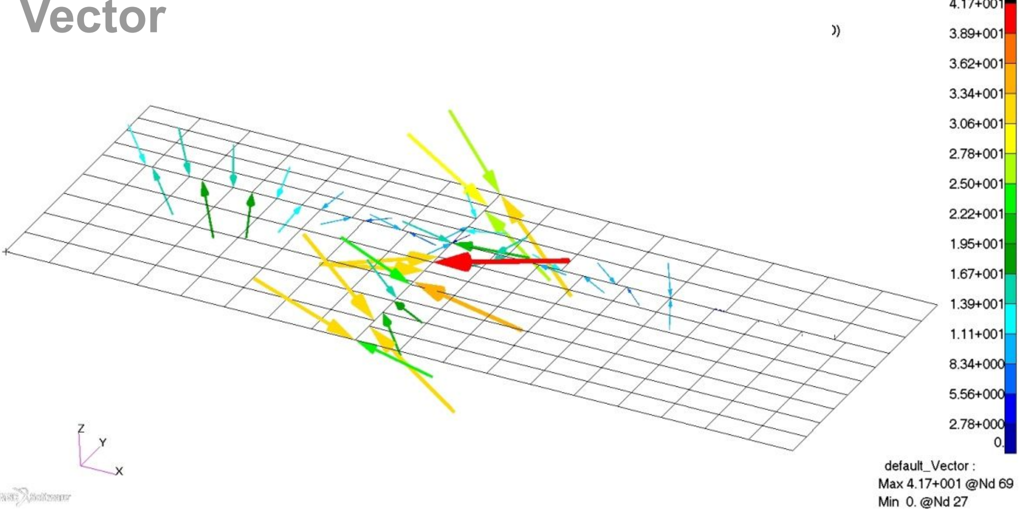
Scalar



Fringe



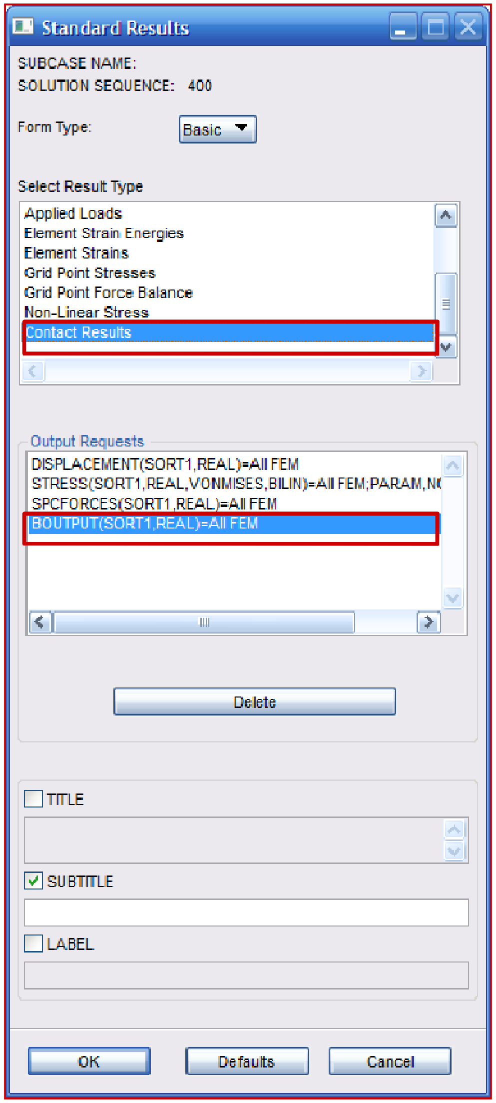
Vector



default_Vector :
Max 4.17+001 @Nd 69
Min 0. @Nd 27

PLOT THE CONTACT STATUS

- **Plotting the Contact Status**
  - First request for Contact Status as an output quantity
  - Next specify MASTER/DBALL as results output format (contact results are not available in the XDB format)

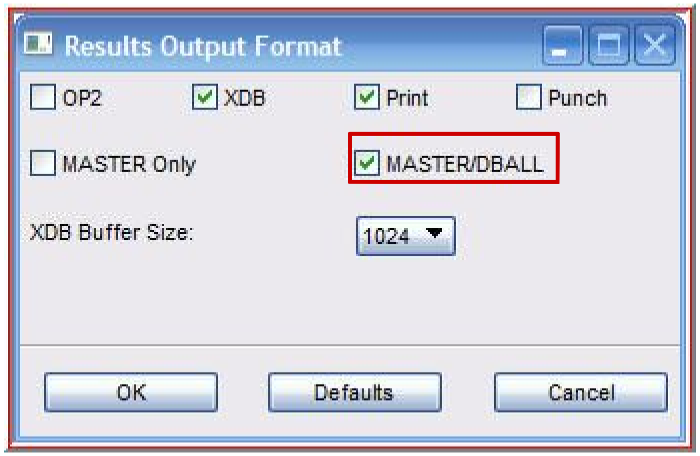


SUBCASE NAME:
SOLUTION SEQUENCE: 400

Form Type: Basic

```
DISPLACEMENT(SORT1,REAL)=All FEM
STRESS(SORT1,REAL,VONMISES,BILIN)=All FEM;PARAM,N
SPCFORCES(SORT1,REAL)=All FEM
BOUTPUT(SORT1,REAL)=All FEM
```

☐ TITLE
☑ SUBTITLE
☐ LABEL



☐ OP2 ☑ XDB ☑ Print ☐ Punch

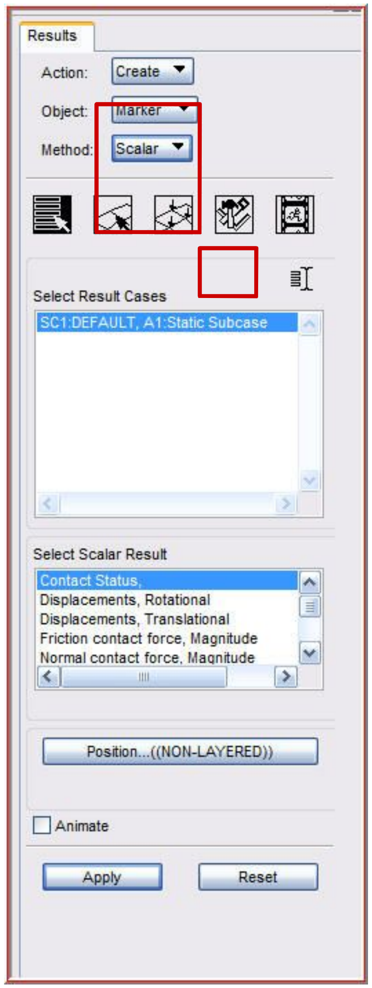
☐ MASTER Only ☑ MASTER/DBALL

XDB Buffer Size: 1024

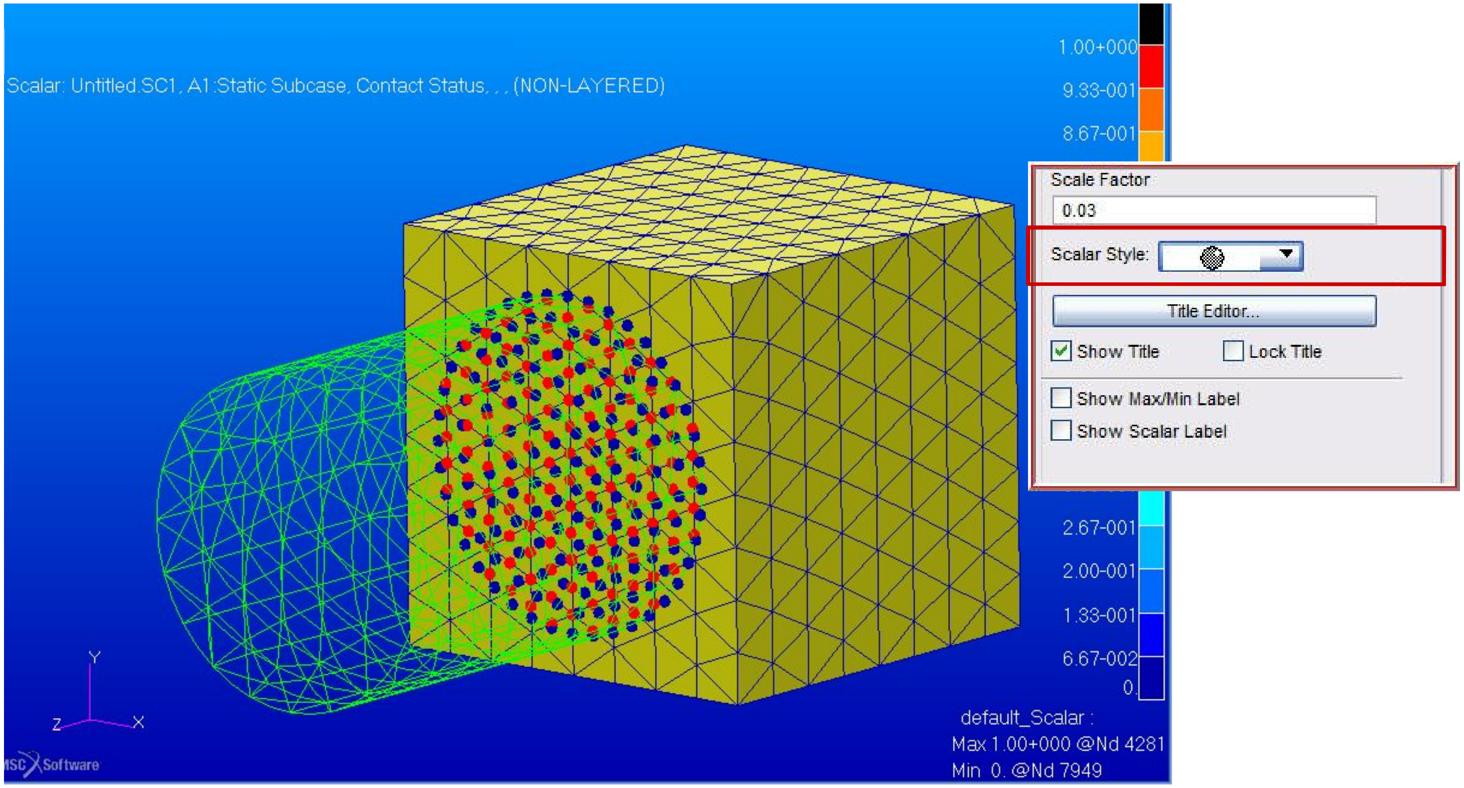
PLOT THE CONTACT STATUS

- **Plotting the Contact Status**
  - Attach the MASTER/DBALL
  - Make a scalar marker plot

Action: Access Results
Object: Attach MASTER
Method: Result Entities



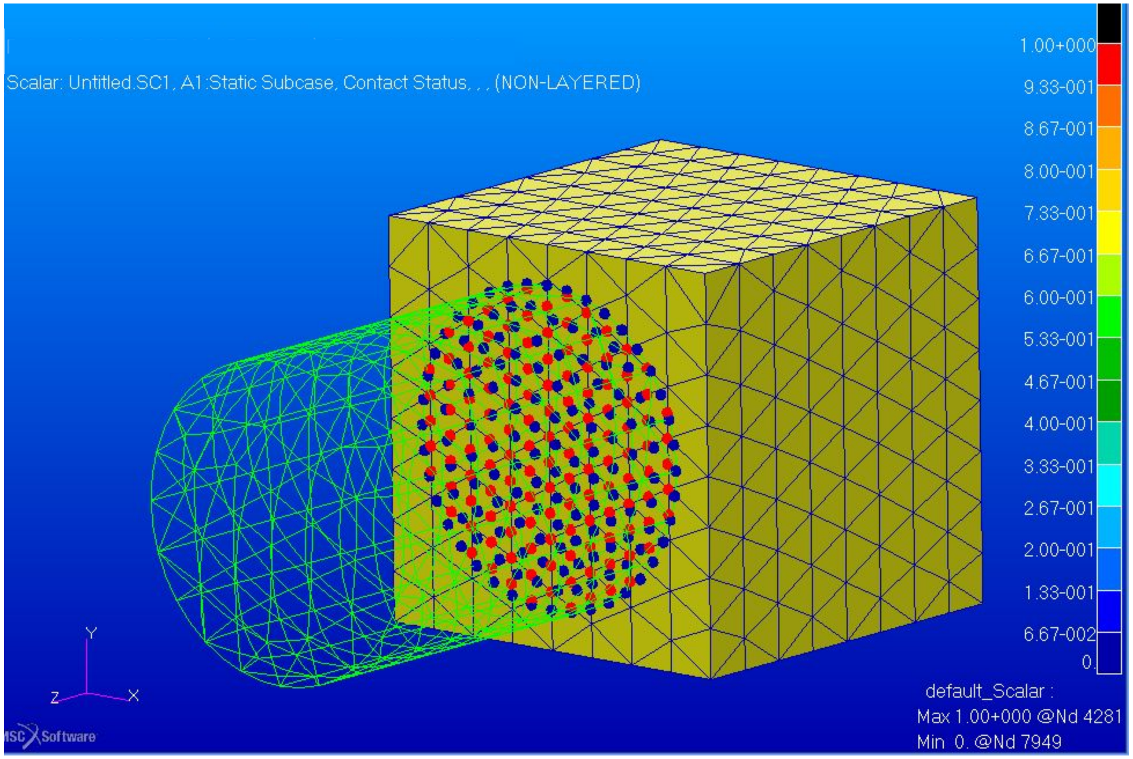
Action: Create
Object: Marker
Method: Scalar
Select Result Cases: SC1:DEFAULT, A1:Static Subcase
Select Scalar Result: Contact Status,



Scale Factor: 0.03
Scalar Style: ●
☑ Show Title ☐ Lock Title
☐ Show Max/Min Label
☐ Show Scalar Label

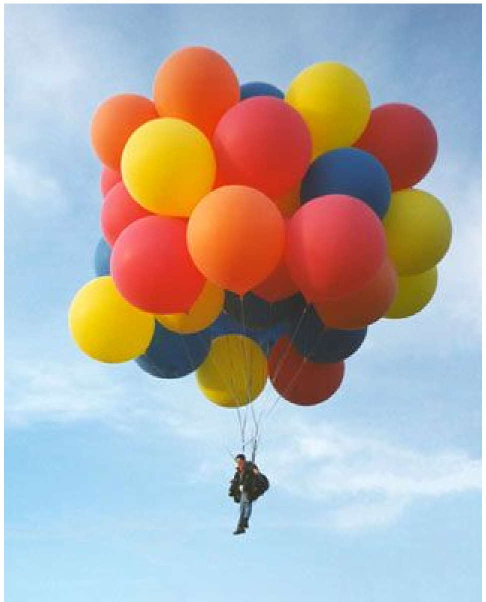
PLOT THE CONTACT STATUS

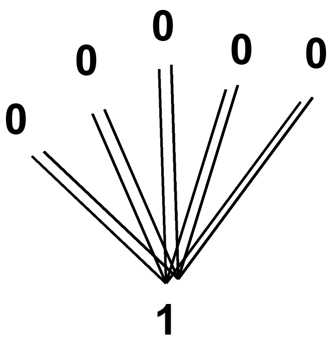
- **Contact status shows 0's and 1's**
  - 0 indicates a retained node (master)
  - 1 indicates a tied node (slave)



Scalar: Untitled.SC1, A1:Static Subcase, Contact Status, , , (NON-LAYERED)

default_Scalar :
Max 1.00+000 @Nd 4281
Min 0. @Nd 7949

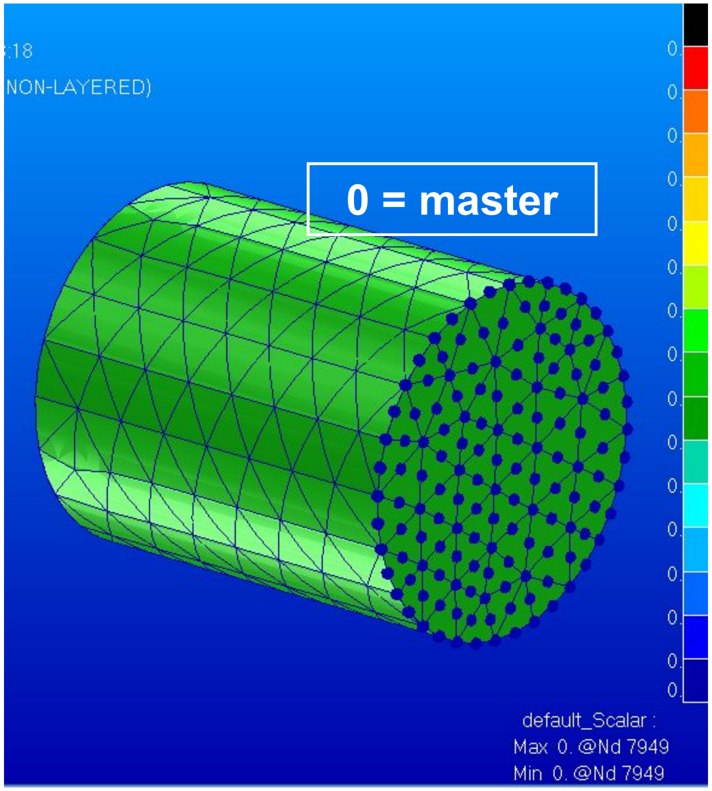


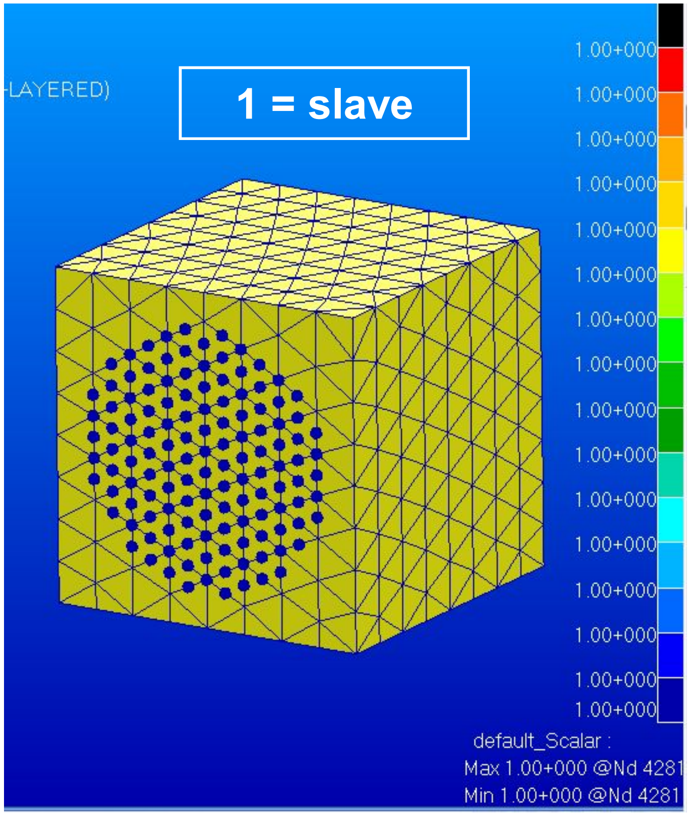


Memory aid

PLOT THE CONTACT STATUS

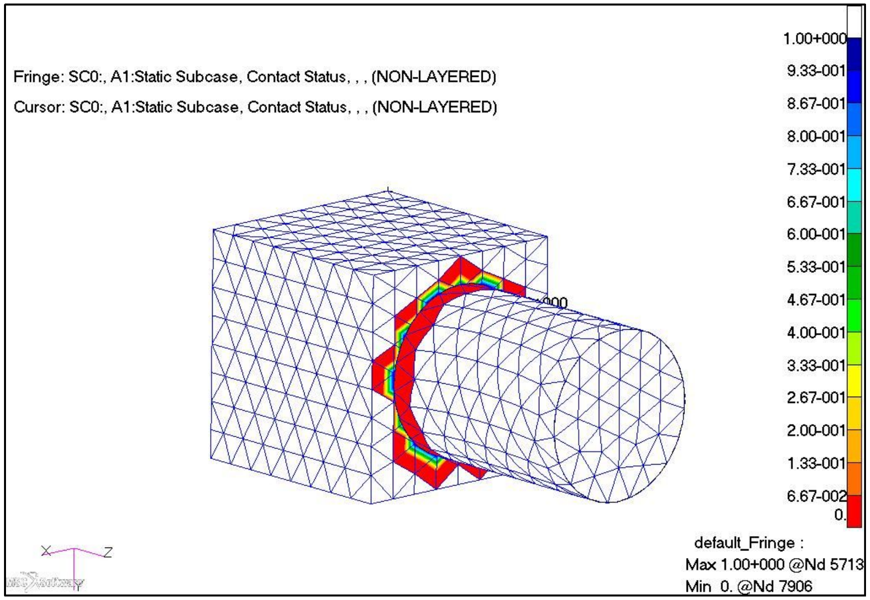
- **0 indicates a retained node (master)**
- **1 indicates a tied node (slave)**





PLOT THE CONTACT STATUS

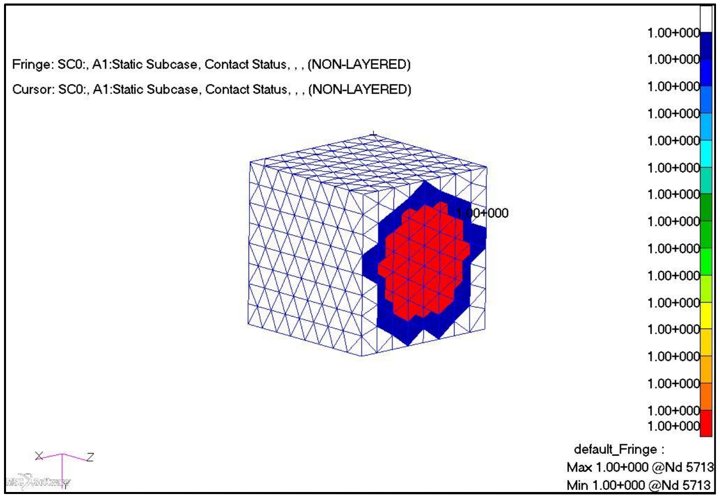
- **Contact status can also be displayed as a fringe plot**



Fringe: SC0:, A1:Static Subcase, Contact Status, , , (NON-LAYERED)

Cursor: SC0:, A1:Static Subcase, Contact Status, , , (NON-LAYERED)

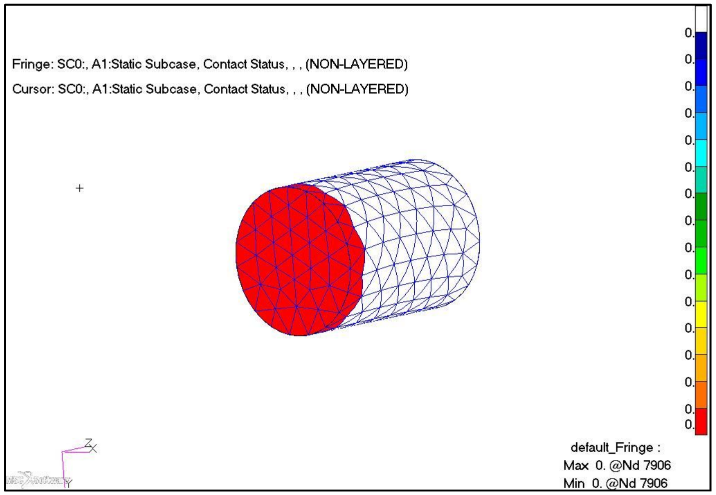
default_Fringe :
Max 1.00+000 @Nd 5713
Min 0. @Nd 7906



Fringe: SC0:, A1:Static Subcase, Contact Status, , , (NON-LAYERED)

Cursor: SC0:, A1:Static Subcase, Contact Status, , , (NON-LAYERED)

default_Fringe :
Max 1.00+000 @Nd 5713
Min 1.00+000 @Nd 5713



Fringe: SC0:, A1:Static Subcase, Contact Status, , , (NON-LAYERED)

Cursor: SC0:, A1:Static Subcase, Contact Status, , , (NON-LAYERED)

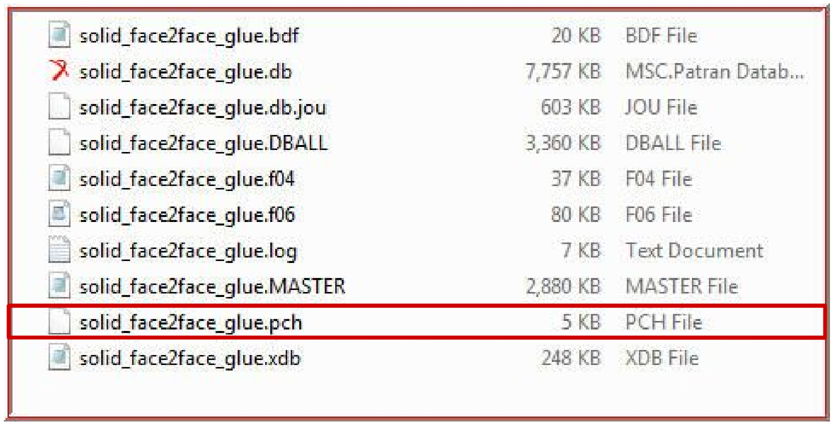
default_Fringe :
Max 0. @Nd 7906
Min 0. @Nd 7906

GLUED CONTACT – STATUS

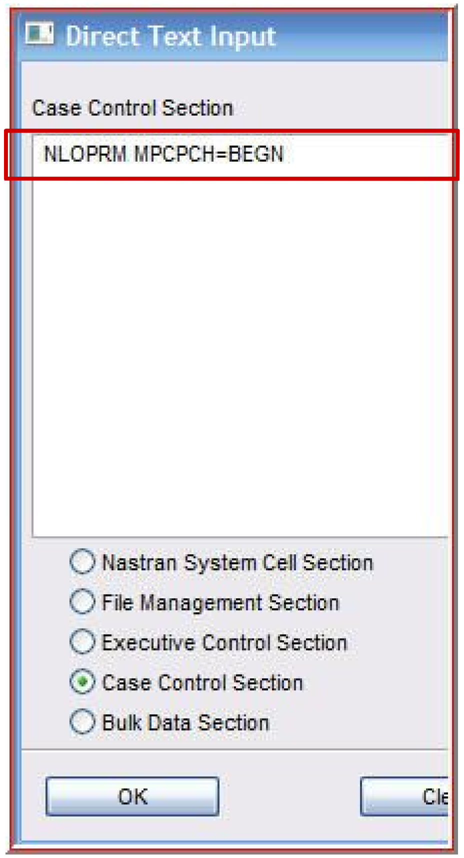
- **Determining the glued contact status is important. It is possible to glue only a few nodes on a large surface. This leads to incorrect deflections and stresses**
- **Glued contact can be evaluated by examining the deflected shapes and stresses of the model**
- **The MPCs used to model linear contact can be created during the MSC Nastran run in a punch (*.pch) file**

PLOT THE GLUE MPC EQUATIONS

- **MSC Nastran internally generates MPC equations to represent the permanent glued contact**
- **The user can request for these equations be output to a punch file**
  - Specify NLOPRM MPCPCH=BEGN in the Case Control section



solid_face2face_glue.bdf	20 KB	BDF File
solid_face2face_glue.db	7,757 KB	MSC.Patran Datab...
solid_face2face_glue.db.jou	603 KB	JOU File
solid_face2face_glue.DBALL	3,360 KB	DBALL File
solid_face2face_glue.f04	37 KB	F04 File
solid_face2face_glue.f06	80 KB	F06 File
solid_face2face_glue.log	7 KB	Text Document
solid_face2face_glue.MASTER	2,880 KB	MASTER File
solid_face2face_glue.pch	5 KB	PCH File
solid_face2face_glue.xdb	248 KB	XDB File



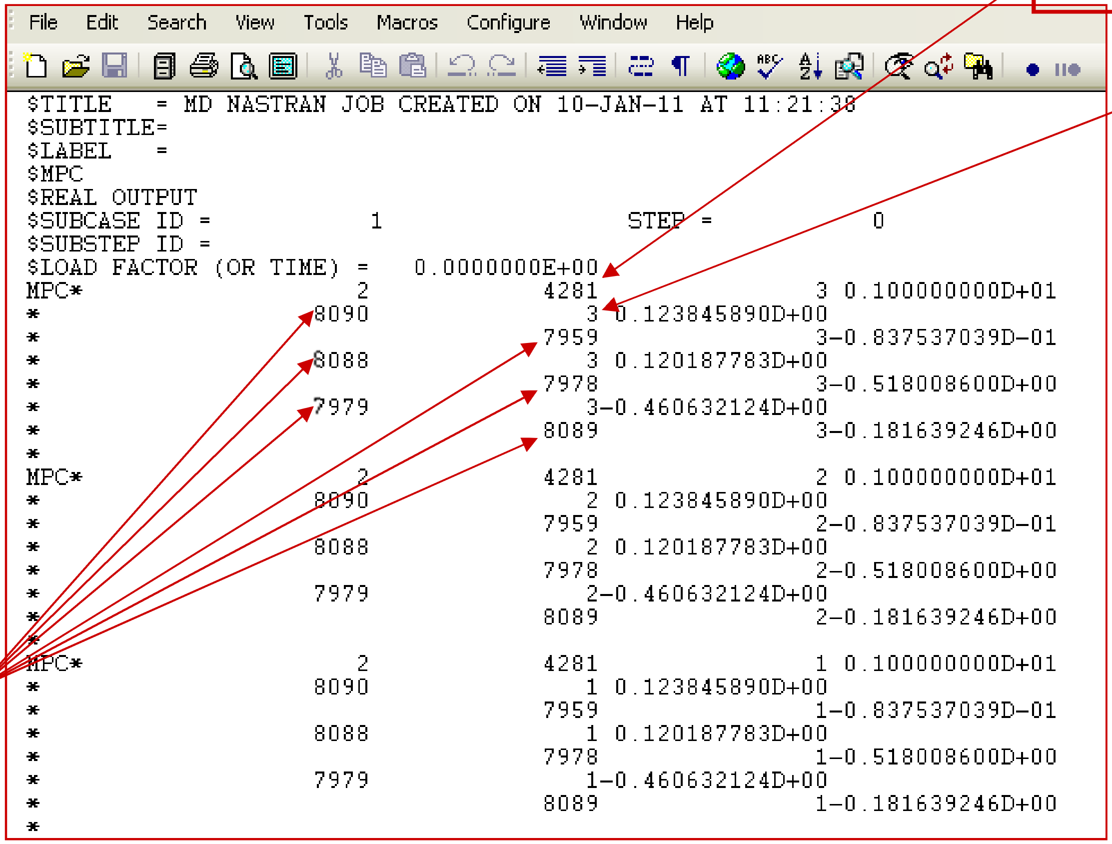
Case Control Section

NLOPRM MPCPCH=BEGN

○ Nastran System Cell Section
○ File Management Section
○ Executive Control Section
● Case Control Section
○ Bulk Data Section

PLOT THE GLUE MPC EQUATIONS

- **Review the MPC equations**
  - Open the .pch file in any text editor



```
$TITLE   = MD NASTRAN JOB CREATED ON 10-JAN-11 AT 11:21:38
$SUBTITLE=
$LABEL   =
$MPC
$REAL OUTPUT
$SUBCASE ID =           1                 STEP =           0
$SUBSTEP ID =
$LOAD FACTOR (OR TIME) =   0.0000000E+00
MPC*                   2            4281               3 0.100000000D+01
*                   8090               3 0.123845890D+00
*                                   7959               3-0.837537039D-01
*                   8088               3 0.120187783D+00
*                                   7978               3-0.518008600D+00
*                   7979               3-0.460632124D+00
*                                   8089               3-0.181639246D+00
*
MPC*                   2            4281               2 0.100000000D+01
*                   8090               2 0.123845890D+00
*                                   7959               2-0.837537039D-01
*                   8088               2 0.120187783D+00
*                                   7978               2-0.518008600D+00
*                   7979               2-0.460632124D+00
*                                   8089               2-0.181639246D+00
*
MPC*                   2            4281               1 0.100000000D+01
*                   8090               1 0.123845890D+00
*                                   7959               1-0.837537039D-01
*                   8088               1 0.120187783D+00
*                                   7978               1-0.518008600D+00
*                   7979               1-0.460632124D+00
*                                   8089               1-0.181639246D+00
*
```

Slave node

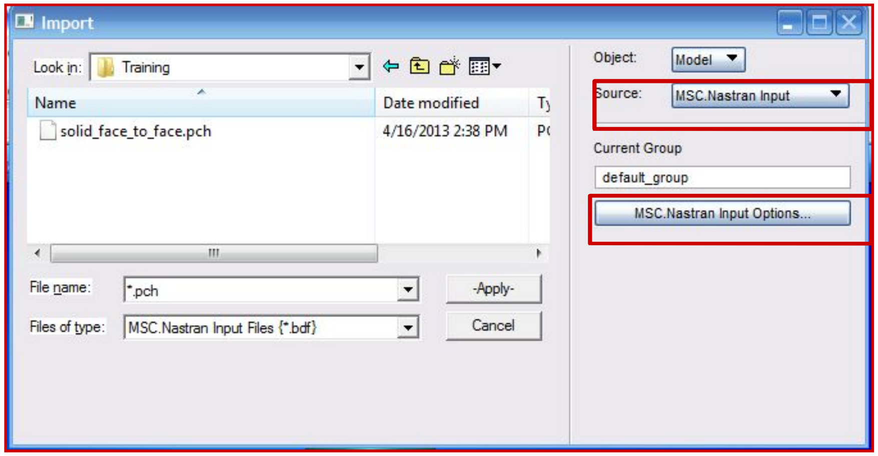
Degree of Freedom

Coefficient

Master nodes

PLOT THE GLUE MPC EQUATIONS

- **Plot the MPC equations**
  - Import the .pch file into Patran
  - Need to set Nodes ID offset to 0 before importing



Look in: Training

Name	Date modified
solid_face_to_face.pch	4/16/2013 2:38 PM

File name: *.pch

Files of type: MSC.Nastran Input Files (*.bdf)

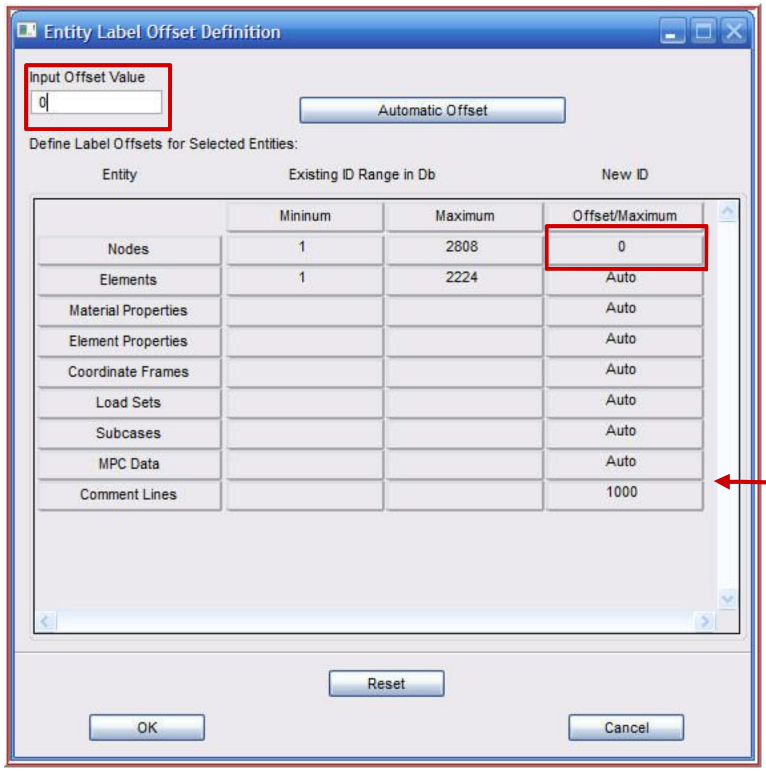
Object: Model

Source: MSC.Nastran Input

Current Group: default_group

☐ Create Prop. & Matl. Groups

☑ Retrieve Names From Comments



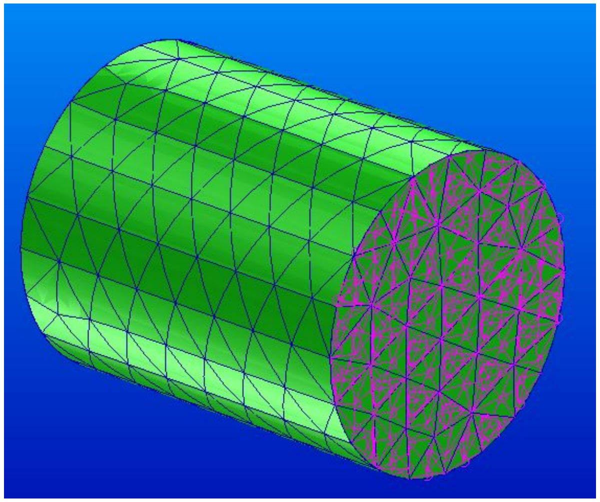
Input Offset Value: 0

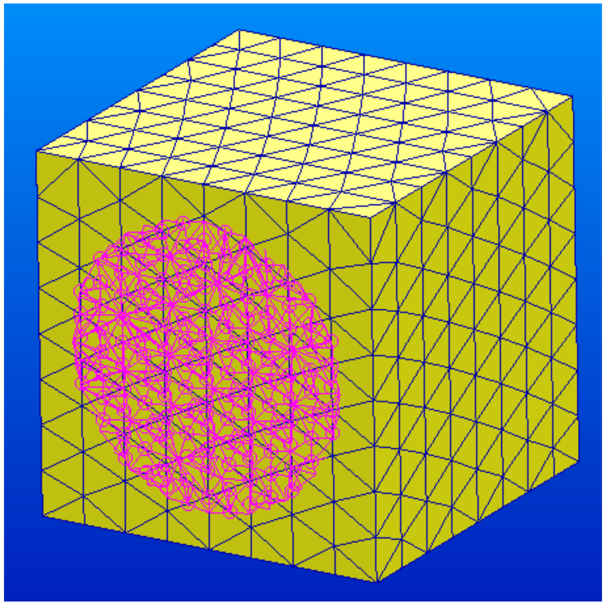
Define Label Offsets for Selected Entities:

Entity	Existing ID Range in Db		New ID
	Mininum	Maximum	Offset/Maximum
Nodes	1	2808	0
Elements	1	2224	Auto
Material Properties			Auto
Element Properties			Auto
Coordinate Frames			Auto
Load Sets			Auto
Subcases			Auto
MPC Data			Auto
Comment Lines			1000

PLOT THE GLUE MPC EQUATIONS

- **Plot the MPC equations**
  - Visually inspect the glued joint. Does the MPC pattern make sense? Is it uniformly distributed or spotty? Would reversing the contact search order improve the glued joint?
  - After reviewing the MPCs, don’t forget to remove them from the database by using UNDO or deleting them manually.





NLOPRM

- **This Case Control command provides additional information on the contact procedure and can be used to debug control runs.**
- **Complete description of this command can be found in the MSC Nastran Quick Reference Guide**
- **It can additionally provide direct access to nonlinear solutions even when the job is still running.**
- **NLOPRM NLDBG**
  - Provides information for 3D contact in the F06 file
    - = N3DBAS
      - prints error tolerance of each contact body.
    - = N3DMED
      - prints summary table of all contact parameters (in addition to the output for N3DBAS)
    - = N3DADV
      - prints body contact information in long form (in addition to the output for N3DBAS and N3DMED)
  - This information is available for touching contact only.

NLOPRM

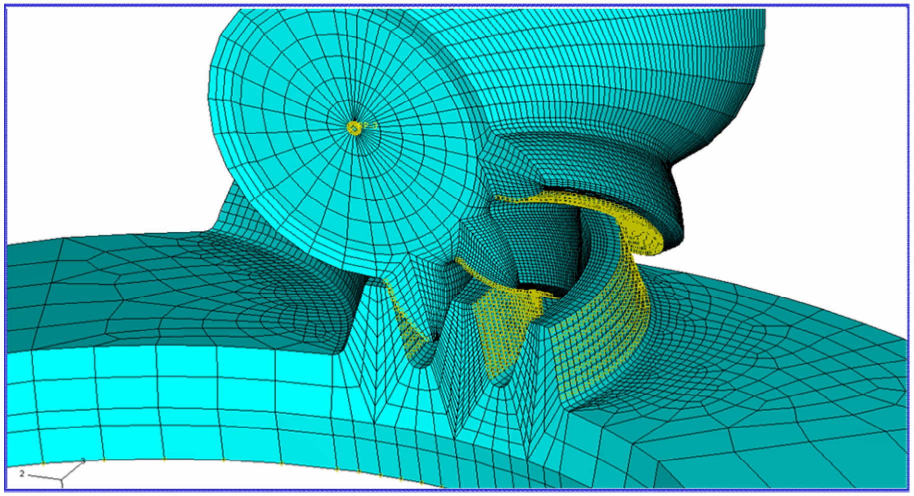
- **NLOPRM DBGPOST**
  - Selects POST options
    - = LTIME
      - Output all iterations in last load or time increment
    - = LSTEP
      - Output all iterations in last STEP
    - = LSUBC
      - Output all iterations in last SUBCASE
    - = ALL
      - Output all iterations
- **NLOPRM MPCPCH**
  - Create a Punch file of contact constraint equations as MPC Constraints
    - = BEGN
      - at beginning of the very first iteration
    - = OTIME
      - at every user requested output step
    - = STEP
      - at the end of each load case

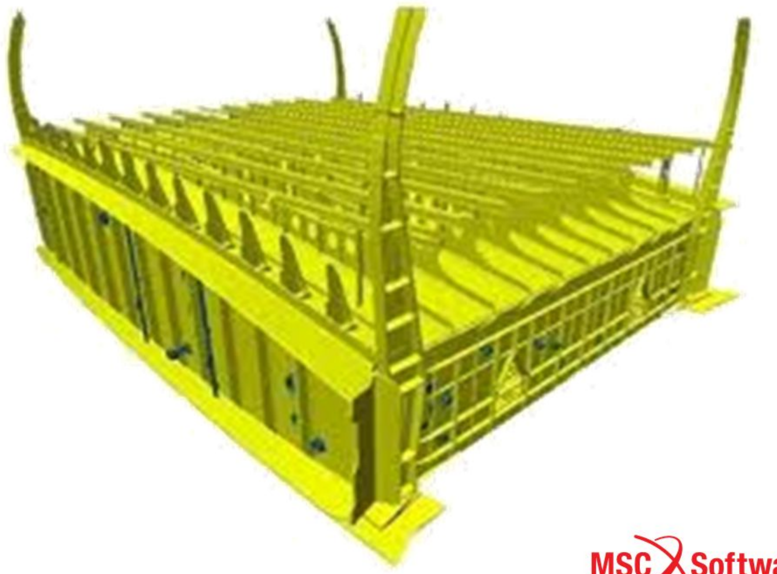
APPENDIX A

CONTACT PAIRS

CONTACT PAIRS: NEW APPROACH TO CONTACT MODEL SETUP

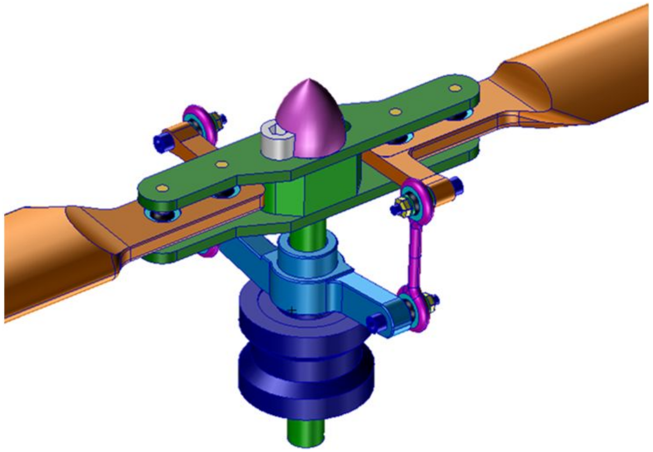
- **Multi-body Contact and Contact Tables works great for models with few contact bodies as shown left below**
- **But may not be easy to use for models with hundreds of contact bodies as shown right below. The new approach of creating contact bodies and contact pairs automatically through Patran GUI makes it much easier for these models.**





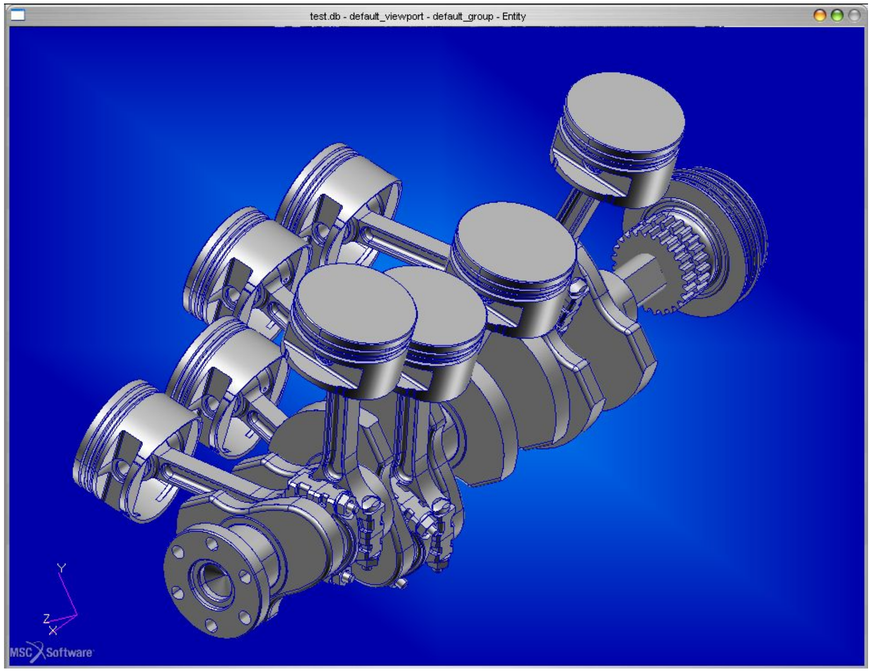
CONTACT PAIR MODELING

- Patran 2013 Supports MSC Nastran 2013.1 Contact Pair Modeling
- Patran 2013 Supports Automatic creation of Contact Bodies
- New Contact Pair Load and Boundary Condition (LBC) GUI
- The GUI allows user to create contact pairs using individual contact property sets, or reuse existing property sets.
- Along with the new contact pair user interface, is a set of tools that allow user to automatically create contact bodies and contact pairs based on user-defined criteria.



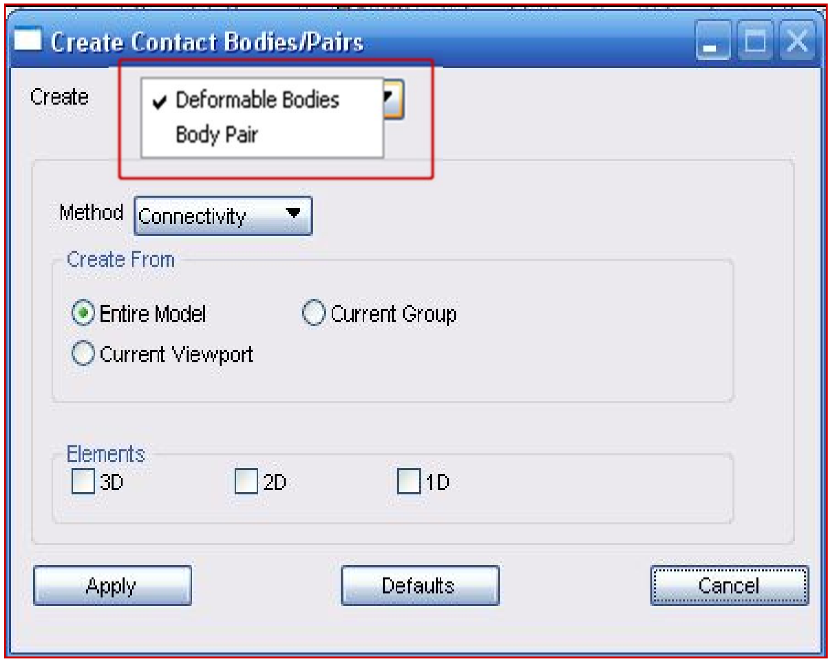
CONTACT INTERACTION: OVERVIEW

- **Contact body creation**
  - No change to manual body creation
  - Automatic body generation - many methods
- **Contact pair creation**
  - Manual pair creation
    - Use existing bodies
    - Create bodies with the pair
  - Automatic pair detection based on proximity
  - Global property editor provides scalability
- **Model browser contact interaction**



CONTACT INTERACTION

- **Simplify and automate contact set-up**
  - Automatic body generation
  - Automatic pair detection based on proximity
  - Global property editor to modify multiple pairs in one operation
- **Benefits**
  - Reduced assembly modeling effort
  - Fewer entries to define contact interaction



Create Contact Bodies/Pairs

Create: ✓ Deformable Bodies / Body Pair

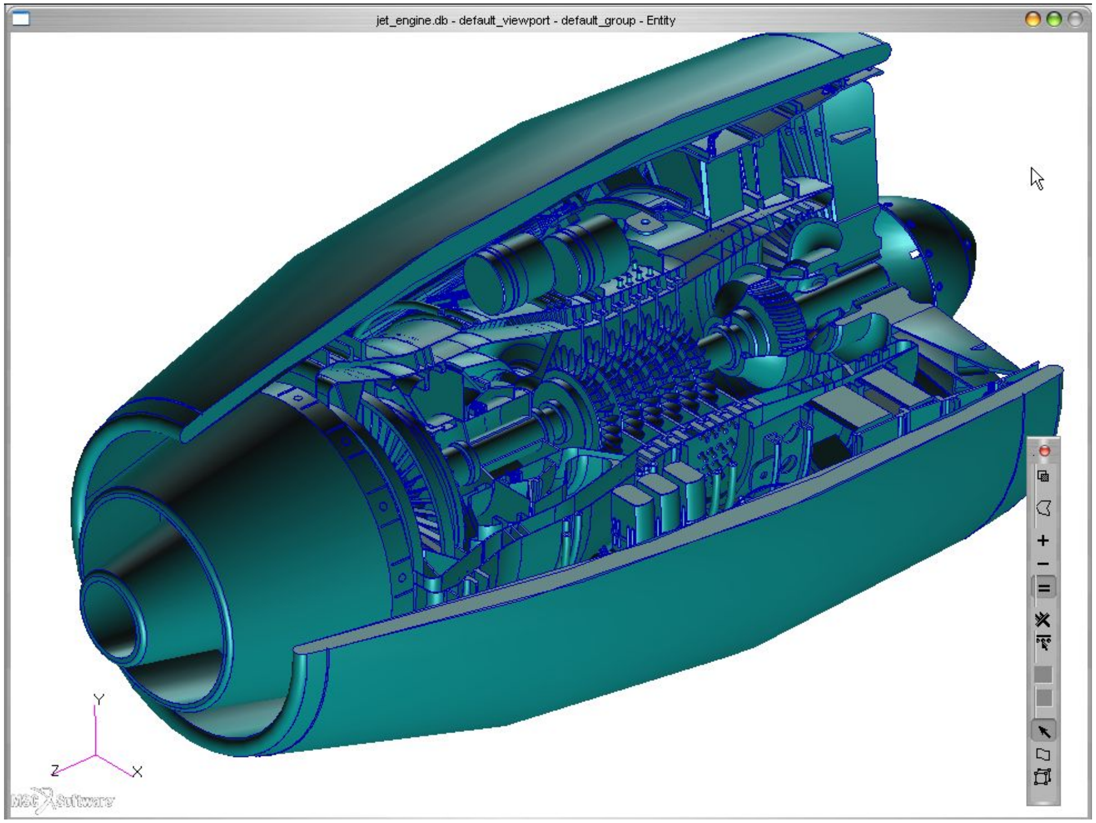
Method: Connectivity

Create From: ● Entire Model ○ Current Group ○ Current Viewport

Elements: ☐ 3D ☐ 2D ☐ 1D

AUTOMATIC CONTACT BODY GENERATION

- **Create contact bodies automatically based on**
  - Property set
  - Group
  - Material set
  - Topology
  - Connectivity
  - Automatic Name

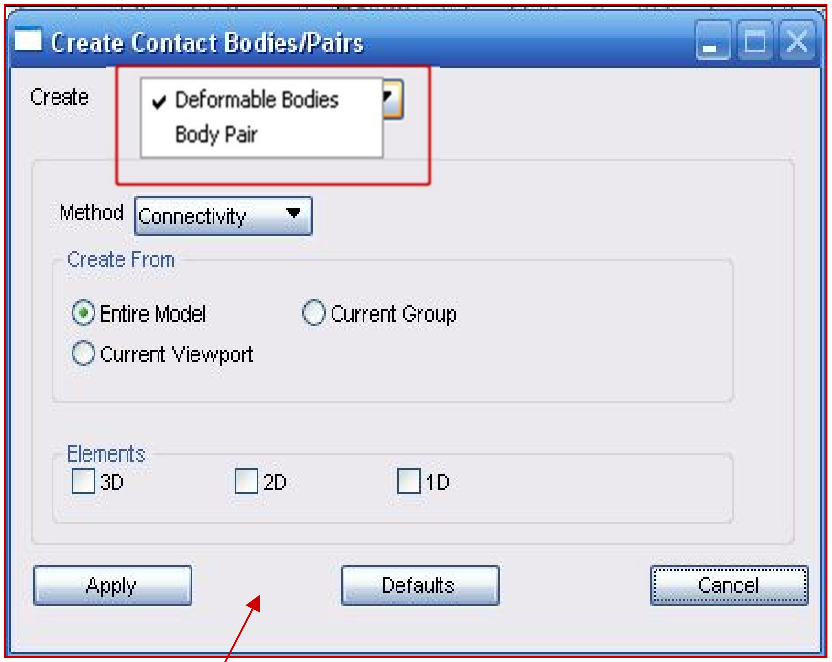


PATRAN CAPABILITIES OF CONTACT PAIR MODELING

- **Writes New BCTABLE1 / BCONECT Nastran bulk data entry**
- **Contact Pair Graphical User Interface**
- **Automatic Contact Body Creation Tool**
  1. Based on Property Sets
  2. Based on Material Sets
  3. Based on Group Membership
  4. Based on Element Membership
  5. Based on Element Topology
  6. Based on Connectivity
  7. Based on Geometry
- **Automatic Contact Pair Creation Tool Based on Proximity**
- **Model Brower Tree Interaction**
- **Powerful Global Property Editing**
- **Scalability for Large Number of Contact Bodies**

AUTOMATIC CONTACT BODY GENERATION IN PATRAN GUI

- **Create contact bodies automatically based on**
  - Property set
  - Group
  - Material set
  - Topology
  - Connectivity
  - Automatic Name Generation



Create Contact Bodies/Pairs

Create ✓ Deformable Bodies

Body Pair

Method Connectivity

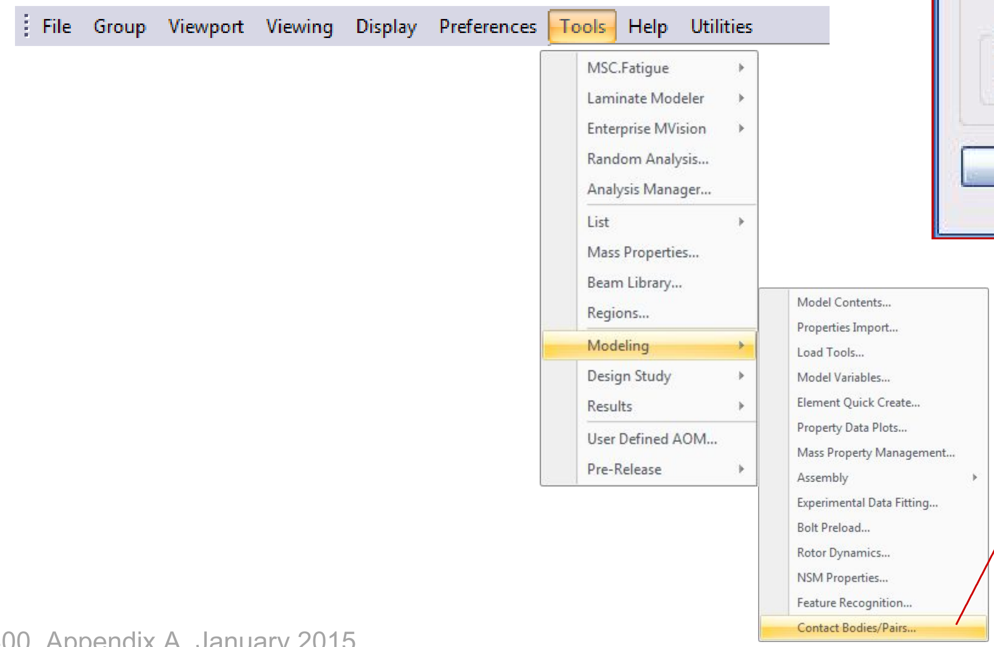
Create From

● Entire Model ○ Current Group

○ Current Viewport

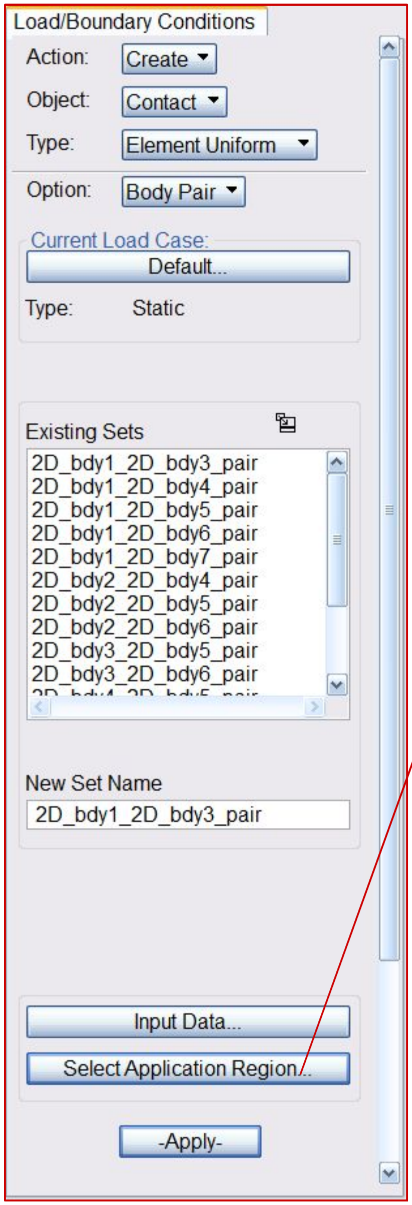
Elements

☐ 3D ☐ 2D ☐ 1D



CONTACT PAIR CREATION

- **Manual approach - using contact bodies already existing or created on the fly**
- **In Loads/BCs – Create/Contact/Body Pair**
- **Select existing bodies**
- **Create bodies as you create the body pairs**



Load/Boundary Conditions

Action: Create

Object: Contact

Type: Element Uniform

Option: Body Pair

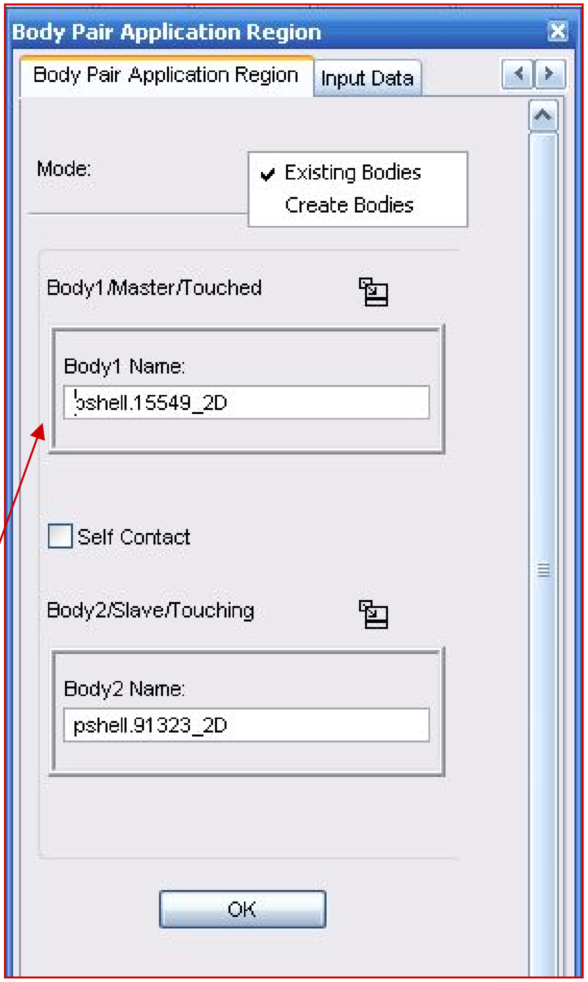
Current Load Case: Default...

Type: Static

Existing Sets

- 2D_bdy1_2D_bdy3_pair
- 2D_bdy1_2D_bdy4_pair
- 2D_bdy1_2D_bdy5_pair
- 2D_bdy1_2D_bdy6_pair
- 2D_bdy1_2D_bdy7_pair
- 2D_bdy2_2D_bdy4_pair
- 2D_bdy2_2D_bdy5_pair
- 2D_bdy2_2D_bdy6_pair
- 2D_bdy3_2D_bdy5_pair
- 2D_bdy3_2D_bdy6_pair

New Set Name: 2D_bdy1_2D_bdy3_pair



Body Pair Application Region

Mode: ✓ Existing Bodies / Create Bodies

Body1/Master/Touched

Body1 Name: pshell.15549_2D

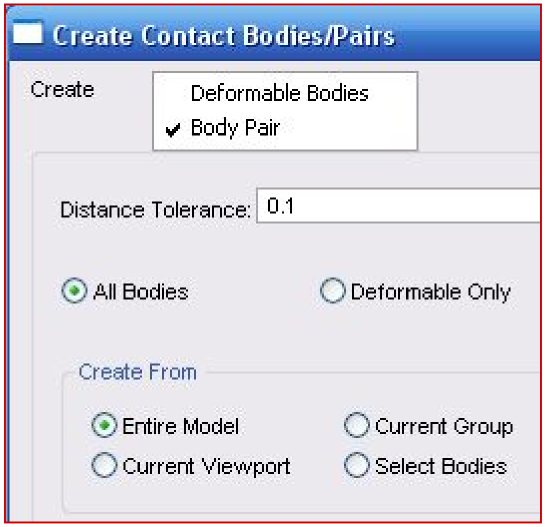
☐ Self Contact

Body2/Slave/Touching

Body2 Name: pshell.91323_2D

CONTACT PAIR CREATION AUTOMATIC APPROACH

- **User controls what parts of the model are considered**
  - Entire Model, Current Group or Current Viewport or Selected bodies
- **Automatic pair creation based on proximity (within given Distance Tolerance)**
- **Tools / Modeling / Contact Bodies/Pairs**
- **There will be a geometric property and a physical property created for each set of Body Pairs created.**



Create Contact Bodies/Pairs

Create: Deformable Bodies, ✓ Body Pair

Distance Tolerance: 0.1

● All Bodies ○ Deformable Only

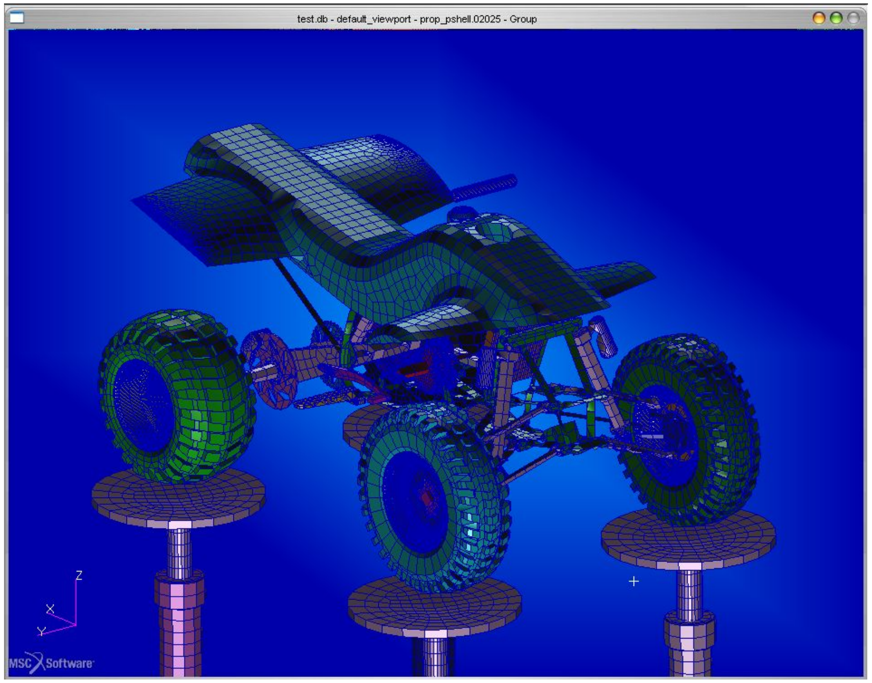
Create From

● Entire Model ○ Current Group

○ Current Viewport ○ Select Bodies

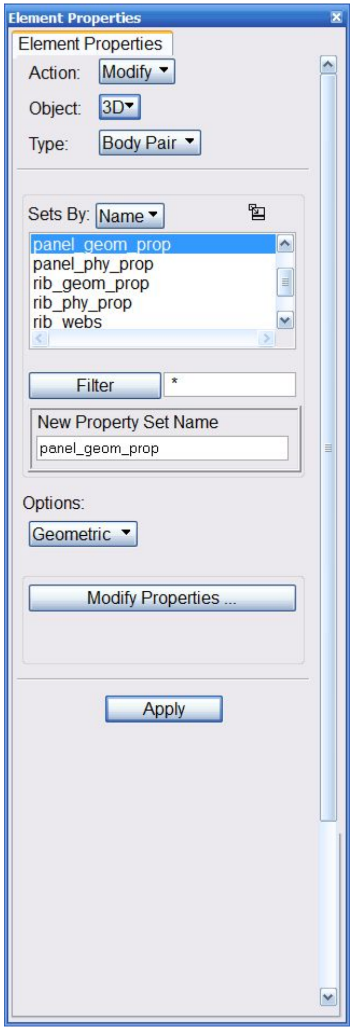
CONTACT PROPERTIES

- **Contact pairs consist of 3 data sets**
  - Geometric properties (BCONPRG)
  - Physical properties (BCONPRP)
  - LBC data
- **Easy creation and update of reference property sets**
  - User controls what parts of the model are considered
    - Group or viewport control
    - Create bodies with the pair



CONTACT PROPERTIES CAN BE EDITED UNDER PROPERTIES FORM

- **Contact Geometric Property**



Element Properties

Action: Modify

Object: 3D

Type: Body Pair

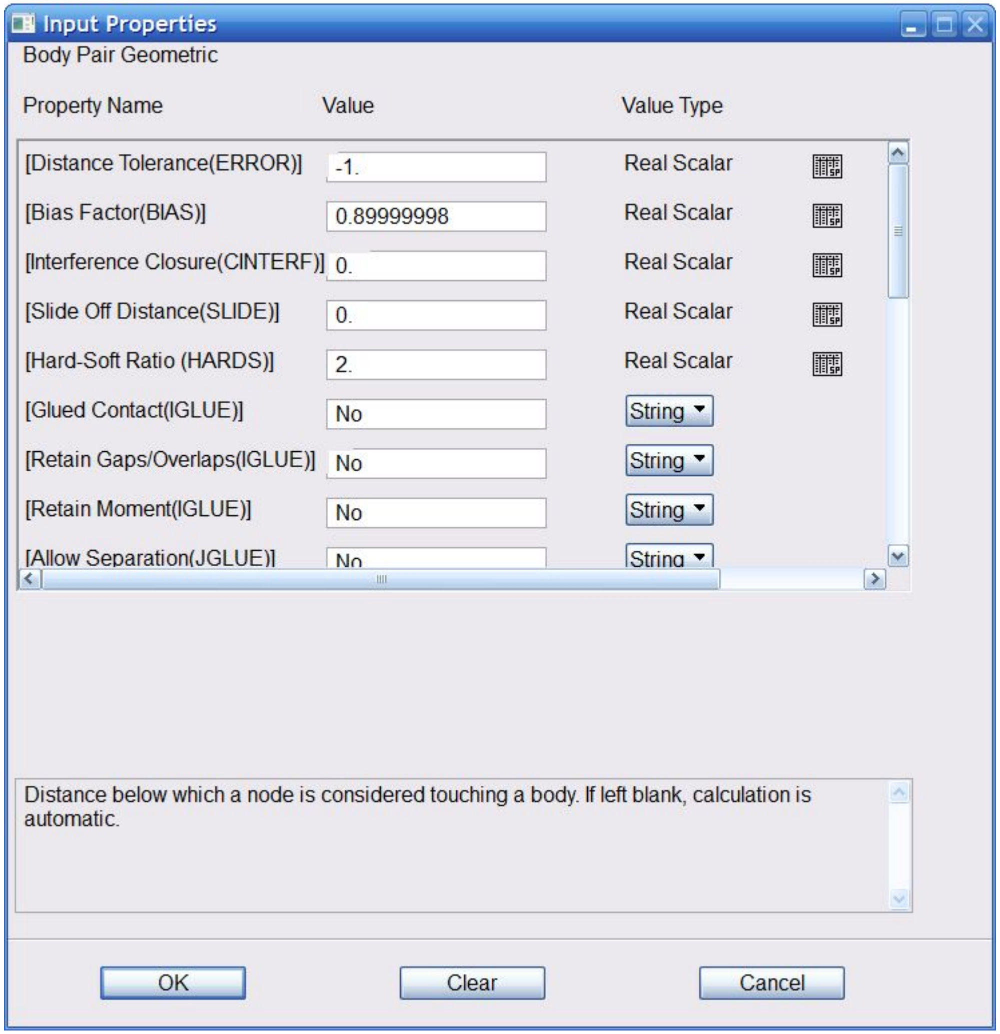
Sets By: Name

panel_geom_prop
panel_phy_prop
rib_geom_prop
rib_phy_prop
rib_webs

Filter: *

New Property Set Name: panel_geom_prop

Options: Geometric



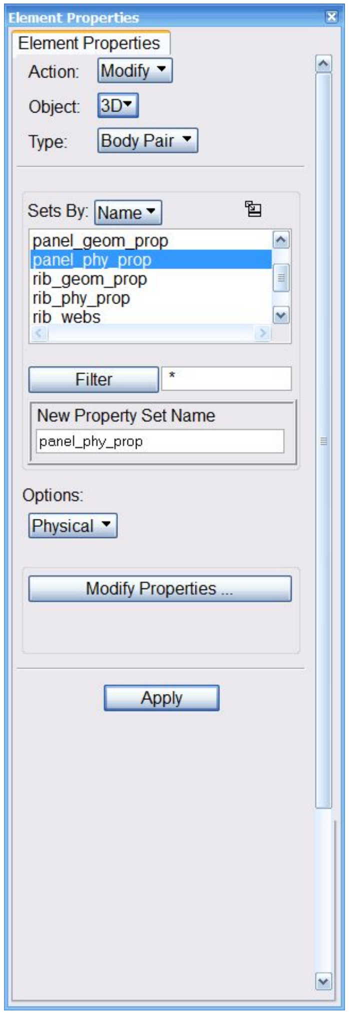
Body Pair Geometric

Property Name	Value	Value Type
[Distance Tolerance(ERROR)]	-1.	Real Scalar
[Bias Factor(BIAS)]	0.89999998	Real Scalar
[Interference Closure(CINTERF)]	0.	Real Scalar
[Slide Off Distance(SLIDE)]	0.	Real Scalar
[Hard-Soft Ratio (HARDS)]	2.	Real Scalar
[Glued Contact(IGLUE)]	No	String
[Retain Gaps/Overlaps(IGLUE)]	No	String
[Retain Moment(IGLUE)]	No	String
[Allow Separation(JGLUE)]	No	String

Distance below which a node is considered touching a body. If left blank, calculation is automatic.

CONTACT PROPERTIES CAN BE EDITED UNDER PROPERTIES FORM

- **Contact Physical Property**



Action: Modify
Object: 3D
Type: Body Pair

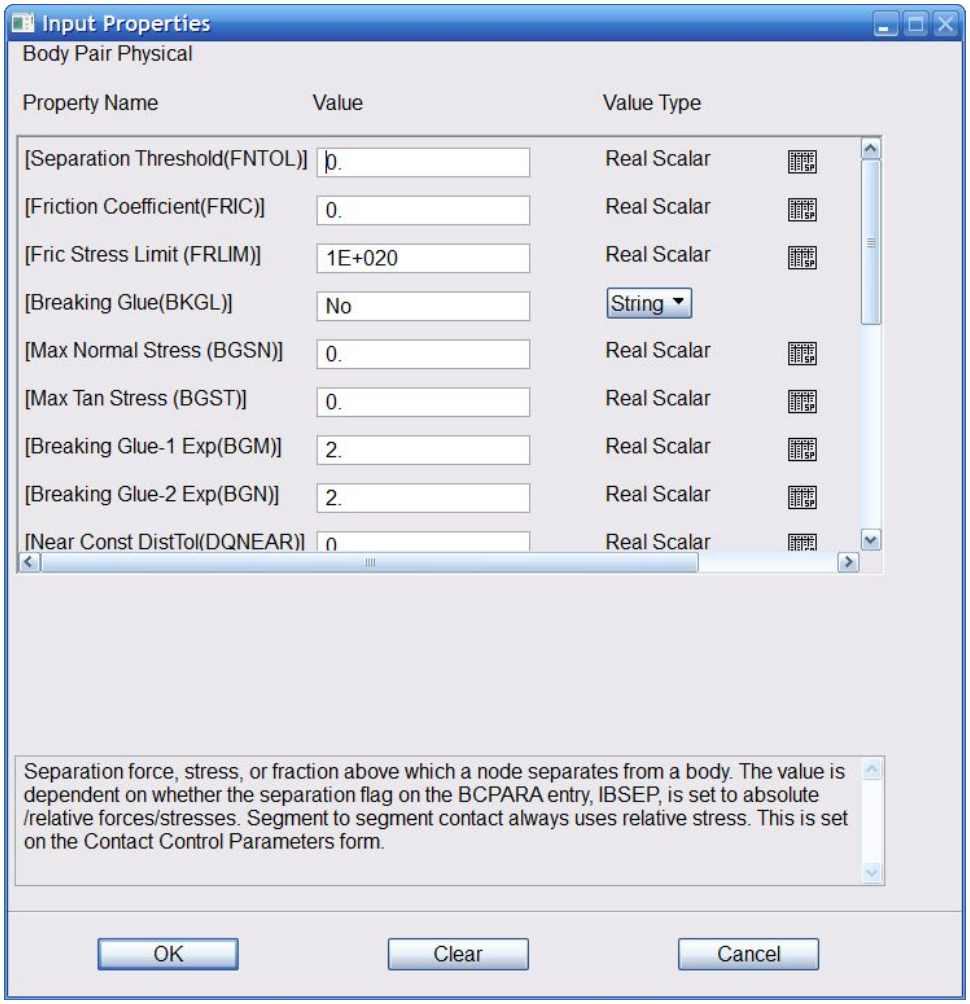
Sets By: Name

- panel_geom_prop
- panel_phy_prop
- rib_geom_prop
- rib_phy_prop
- rib_webs

Filter: *

New Property Set Name: panel_phy_prop

Options: Physical



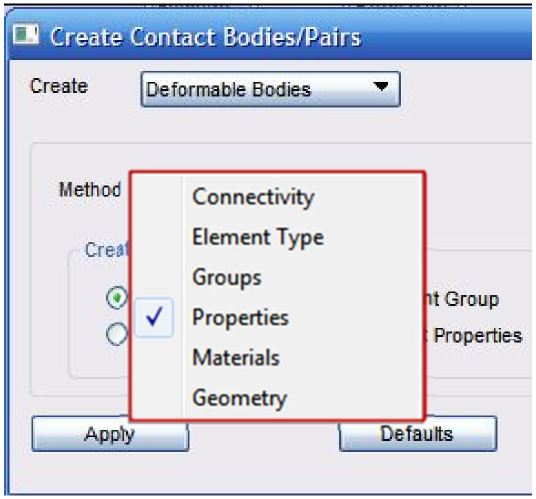
Body Pair Physical

Property Name	Value	Value Type
[Separation Threshold(FNTOL)]	0.	Real Scalar
[Friction Coefficient(FRIC)]	0.	Real Scalar
[Fric Stress Limit (FRLIM)]	1E+020	Real Scalar
[Breaking Glue(BKGL)]	No	String
[Max Normal Stress (BGSN)]	0.	Real Scalar
[Max Tan Stress (BGST)]	0.	Real Scalar
[Breaking Glue-1 Exp(BGM)]	2.	Real Scalar
[Breaking Glue-2 Exp(BGN)]	2.	Real Scalar
[Near Const DistTol(DQNEAR)]	0	Real Scalar

Separation force, stress, or fraction above which a node separates from a body. The value is dependent on whether the separation flag on the BCPARA entry, IBSEP, is set to absolute /relative forces/stresses. Segment to segment contact always uses relative stress. This is set on the Contact Control Parameters form.

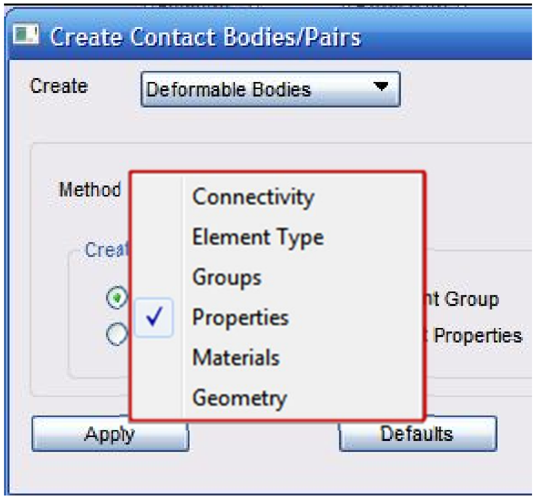
RECOMMENDED WORKFLOW FOR USING CONTACT BODIES / PAIRS

- **Auto-create Contact Bodies**
  - – Tools
  - – Modeling
  - – Contact Bodies/Pairs
  - – Create – Deformable Bodies



RECOMMENDED WORKFLOW FOR USING CONTACT BODIES / PAIRS

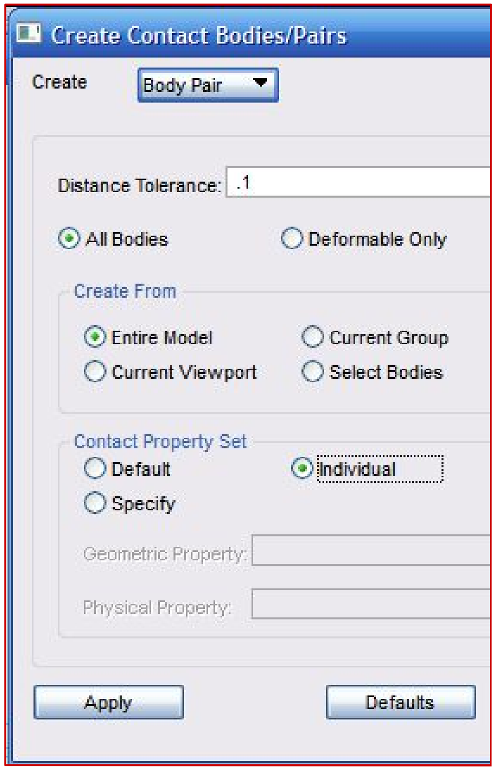
- **Auto-create Contact Bodies**
  - Tools
  - Modeling
  - Contact Bodies/Pairs
  - Create – Deformable Bodies



Create: Deformable Bodies

RECOMMENDED WORKFLOW FOR USING CONTACT BODIES / PAIRS

- **Auto Create Contact Pairs**
  - By group if model is complex
  - Using appropriate distance tolerance
  - Using individual property sets



Create: Body Pair

Distance Tolerance: .1

● All Bodies ○ Deformable Only

Create From

● Entire Model ○ Current Group
○ Current Viewport ○ Select Bodies

Contact Property Set

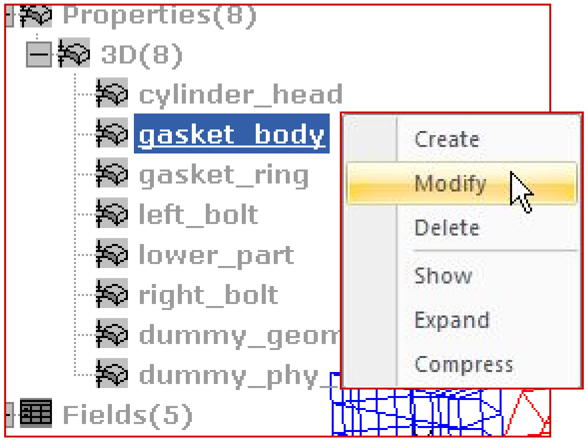
○ Default ● Individual
○ Specify

Geometric Property:

Physical Property:

RECOMMENDED WORKFLOW FOR USING CONTACT BODIES / PAIRS

- **Explore Pairs Created**
  - Use model tree
  - Right click on pair name in tree
  - Select “Plot Contours” – this brings up the form with pair pre-selected
  - Hit Apply of form to color-code/ID the pair
  - If it’s one you want right click on pair in tree and select “Modify” if you want to specify non-default contact interaction
  - If it’s NOT one you want select “Delete”



IMPLEMENTATION DETAILS

- When you create a rigid contact body Patran creates an "invisible" property set in the background.
- This property set contains the control data for the rigid body such as velocity/position/force-moment type, etc.
- A similar property set could be used to describe the interaction data of the contact pair.
- The advantage of this approach is that the user could re-use the set for creating subsequent contact pairs
- This would also allow the user of the "global property editor" to change a large batch of contact interaction sets all at one time.

IMPLEMENTATION DETAILS

- **You will NOT be able to "mix-n-match" contact pair with contact table use, although it should be possible to create a utility to map contact interactions back and forth**
- **Method of operation is as follows:**
  - If any contact pair exists in the data base, contact pair contact will be assumed (and the deck written using BCONECT)
  - If no contact pairs exist in the database, multi-body contact will be assumed (and the deck written using BCTABLE)
  - There will be a toggle labeled "Use Contact Table" or "Ignore existing contact pairs" that will over-ride the default logic and tell the translator to use multi-body contact, and ignore existing pairs (so they can be used later if desired)

APPENDIX B

NONLINEAR TRANSIENT DYNAMICS

REVIEW OF TRANSIENT ANALYSIS

- **Static analysis:**
  - Compute a solution U that satisfies the equilibrium equation: All follower effects on right hand side.

$$\mathbf{F(U) = P}$$

- **Transient analysis:**
  - Compute a solution U that satisfies the equilibrium equation:

$$\underset{\text{Inertia Forces}}{I(\ddot{U},t)} + \underset{\text{Damping Forces}}{D(\dot{U},t)} + \underset{\text{Element Forces}}{F(U,t)} = \underset{\text{External Load}}{P(t,U)}$$

REVIEW OF TRANSIENT ANALYSIS

- **For a linear system:**

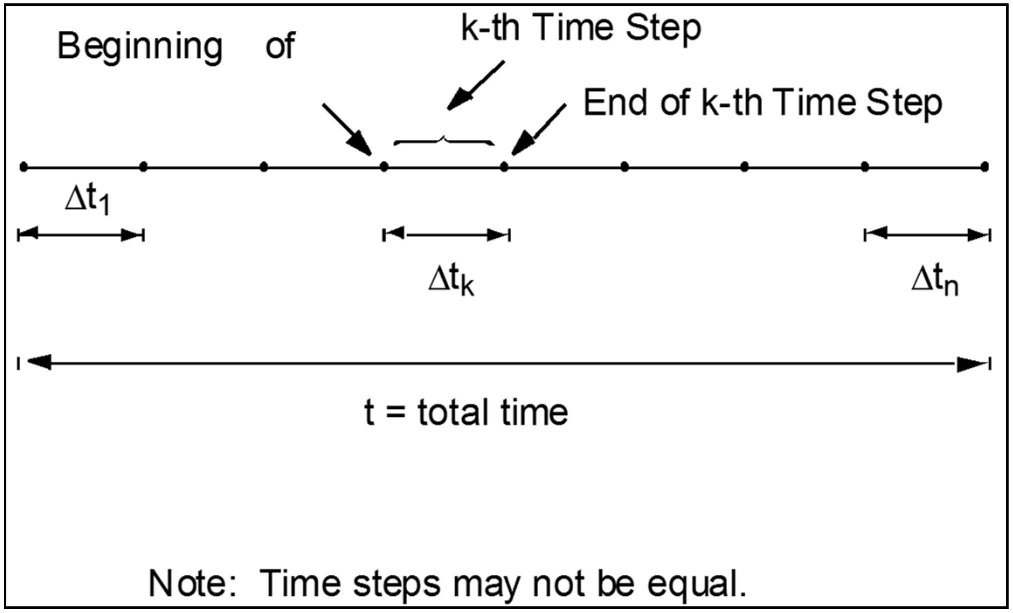
$$M\ddot{U} + B\dot{U} + KU = P(t)$$

- **For a general nonlinear system**
  - Mass of the system may change
  - Damping may change
  - Stiffness may change
  - Load may be function of system response
  - In MSC Nastran mass and damping cannot change (except for CBUSH1D). The equilibrium equation is

$$M\ddot{U}(t) + [\mathrm{B}\dot{\mathrm{U}}(\mathrm{t}) + \mathrm{F}_{\mathrm{bush1d}}(\dot{\mathrm{U}}(\mathrm{t}))] + \mathrm{K}(\mathrm{U}(\mathrm{t})) = \mathrm{P}(\mathrm{t}, \mathrm{U})$$

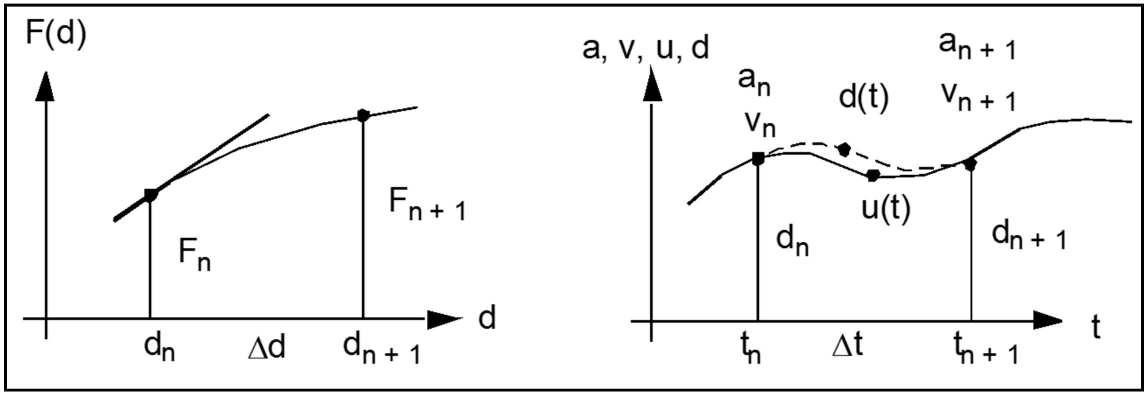
REVIEW OF TRANSIENT ANALYSIS

- **Nonlinear Transient Analysis**
  - Nonlinear transient analysis proceeds by dividing the time into a number of small time steps. (Finite difference solution)



REVIEW OF TRANSIENT ANALYSIS

- **The solution at the end of a time step provides the initial conditions for the next time step.**
- **For each time step, a relationship is assumed between displacement, velocity, and acceleration (integration scheme).**



u_n	Displacement at time t_n approximated by d_n.
$\dot{u}_n$	Velocity at time t_n approximated by v_n.
$\ddot{u}_n$	Acceleration at time t_n approximated by a_n.

REVIEW OF TRANSIENT ANALYSIS

- **There are a number of different integration schemes available in the literature.**
  - Implicit integration: d_{n+1} is obtained by using the equilibrium conditions at time t_{n+1}. (SOL 400)
  - Explicit integration: d_{n+1} is obtained by using the equilibrium conditions at time t_n. (SOL 700)
- **Use of the integration scheme reduces the transient equilibrium equation to a static equilibrium equation form.**

$$\underbrace{K^*(M, B, K, \Delta t)\,\Delta U}_{\text{Effective Dynamics Stiffness}} \qquad = \qquad \underbrace{P^*(\Delta t, \dot{\mathring{U}}, \ddot{U}, M, B, \Delta P)}_{\text{Effective Dynamic Load Vector}}$$

- **Effective dynamic stiffness and load vector depend on the integration scheme used.**
- **For example, for the average acceleration scheme, also called the trapezoidal rule or Newmark scheme ($\gamma = 1/2$, $\beta = 1/4$)… (see next slide)**

REVIEW OF TRANSIENT ANALYSIS

$$\boldsymbol{K}^* = \frac{4}{\Delta \boldsymbol{t}^2}\boldsymbol{M} + \frac{2}{\Delta \boldsymbol{t}}\boldsymbol{B} + \boldsymbol{K}$$

$$P^* = \Delta P(t) + M[2\ddot{U}(t) + \frac{4}{\Delta t}\dot{U}(t)] + 2B\dot{U}(t)$$

- The equilibrium is satisfied at the beginning and at the end of a time step.
- The equilibrium is not satisfied within the time step. Therefore, the selection of Δt is important.
- A large value of Δt reduces accuracy.
- A small value of Δt increases computing cost.
- A strategy is needed that automatically adjusts the time step value to achieve an optimum value in terms of accuracy and computing cost.
- Adjustment of time step value requires the reformation and decomposition of the dynamic stiffness.

USER INTERFACE

- **Solution sequences**
  - SOL 400
- **Solution strategy**
  - TSTEPNL – Defines parameters and data for nonlinear transient.
  - NLSTEP – New time stepping entry (MSC Nastran 2010).
- **Mass specification**
  - RHO field in MATi Bulk Data entries.
  - CMASSi Bulk Data entries for scalar mass elements.
  - CONMi Bulk Data entries for concentrated mass elements.
  - PARAM,COUPMASS, to specify the generation of coupled rather than lumped mass matrices for elements with coupled mass capability.
  - PARAM,WTMASS.

USER INTERFACE

- **Damping specification**
  - CVISC Bulk Data entry for the viscous damper element
  - Field GE in MATi Bulk Data entries for nonlinear element damping
  - PARAM, G for overall structural damping

USER INTERFACE

- **Damping specification**
  - PARAM, W3 to convert structural damping to equivalent viscous damping
  - PARAM, W4 to convert element damping to equivalent viscous damping
  - PARAM, NDAMP (.01 default) to specify numerical damping
  - Most general damping (non CBUSH1D):

$$B = B^1 + B^{2gg} + \frac{g}{w_3}(K^1 + K^{2gg}) + \frac{1}{w_4}K^4 + \alpha_1 M + \alpha_2 (K^1 + K^{2gg}) + BH_{(\text{if componentmodes})}$$

$$K^4 = \sum g_e K_e$$

USER INTERFACE

- **Load Specification**

TLOAD1	Transient load as ordered time, force pairs.
TLOAD2	Transient load as defined by analytical functions.
DAREA	Transient load scale factors.
LSEQ	Generate transient load history for static loads.
NOLIN1	Nonlinear transient load as a tabular function.
NOLIN2	Nonlinear transient load as the product of two variables.
NOLIN3	Nonlinear transient load as a positive variable raised to a power.
NOLIN4	Nonlinear transient load as a negative variable raised to a power.

  - Selected by DLOAD, LOADSET, and NONLINEAR Case Control commands.

USER INTERFACE

- **Initial conditions specification**
  - TIC Bulk Data entry
  - IC Case Control command
- **Additional entries for nonlinear analysis**
  - Similar to nonlinear static analysis
  - Material nonlinear
    - MATS1, MATEP, MATHE …
  - Geometric nonlinear
    - PARAM,LGDISP,+1

USER INTERFACE

- **Additional entries for nonlinear analysis (Cont.)**
  - Contact
    - BCTABLE/BCBODY, BCONECT/BCBODY1
  - Combined material and geometric nonlinear

EXAMPLE INPUT FOR SOL 400

```
SOL 400
CEND
TITLE = MD Nastran pipe job
IC = 50
SUBCASE 1
 STEP 1
   TITLE=This is a default subcase.
   ANALYSIS = NLTRAN
   DLOAD = 100
   TSTEPNL = 1
   DISPLACEMENT(SORT1,REAL)=ALL
   SPCFORCES(SORT1,REAL)=ALL
   STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
BEGIN BULK
PARAM   LGDISP  1
$
TSTEPNL, 1,800, 1.E-6, 1, PFNT, , ,UV
$ Elements and Element Properties for region : 2D_1
PLPLANE 1       1
PSHLN2  1       1
        C4      AXSOLID L
$ Pset: "2D_1" will be imported as: "plplane.1"
CQUADX  1       1       8       7       1       5
CQUADX  2       1       9       8       5       6
....
....
NLMOPTS LRGSTRN 1
MATEP   1       Table           1               Isotrop Addmean
MAT1    1      1.+11           .3      7000.
$ Stress/Strain Curve : Field_StressStrain_1
TABLES1 1       2
        0.     1.+8    1.      4.00+8  ENDT
$ Nodes of the Entire Model
GRID    1               .019345 0.      0.
GRID    2               .019345 .08     0.
....
$ LOAD ENTRIES
TLOAD2,100,10,,0,0.0,10.0,1.0
DAREA,10,15,1,10.0
$ INITIAL CONDITIONS
TIC,50,5,1,1.0,-2.0
$
ENDDATA
```

GENERAL FEATURES

- Transient material nonlinear, geometric nonlinear, combined geometric and material nonlinear, and contact problems can be solved using this solution sequence
- Linear superelements can be combined with nonlinear elements
- Component Modal reduction (SEQSET,EIGR) available for the linear superelements

INTEGRATION SCHEMES

- **Two-Point Integration Scheme**
  - Use the following equilibrium equation:

$$M\,\ddot{\mathrm{U}}_{\mathrm{n+1}} + B\,\dot{\mathrm{U}}_{\mathrm{n+1}} + F_{n+1} = P_{n+1}$$

  - Assume that the acceleration for a time step is equal to the average of the beginning and end of the step.

$$\ddot{\mathrm{U}}(\mathrm{t}) = \frac{\ddot{U}_n + \ddot{U}_{n+1}}{2}$$

  - Velocity and displacement are obtained by integration.

$$\dot{U}_{n+1} = \dot{U}_n + \left(\frac{\ddot{U}_n + \ddot{U}_{n+1}}{2}\right)\Delta t$$

$$U_{n+1} = U_n + \dot{U}_n \Delta t + \left(\frac{\ddot{U}_n + \ddot{U}_{n+1}}{4}\right)\Delta t^2$$

INTEGRATION SCHEMES

- Rearrange the equilibrium equation in terms of incremental values.

$$\left[\frac{4}{\Delta t^2}M+\frac{2}{\Delta t}B+K_T\right]\Delta U=P_{n+1}+P_n-F_n+\frac{4}{\Delta t}M\,\dot{U}_n-\left[\frac{4}{\Delta t^2}M+\frac{2}{\Delta t}C\right][U_{n+1}-U_n]$$

Dynamic Stiffness

Dynamic Load Factor

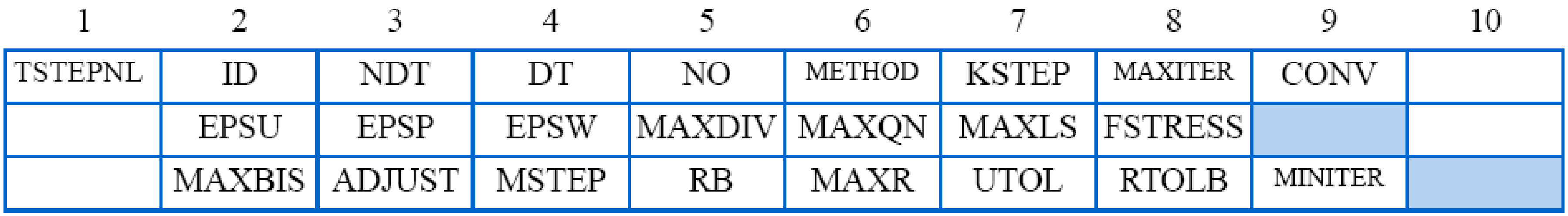
- Calculate velocity as follows:

$$\dot{U}_{n+1}=(U_{n+1}-U_n)\frac{2}{\Delta t}-\dot{U}_n$$

- Note that the acceleration need not be calculated since it does not appear in the incremental equilibrium equation.
- For postprocessing purposes, acceleration is calculated as:

$$\{\ddot{U}_n\}=\frac{1}{\Delta t_n+\Delta T_{n+1}}\left[\frac{\Delta t_n}{\Delta t_{n+1}}\{\ddot{U}_{n-1}\}+\left(\frac{\Delta t_{n+1}}{\Delta t_n}-\frac{\Delta t_n}{\Delta t_{n+1}}\right)\{\dot{U}_n\}-\frac{\Delta t_{n+1}}{\Delta t_n}\{\ddot{U}_{n-1}\}\right]$$

TSTEPNL FOR NONLINEAR TRANSIENT



1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT	NO	METHOD	KSTEP	MAXITER	CONV	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS		
	MAXBIS	ADJUST	MSTEP	RB	MAXR	UTOL	RTOLB	MINITER	

- HHT (Houbolt, Hughes, and Taylor) time integration scheme is used. It has better numerical damping than the Newmark-Beta.
- NDAMP (for B, F and P) set to 0.0 (i.e., Default, for contact)
- NDAMPM (for M) set to 1.0 (i.e., Default, for contact)

The above default settings for NDAMP, and NDAMPM results in use of the Single Step Houbolt Operator, and is recommended for problems with contact.

- NDAMP (for B, F and P) set to -0.05 (i.e., Default, for non-contact)
- NDAMPM (for M) set to 0.0 (i.e., Default, for non-contact)

M, B, F, and P are respectively the mass, damping, internal force, and external load.

GENERALIZED-ALPHA SCHEME

- **Existing HHT scheme has been extended to Generalized-Alpha Scheme**
  - Generalized-Alpha Scheme is a two-parameter scheme that allows spectral radius to vary between 0.0 and 1.

$$\boxed{M\ddot{u}_{n+1+\alpha_m} + C\dot{u}_{n+1+\alpha_f} + F^{int}_{n+1+\alpha_f} = F^{ext}_{n+1+\alpha_f}}$$

- **No Special input needed to flag new scheme**
  - For contact / impact problems, SOL400 by default uses $\alpha_f = 0.0$, $\alpha_m = 1.0$ (this corresponds to spectral radius of 0.0).
  - For non-contact problems, SOL400 continues to use $\alpha_f = -0.05$, $\alpha_m = 0.0$ (this ensures back-ward compatibility)

GENERALIZED-ALPHA SCHEME

- **α_f and α_m can be manually changed by user via PARAM,NDAMP,xxx and PARAM,NDAMPM,yyy respectively in the Case Control or Bulk Data sections.**
- **Valid range of α_f is [-0.5,0.0]. Valid range of α_m is [0.0,1.0]. A variety of time-stepping schemes are obtained by varying α_f and α_m**
  - $\alpha_f = 0.0$ $\alpha_m = 0.0$ Newmark-Beta
  - $\alpha_f = -0.05$ $\alpha_m = 0.0$ HHT (Default for non-contact)
  - $\alpha_f = 0.0$ $\alpha_m = 1.0$ Single-Step Houbolt (Default for contact)

TSTEPNL – TIME STEP AND RESULTS

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT	NO					

- **DT, NDT – time increment and how many**
- **NO – number of increments after which output is desired**
  - – if positive, the output locations are controlled by DT. Default=1.
  - – if negative, output will be performed after NO actual computed increments (SOL 129 style)

TSTEPNL – RESULTS OUTPUT (CONT.)

- **Additional User Control over Output using OTIME. Example:**

```
OTIME = 99
SET 99 = 0.025, 0.035
$
SUBCASE 10
  STEP 1
         LOAD = 1
         NLPARM = 110
  STEP 2
         ANALYSIS = NLTRAN
         DLOAD = 3
         TSTEPNL = 130
BEGIN BULK
$.......2.......3.......4.......5.......6.......7.......8.......9.......0
TSTEPNL 130     100     0.01    10      AUTO
```

additional output

“NO” field, output for t = 0, 1*10*0.01, 2*10*0.01, …

TSTEPNL – RESULTS OUTPUT (CONT.)

- **The Data grouping option allows to minimize disk space usage (especially important for Transient because of many increments).**
  - PARAM, NLPACK, n
    - N=-1 – output data as requested, but the restart data are stored only for the end of a STEP. So, restart is possible from end of step only.
    - N=1 – each package of data on the database includes the output data for one output time step and the corresponding restart data. Restart can be done from each output time step.

TSTEPNL – STIFFNESS UPDATE

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID				METHOD	KSTEP			
					MAXQN	MAXLS			
								MINITER	

- **METHOD (default=AUTO, for contact: FNT)**
  - ADAPT – Automatic adjustment of the time steps. Stiffness updates for every KSTEPth converged bisection solution.
  - AUTO, SEMI – Automatic stiffness updates to improve convergence. Automatic adjustment of time steps. SEMI = stiffness update after the first iteration.
  - ITER – Stiffness update at every KSTEPth iteration. Automatic adjustment of time steps.

TSTEPNL – STIFFNESS UPDATE (CONT.)

- FNT – Full Newton iteration. Stiffness update at every iteration. Defaults of EPSU, EPSP and EPSW = 0.01 and MAXLS=2. Default for contact.
- PFNT – Pure Full Newton iteration. Same as FNT except that the defaults for EPSU and EPSW = -0.01, EPSP=0.01 and MAXLS=0.
- For FNT and PFNT, whether the stiffness will be updated upon convergence and the start of the next increment, depends on KSTEP field:
- KSTEP="BLANK" – decided by SOL 400 based on element type. Default.
- KSTEP= 1 – stiffness will not be updated
- KSTEP=-1 – stiffness will always be updated

TSTEPNL – STIFFNESS UPDATE (CONT.)

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID				METHOD	KSTEP			
					MAXQN	MAXLS			
								MINITER	

- **MAXQN (Default=10)**
  - number of Quasi Newton Vectors for which an exact stiffness update takes place. Default=10.
- **MAXLS (Default=2)**
  - maximum number of line searches. For PFNT, Default=0.
- **MINITER (Default=1)**
  - minimum number of iterations. Default for Contact=2.

TSTEPNL – CONVERGENCE

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID							CONV	
	EPSU	EPSP	EPSW						

- **Same as NLPARM**

TSTEPNL – DIVERGENCE

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID						MAXITER		
				MAXDIV			FSTRESS		
	MAXBIS				MAXR		RTOLB		

- **Same as NLPARM**

TSTEPNL – ADAPTIVE TIME STEPPING

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT						
		ADJUST	MSTEP	RB	MAXR	UTOL			

- **Implemented for METHOD=ADAPT, AUTO, SEMI, ITER, FNT and PFNT**
- **Controlled via ADJUST and related parameters**
- **Time step will be adjusted to T / MSTEP whereby T is the dominant period of response**

$$\lambda_n = \frac{\Delta u^T_n \, K \, \Delta u_n}{\Delta u^T_n \, M \, \Delta u_n} \longrightarrow T = \frac{2\pi}{\mathrm{sqrt}(\lambda)}$$

TSTEPNL – ADAPTIVE STEPPING (CONT.)

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT						
		ADJUST	MSTEP	RB	MAXR	UTOL			

- **To prevent thrashing a stepping function is being applied**

$$\Delta t_{n+1} = f(r)\Delta t_n$$

$$\text{with}\quad \begin{array}{lll} f = 0.25 & \text{for} & r < 0.5 * RB \\ f = 0.5 & \text{for} & 0.5 < RB \le r < RB \\ f = 1.0 & \text{for} & RB \le r < 2.0 \\ f = 2.0 & \text{for} & 2.0 \le r < 3.0/RB \\ f = 4.0 & \text{for} & r \ge 3.0/RB \end{array}$$

Bounds on Δt: $\mathrm{MIN}\left(\frac{DT}{2^{MAXBIS}}, \frac{DT}{MAXR}\right) < \Delta t < MAXR * DT$

TSTEPNL – ADAPTIVE STEPPING (CONT.)

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT						
		ADJUST	MSTEP	RB	MAXR	UTOL			

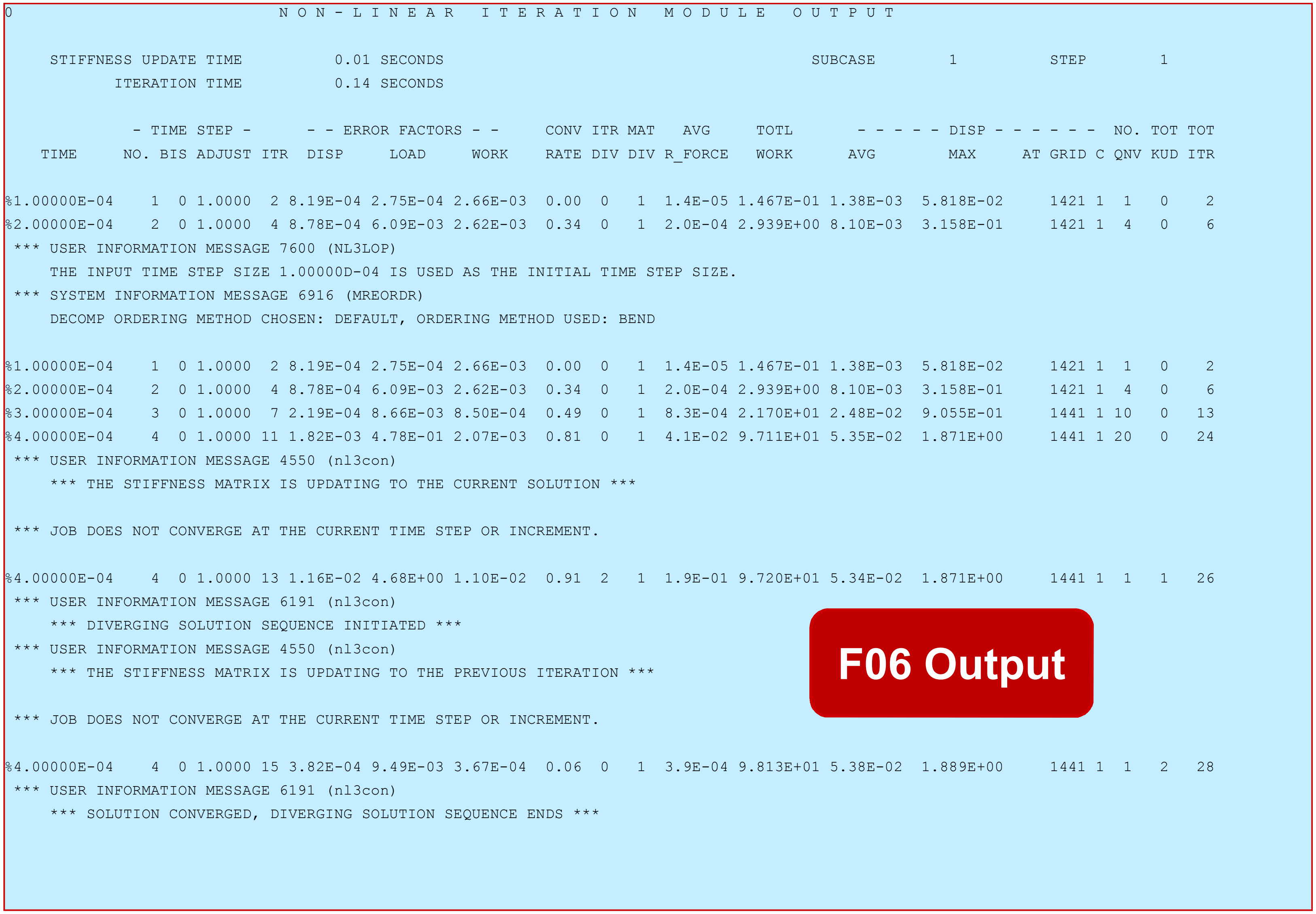
- **UTOL, tolerance on velocities**
  - If the velocities are low time step will not be adjusted
- **To switch Adaptive Time Stepping off, just set ADJUST to zero**

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT	NO	METHOD	KSTEP	MAXITER	CONV	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS		
	MAXBIS	ADJUST	MSTEP	RB	MAXR	UTOL	RTOLB	MINITER	

TSTEPNL – RECOMMENDATIONS

- **For the advanced nonlinear elements and materials and Contact Analysis FNT or PFNT with CONV=PV should be used. Default.**
- **In all other cases (excluding GAPs) AUTO and CONV=UPW can be used. For GAPs use ITER with KSTEP=1.**
- **For contact with friction FNT and UPW has been found to be more precise**
- **The PFNT method is more stringent than the FNT. However if CONV=UPW is used, both methods give similar results.**

OUTPUT – NONLINEAR ITERATION



```
0                             N O N - L I N E A R   I T E R A T I O N   M O D U L E   O U T P U T


     STIFFNESS UPDATE TIME          0.01 SECONDS                                         SUBCASE         1          STEP         1
            ITERATION TIME          0.14 SECONDS


              - TIME STEP -        - - ERROR FACTORS - -      CONV ITR MAT   AVG      TOTL       - - - - - DISP - - - - - -   NO. TOT TOT
    TIME      NO. BIS ADJUST ITR  DISP      LOAD      WORK    RATE DIV DIV R_FORCE   WORK       AVG         MAX      AT GRID C QNV KUD ITR

%1.00000E-04    1  0 1.0000  2 8.19E-04 2.75E-04 2.66E-03  0.00  0   1  1.4E-05 1.467E-01 1.38E-03  5.818E-02     1421 1  1   0   2
%2.00000E-04    2  0 1.0000  4 8.78E-04 6.09E-03 2.62E-03  0.34  0   1  2.0E-04 2.939E+00 8.10E-03  3.158E-01     1421 1  4   0   6
 *** USER INFORMATION MESSAGE 7600 (NL3LOP)
     THE INPUT TIME STEP SIZE 1.00000D-04 IS USED AS THE INITIAL TIME STEP SIZE.
 *** SYSTEM INFORMATION MESSAGE 6916 (MREORDR)
     DECOMP ORDERING METHOD CHOSEN: DEFAULT, ORDERING METHOD USED: BEND

%1.00000E-04    1  0 1.0000  2 8.19E-04 2.75E-04 2.66E-03  0.00  0   1  1.4E-05 1.467E-01 1.38E-03  5.818E-02     1421 1  1   0   2
%2.00000E-04    2  0 1.0000  4 8.78E-04 6.09E-03 2.62E-03  0.34  0   1  2.0E-04 2.939E+00 8.10E-03  3.158E-01     1421 1  4   0   6
%3.00000E-04    3  0 1.0000  7 2.19E-04 8.66E-03 8.50E-04  0.49  0   1  8.3E-04 2.170E+01 2.48E-02  9.055E-01     1441 1 10   0  13
%4.00000E-04    4  0 1.0000 11 1.82E-03 4.78E-01 2.07E-03  0.81  0   1  4.1E-02 9.711E+01 5.35E-02  1.871E+00     1441 1 20   0  24
 *** USER INFORMATION MESSAGE 4550 (nl3con)
     *** THE STIFFNESS MATRIX IS UPDATING TO THE CURRENT SOLUTION ***

 *** JOB DOES NOT CONVERGE AT THE CURRENT TIME STEP OR INCREMENT.

%4.00000E-04    4  0 1.0000 13 1.16E-02 4.68E+00 1.10E-02  0.91  2   1  1.9E-01 9.720E+01 5.34E-02  1.871E+00     1441 1  1   1  26
 *** USER INFORMATION MESSAGE 6191 (nl3con)
     *** DIVERGING SOLUTION SEQUENCE INITIATED ***
 *** USER INFORMATION MESSAGE 4550 (nl3con)
     *** THE STIFFNESS MATRIX IS UPDATING TO THE PREVIOUS ITERATION ***

 *** JOB DOES NOT CONVERGE AT THE CURRENT TIME STEP OR INCREMENT.

%4.00000E-04    4  0 1.0000 15 3.82E-04 9.49E-03 3.67E-04  0.06  0   1  3.9E-04 9.813E+01 5.38E-02  1.889E+00     1441 1  1   2  28
 *** USER INFORMATION MESSAGE 6191 (nl3con)
     *** SOLUTION CONVERGED, DIVERGING SOLUTION SEQUENCE ENDS ***
```

F06 Output

OUTPUT NON-LINEAR ITERATION

```
0                             N O N - L I N E A R   I T E R A T I O N   M O D U L E   O U T P U T

     STIFFNESS UPDATE TIME          0.01  SECONDS                                         SUBCASE        1          STEP        1
            ITERATION TIME          0.14  SECONDS

                 - TIME STEP -       - - ERROR FACTORS - -      CONV  ITR MAT   AVG      TOTL         - - - - - DISP - - - - - -   NO.  TOT  TOT
    TIME       NO. BIS ADJUST ITR  DISP      LOAD      WORK      RATE DIV DIV R_FORCE   WORK       AVG        MAX      AT GRID C QNV KUD ITR

%1.00000E-04     1   0 1.0000   2 8.19E-04 2.75E-04 2.66E-03  0.00   0    1  1.4E-05 1.467E-01 1.38E-03  5.818E-02     1421 1  1    0    2
```

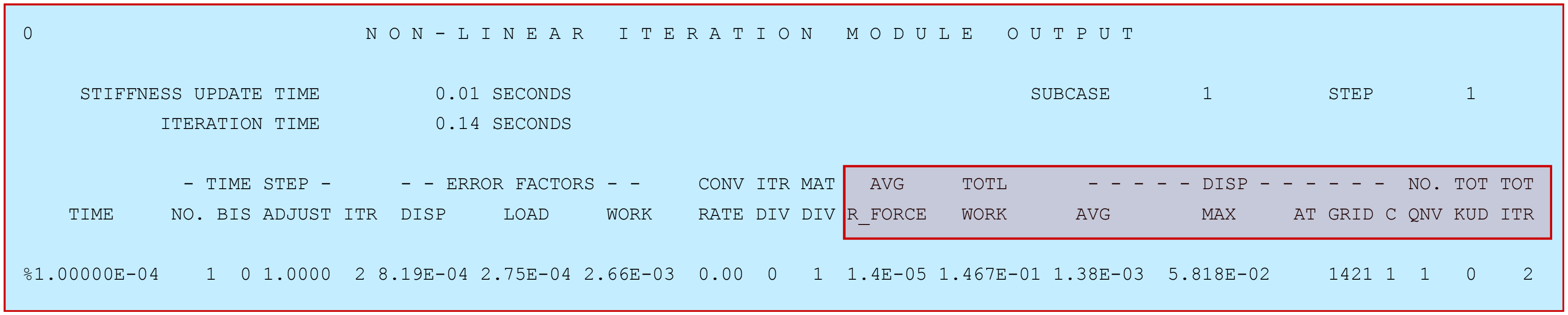
- **TIME**
  - current time
- **TIME STEP NO.**
  - time step number. Initialized to 0 at the beginning of each STEP.
- **TIME STEP BIS**
  - number of bisections performed
- **TIME STEP ADJUST**
  - ratio of the current time increment to the original DT
- **TIME STEP ITR**
  - number of iterations within the time increment

OUTPUT NON-LINEAR ITERATION

```
0                              N O N - L I N E A R   I T E R A T I O N   M O D U L E   O U T P U T

     STIFFNESS UPDATE TIME          0.01 SECONDS                                    SUBCASE        1          STEP         1
            ITERATION TIME          0.14 SECONDS

              - TIME STEP -      - - ERROR FACTORS - -       CONV ITR MAT  AVG     TOTL        - - - - - DISP - - - - - -  NO. TOT TOT
    TIME      NO. BIS ADJUST ITR  DISP      LOAD      WORK     RATE DIV DIV R_FORCE  WORK      AVG        MAX       AT GRID C QNV KUD ITR

%1.00000E-04    1   0 1.0000   2 8.19E-04 2.75E-04 2.66E-03  0.00   0    1  1.4E-05 1.467E-01 1.38E-03  5.818E-02     1421 1  1   0    2
```

- **DISP LOAD WORK**
  - displacement, load and energy errors, must be smaller than the tolerances EPSU, EPSP and EPSW
- **CONV RATE**
  - should be between 0 and 1, bigger than 1 means the solution will never converge
- **ITR DIV**
  - divergence counter, > MAXDIV triggers the divergence process
- **MAT DIV**
  - divergence counter for element and material routines

OUTPUT – NONLINEAR ITERATION



```
0                             N O N - L I N E A R   I T E R A T I O N   M O D U L E   O U T P U T

     STIFFNESS UPDATE TIME          0.01 SECONDS                                         SUBCASE        1          STEP        1
            ITERATION TIME          0.14 SECONDS

               - TIME STEP -      - - ERROR FACTORS - -    CONV ITR MAT   AVG     TOTL        - - - - - DISP - - - - - -  NO. TOT TOT
    TIME     NO. BIS ADJUST ITR  DISP     LOAD     WORK    RATE DIV DIV R_FORCE   WORK      AVG         MAX     AT GRID C QNV KUD ITR

%1.00000E-04    1  0 1.0000  2 8.19E-04 2.75E-04 2.66E-03  0.00  0   1  1.4E-05 1.467E-01 1.38E-03   5.818E-02     1421 1  1   0    2
```

- **AVG R_FORCE**
  - – average residual force, should be small
- **TOTL WORK**
  - – approximate total energy
- **DISP: AVG MAX AT GRID C**
  - – average, maximum displacement at grid in direction c
- **NO. QNV**
  - – Number of Quasi Newton Vectors
- **TOT KUD / ITR**
  - – Total of stiffness updates / iterations

MASS SPECIFICATION

- Similar to linear transient analysis.
- CMASS1 and CMASS2 define scalar mass elements.
- CMASS3 and CMASS4 define scalar mass elements connected only to scalar points.
- CONM1 defines a 6 x 6 mass matrix for a grid point.
- CONM2 defines a diagonal mass matrix for translational degrees of freedom and a 3 x 3 full matrix for rotational degrees of freedom at a grid point.
- Element mass density is defined on the RHO field of the MATi Bulk Data entry.
- PARAM,COUPMASS,1 specifies the coupled mass matrix for elements with coupled mass capability (BAR, BEAM, ROD, HEXA, PENTA, TRIA, and TUBE elements).
- Non structural mass PSHELL, PBEAM, etc.

DAMPING

- **Damping represents energy dissipation observed in structures.**
- **Difficult to accurately model since damping results from many mechanisms:**
  - Viscous effects (dashpot, shock absorber)
  - External friction (slippage in structural joints)
  - Internal friction (characteristic of material type)
  - Structural nonlinearities (plasticity)
- **Analytical conveniences are used to model damping.**
- **Viscous damping force proportional to velocity**

$$f_v = b\dot{u}$$

$$m\ddot{u} + b\dot{u} + ku = p$$

DAMPING

- **Structural damping force proportional to displacement**

$$f_s = i\,Gk\,u \qquad i = \sqrt{-1}$$

$$m\ddot{u} + (1 + iG)ku = p \qquad \mathrm{G} = \text{structural damping coefficient}$$

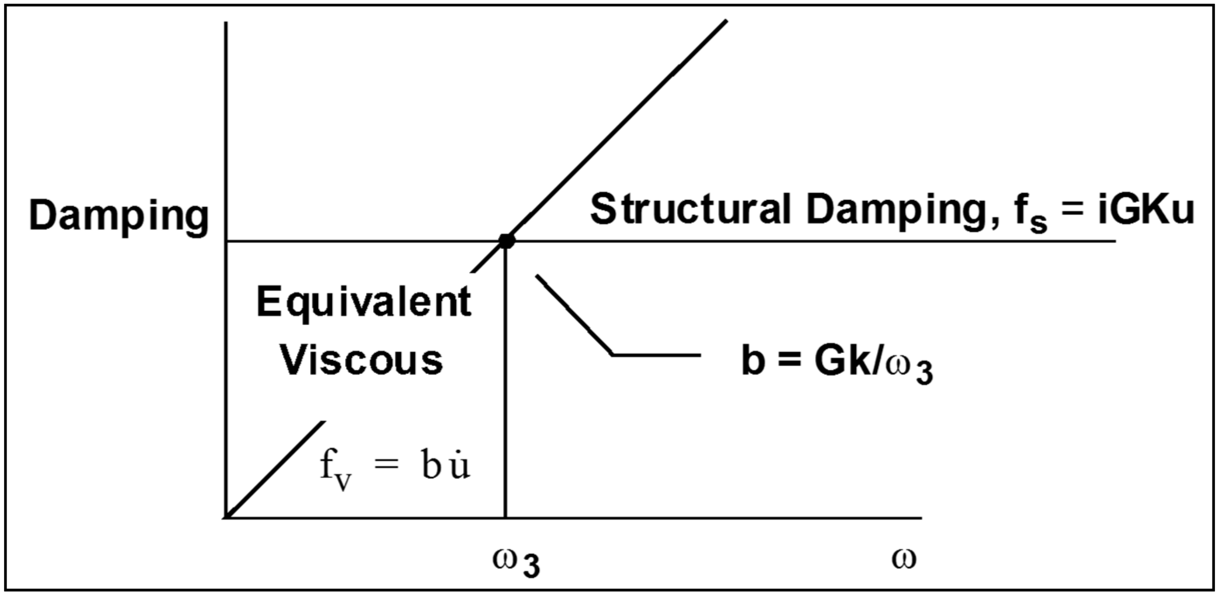
- **Viscous and structural damping are equivalent at frequency ω_3.**

with

$$\mathrm{G} = \frac{\mathrm{b}\omega_3}{\mathrm{k}} = \frac{2\xi\omega_3}{\omega_n}$$

$$\xi = \frac{\mathrm{c}}{2\mathrm{m}\omega_n}$$

DAMPING


Damping
Structural Damping, $f_s = iGKu$
Equivalent Viscous
$b = Gk/\omega_3$
$f_v = b\dot{u}$
ω_3
ω

DAMPING SPECIFICATION

- **Similar to linear transient analysis.**
- **Damping matrix B comprised of several matrices:**

$$B = B^1 + B^{2gg} + \frac{g}{w_3}(K^1 + K^{2gg}) + \frac{1}{w_4}\sum g_e K_e + F_{CBUSH1D}(\dot{U}, t)$$

– Where B^1 = damping elements (VISC,DAMP)
G = overall structural damping coefficient (PARAM,G)
W_3 = frequency of interest - rad/sec (PARAM,W3)
K^1 = global stiffness matrix
G_e = element structural damping coefficient (GE on the MATi entry)
W_4 = frequency of interest - rad/sec (PARAM,W4
K_e = element stiffness matrix

DAMPING SPECIFICATION

- **Default values for W_3 and W_4 are 0.0. In this case, the associated damping terms are ignored.**
- **Element damping is provided with PARAM,W4 and field GE in the MATi entry.**
- **Damping matrix is not rotated.**
  - Caution for large rotation.

DAMPING SPECIFICATION

- **DAMPING PARAMETERS**
  - PARAM,G, factor (default = 0.0)
    - Overall structural damping coefficient to multiply stiffness matrix for linear elements.
  - PARAM,W3, factor (default = 0.0)
    - Converts overall structural damping to equivalent viscous damping.
  - PARAM,W4, factor (default = 0.0)
    - Converts element structural damping to equivalent viscous damping.
  - Units for W3, W4 are radians/unit time.
  - If PARAM,G is used; PARAM,W3 must be set to greater than zero or PARAM,G will be ignored.

LOAD SPECIFICATION

- **Three ways:**
  - Dynamic loads
  - Static loads
  - Nonlinear loads

DYNAMIC LOADS

- **Dynamic loads require both temporal and spatial distribution.**
- **A user needs to follow four steps to specify dynamic loads.**
- **The four steps are:**
  1. Define the load as a function of time (TLOADi).
  2. Define the spatial distribution of the load (DAREA, FORCE, etc.).
  3. Combine the TLOADi entries via DLOAD entry.
  4. Select the loads via the DLOAD Case Control command.

DYNAMIC LOADS

- **TLOAD1 Bulk Data Entry**
  - Description: Defines a time-dependent dynamic load or enforced motion of the form

$$\{P(t)\} = \{A * F(t-\tau)\}$$

for use in transient response analysis.

  - Format:

1	2	3	4	5	6	7	8	9	10
TLOAD1	SID	DAREA	DELAY	TYPE	TID				

  - Example:

TLOAD1	5	7			13				

DYNAMIC LOADS

Field	Contents
SID	Set identification number. (Integer > 0).
DAREA	Identification number of DAREA entry set or a thermal load set (in heat transfer analysis) which defines A. (Integer > 0).
DELAY	Identification number of DELAY entry set that defines t. (Integer ≥ 0, or blank).
TYPE	Defines the nature of the dynamic excitation. (Integer 0, 1, 2, 3, or blank).

Integer	**Excitation Function**
0 or blank	Force or Moment
1	Enforced Displacement
2	Enforced Velocity
3	Enforced Acceleration

TID	Identification number of TABLEDi entry that gives F(t-t). (Integer > 0).

DYNAMIC LOADS

- **DAREA Bulk Data Entry**
  - Description: Defines scale (area) factors for dynamic loads. DAREA is used in conjunction with RLOADi and TLOADi entries.
  - Format:

1	2	3	4	5	6	7	8	9	10
DAREA	SID	P1	C1	A1	P2	C2	A2		

  - Example:

DAREA	3	6	2	8.2	15	1	10.1		

Field	Contents
SID	Identification number. (Integer > 0).
Pi	Grid, extra, or scalar point identification number.(Integer > 0).
Ci	Component number. (Integer 1 through 6 for grid point; blank or 0 for extra or scalar point).
Ai	Scale (area) factor. (Real).

DYNAMIC LOADS

- **TLOAD2 Bulk Data Entry**
  - Description: Defines a time-dependent dynamic load or enforced motion of the form

$$\{P(t)\} = \begin{cases} 0 & , \mathrm{t} < (\mathrm{T1} + \tau) \text{ or } \mathrm{t} > (\mathrm{T2} + \tau) \\ A\tilde{t}\,e^{C\tilde{t}} \cos(2\pi F\tilde{t} + P) & , (\mathrm{T1} + \tau) \leq \mathrm{t} \leq (\mathrm{T2} + \tau) \end{cases}$$

for use in a transient response problem where $\tilde{t}$ = t - T1 - t.

  - Format:

1	2	3	4	5	6	7	8	9	10
TLOAD2	SID	DAREA	DELAY	TYPE	T1	T2	F	P	
	C	B							

  - Example:

TLOAD2	4	10			2.1	4.7	12		
	2								

DYNAMIC LOADS

Field	Contents
SID	Set identification number. (Integer > 0).
DAREA	Identification number of DAREA entry set or a thermal load set (in heat transfer analysis) that defines A. (Integer > 0).
DELAY	Identification number of DELAY entry set that defines t. (Integer ≥ 0, or blank).
TYPE	Defines the nature of the dynamic excitation. (Integer 0, 1, 2, 3 or blank).
T1	Time constant. (Real ≥ 0.0).
T2	Time constant. (Real; T2 > T1).
F	Frequency in cycles per unit time. (Real ≥ 0.0; Default = 0.0).
P	Phase angle in degrees. (Real; Default = 0.0).
C	Exponential coefficient. (Real; Default = 0.0).
B	Growth coefficient. (Real; Default = 0.0).

DYNAMIC LOADS

- **For a constant load, leave fields F, P, C, and B blank.**
- **For a cosine wave, specify F = 1.0, and leave fields P, C, and B blank.**
- **For a sine wave, specify F = 1.0, P = - 90° and leave fields C and B blank.**

DYNAMIC LOADS

- **DLOAD Bulk Data Entry**
  - Description: Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1 or RLOAD2 entries for frequency response or TLOAD1 or TLOAD2 entries for transient response.
  - Format:

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4							

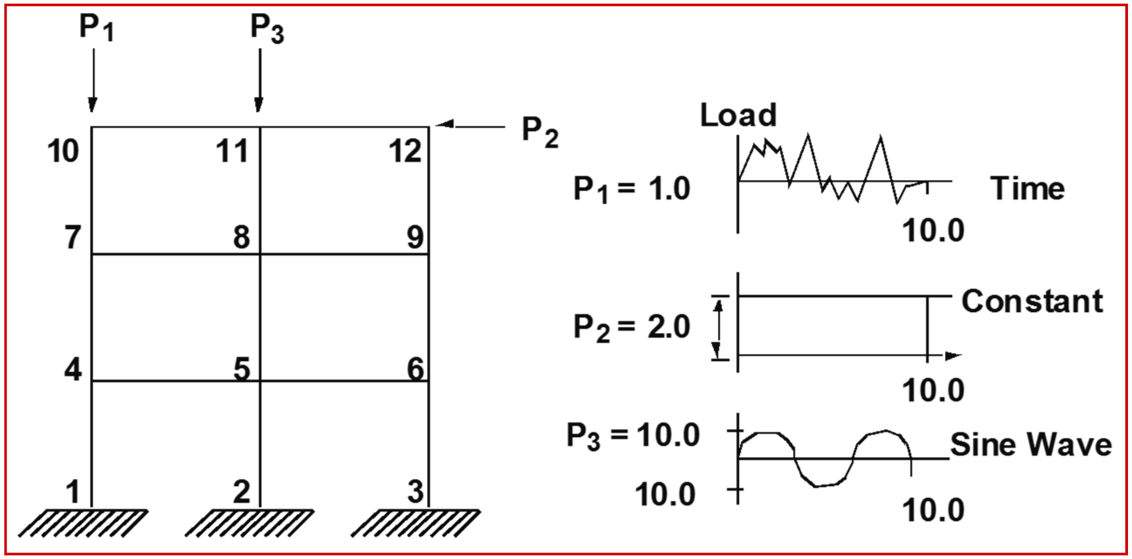
  - Example:

DLOAD	17	1.	2.	6	-2.	7	2.	8	
	-2.	9							

DYNAMIC LOADS

Field	Contents
Sid	Load set identification number. (Integer > 0).
S	Scale factor. (Real).
Si	Scale Factors. (Real).
Li	Load set identification numbers of RLOAD1, RLOAD2, TLOAD1, and TLOAD2 entries. (Integer > 0).

DYNAMIC LOADS EXAMPLE



- **Step 1**
  - P1 : TLOAD1,101,1,0,0,1
  - P2 : TLOAD1,102,2,0,0,2

    or

    TLOAD2,102,2,0,0.0,10.0
  - P3 : TLOAD2,103,3,,0,0.0,10.0,1.0,-90.0

DYNAMIC LOADS EXAMPLE

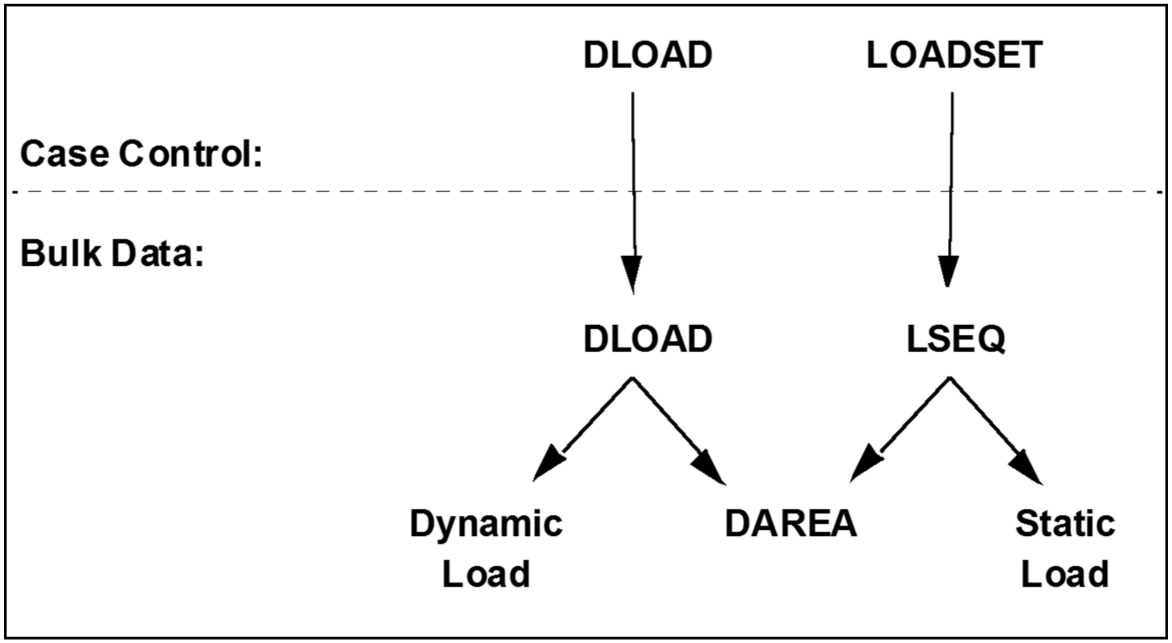
- **Step 2**
  - DAREA,1,10,2,-1.0
  - DAREA,2,12,1,-2.0
  - DAREA,3,11,2,-10.0
- **Step 3**
  - DLOAD,10,1.0,1.0,101,1.0,102,1.0,103
- **Step 4**
  - DLOAD=10 in Case Control

STATIC LOADS IN TRANSIENT ANALYSIS

- **A user needs to follow five steps to specify static loads in transient analysis.**
- **The five steps are:**
  1. Define the static loads using FORCEi, GRAV, MOMENTi, etc., that are referenced by the LOAD Case Control command.
  2. Define a LSEQ Bulk Data entry to point to a TLOADi entry and to a load set that is referenced by a LOADSET Case Control command.
  3. Define a TLOAD1 or TLOAD2 entry to define a constant function with time.
  4. Combine all the TLOADi entries through the DLOAD Bulk Data entry.
  5. Select the DLOAD entry through the DLOAD Case Control command and the LSEQ entry through the LOADSET Case Control command.

LSEQ ENTRY

- Defines static loads that will be applied dynamically.
- Relationship to other commands and entries:



DLOAD

LOADSET

Case Control:

Bulk Data:

DLOAD

LSEQ

Dynamic Load

DAREA

Static Load

LSEQ ENTRY (cont.)

- **LSEQ Bulk Data Entry**
  - Description: Defines a sequence of static load sets.
  - Format:

1	2	3	4	5	6	7	8	9	10
LSEQ	SID	DAREA	LID	TID					

  - Example:

LSEQ	100	200	1000	1001					

Field	Contents
SID	Set identification of the set of LSEQ entries. (Integer > 0).
DAREA	The DAREA set identification assigned to this static load vector. (Integer > 0).

LSEQ ENTRY (cont.)

Field	Contents
LID	Load set identification number of a set of static load entries such as those Referenced by the LOAD Case Control command. (Integer > 0 or blank).
TID	Temperature set identification of a set of thermal load entries such as those referenced by the TEMP(LOAD) Case Control Command. (Integer > 0 or blank).

EXAMPLE: STATIC LOADS IN TRANSIENT ANALYSIS

- **Aim: to specify gravity load in transient analysis.**
- **Solution:**

<u>Case Control Section</u>

– Step 5: DLOAD = 50011

LOADSET = 5000

→ to LSEQ

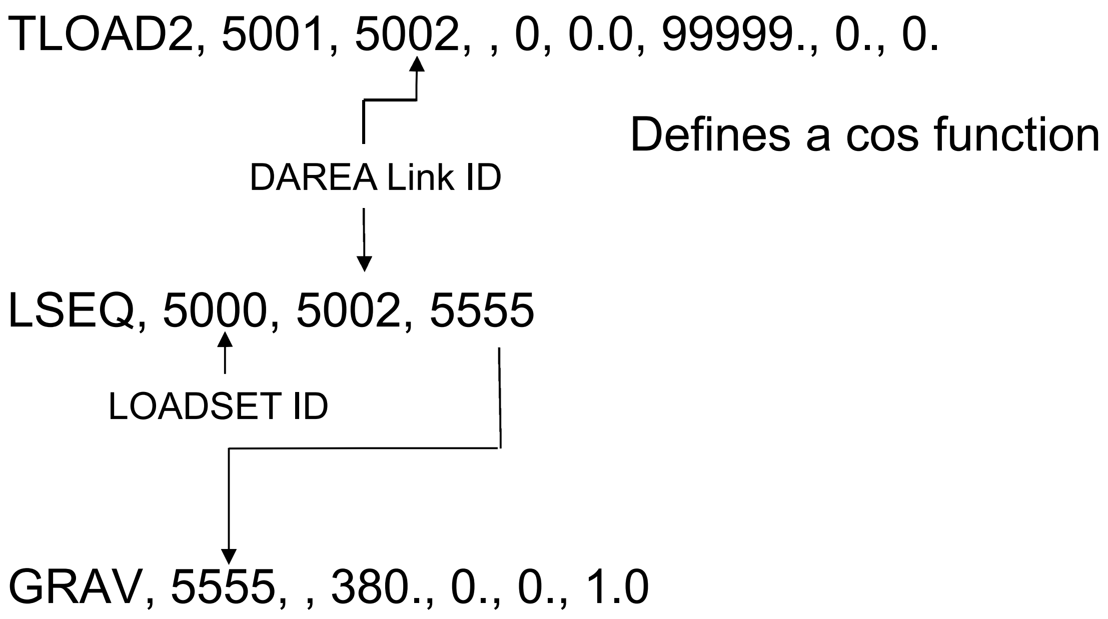
<u>Bulk Data Section</u>

– Step 4: DLOAD, 50011, 1.0, 1.0, 5001, 1.0, 4444,….

Additional Loads

– Step 3: TLOAD2, 5001, 5002, , 0, 0.0, 99999., 0., 0.

EXAMPLE: STATIC LOADS IN TRANSIENT ANALYSIS (cont.)



- Step 3: TLOAD2, 5001, 5002, , 0, 0.0, 99999., 0., 0.

Defines a cos function

DAREA Link ID

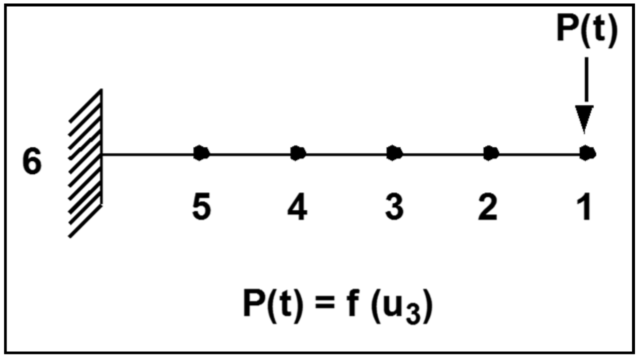
- Step 2: LSEQ, 5000, 5002, 5555

LOADSET ID

- Step 1: GRAV, 5555, , 380., 0., 0., 1.0

NONLINEAR LOADS

- Allows for the specification of load at a particular degree of freedom to be the function of displacement and velocity at another degree of freedom.
- Example:



- Load at grid point 1, displacement component 2 as a function of the displacement component 1 at grid point 3.

NONLINEAR LOADS (cont.)

- Useful for specifying nonlinear springs and nonlinear damping.
- Nonlinear loads are specified using NOLINi entries.
- Four NOLINi entries (NOLIN1, NOLIN2, NOLIN3, and NOLIN4) to specify mechanical loads.
- Nonlinear loads are selected via the NONLINEAR Case Control command.
- Output of nonlinear loads via NLLOAD Case Control command.
- Nonlinear loads cannot be selected via the DLOAD Case Control command.
- All degrees of freedom referenced on NOLINi entry must be members of the solution set.

NONLINEAR LOADS (cont.)

- **Velocity for an independent degree of freedom (for the purpose of loads) is calculated as**

$$\dot{U}_t = \frac{U_t - U_{t-1}}{\Delta_t}$$

  - Note: This may be different from that calculated in the integration scheme. But it is acceptable.
- **In all NOLINi entries a degree of freedom is specified by the grid number and its component number.**
- **All loads generated with NOLINi entries lag behind by one time step Δt.**

NONLINEAR LOADS (cont.)

- **NOLIN1 Bulk Data Entry**
  - Description: Defines nonlinear transient forcing functions of the form.

    Function of displacement: $P_i(t) = S * T(u_j(t))$ (1)

    Function of velocity: $P_i(t) = S * T(u_j(t))$ (2)

    where $u_j(t)$ and $u_j(t)$ are the displacement and velocity at point GJ in the direction of CJ.

  - Format:

1	2	3	4	5	6	7	8	9	10
NOLIN1	SID	G1	C1	S	GJ	CJ	TID		

  - Example:

NOLIN1	21	3	4	2.1	3	10	6		

NONLINEAR LOADS (cont.)

Field	Contents
SID	Nonlinear load set identification number. (Integer > 0).
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0).
CI	Component number for GI. (0 < Integer ≤ 6; blank or zero if GI is a scalar or extra point).
S	Scale factor. (Real).
GJ	Grid, scalar, or extra point identification number. (Integer > 0).
CJ	Component number for GJ according to the following table:

Type of point	**Displacement**	**Velocity**
Grid	1 ≤ Integer ≤ 6	11 ≤ Integer ≤ 16
Scalar	Blank or Zero	Integer = 10
Extra	Blank or Zero	Integer = 10

TID	Identification number of a TABLEDi entry. (Integer > 0).

NONLINEAR LOADS (cont.)

- **NOLIN2 Bulk Data Entry**
  - Description: Defines nonlinear transient forcing functions of the form.

$$P_i(t) = S * X_j(t) * X_k(t)$$

where X_j and X_k can be either displacement or velocity at points G_j and G_k in the directions of C_j and C_k.

  - Format:

1	2	3	4	5	6	7	8	9	10
NOLIN2	SID	G1	C1	S	GJ	CJ	GK	CK	

  - Example:

NOLIN2	14	2	1	2.9	2	1	2		

NONLINEAR LOADS (cont.)

- **NOLIN3 Bulk Data Entry**
  - Description: Defines nonlinear transient forcing functions of the form.

$$P_i(t) = \begin{cases} S * [X_j(t)]^A & , X_j(t) > 0 \\ 0 & , X_j(t) \leq 0 \end{cases}$$

where may be a displacement or a velocity at point GJ in the direction of CJ.

  - Format:

1	2	3	4	5	6	7	8	9	10
NOLIN3	SID	G1	C1	5	GJ	CJ	A		

  - Example:

NOLIN3	4	102		-6.1	2	15	2		

NONLINEAR LOADS (cont.)

- **NOLIN4 Bulk Data Entry**
  - Description: Defines nonlinear transient forcing functions of the form.

$$P_i(t) = \begin{cases} -S*[-X_j(t)]^A & , X_j(t) < 0 \\ 0 & , X_j(t) \geq 0 \end{cases}$$

where may be a displacement or a velocity at point GJ in the direction of CJ.

  - Format:

1	2	3	4	5	6	7	8	9	10
NOLIN4	SID	G1	C1	S	GJ	CJ	A		

  - Example:

NOLIN4	2	4	6	2	101		16.3		

INITIAL CONDITIONS

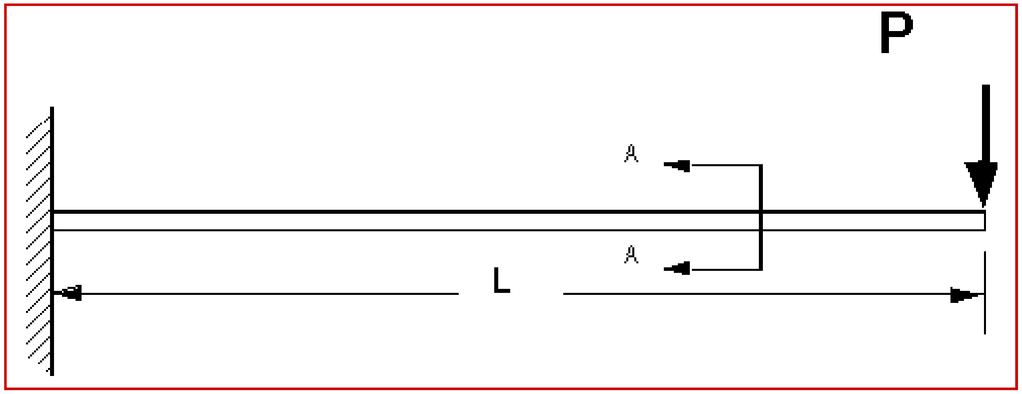
- **May impose initial displacements and/or velocities with a TIC Bulk Data entry.**
- **IC Case Control command selects TIC entries in the Bulk Data Section.**
- **Warning: Initial conditions for unspecified degrees of freedom are set to zero.**
- **Initial conditions may be specified only for A-set degrees of freedom.**

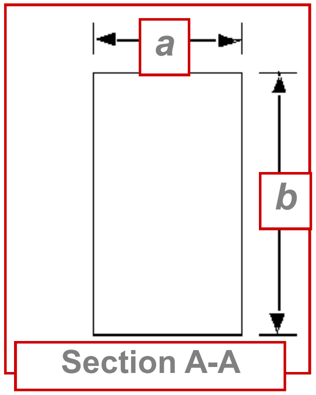
NONLINEAR TRANSIENT DYNAMIC ANALYSIS OF A BEAM

CASE STUDY

PROBLEM DESCRIPTION

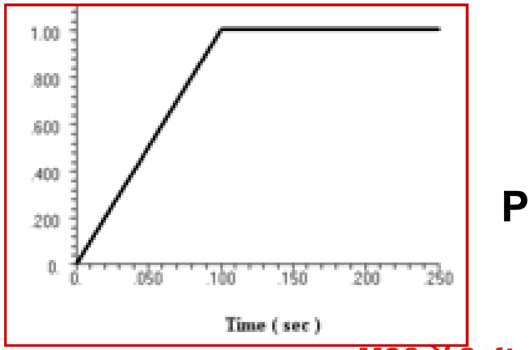
- In this study, a cantilever beam is subjected to a ramped load. Earlier (Workshop 1) we performed a static analysis using both small, and large deflection. Now we will analyze it with a nonlinear transient dynamic procedure, assuming the load is ramped up to the full value of 6000 lbs in 0.10 second.



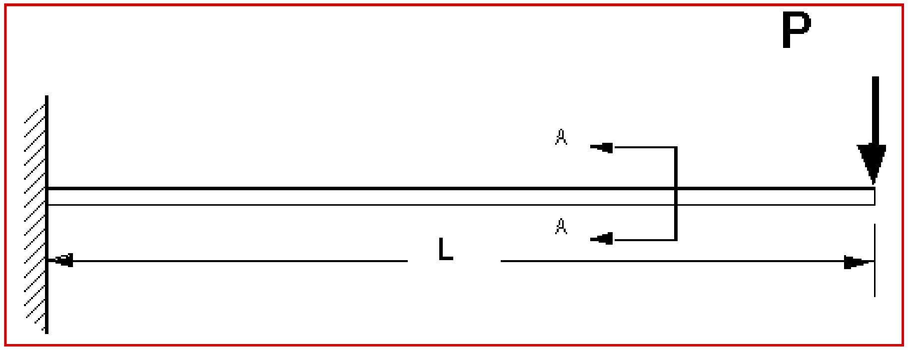


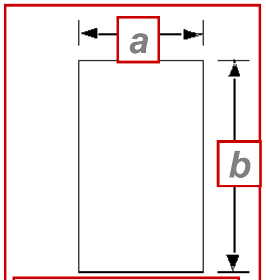
Section A-A

(Data on next page)



PROBLEM DESCRIPTION (CONT.)



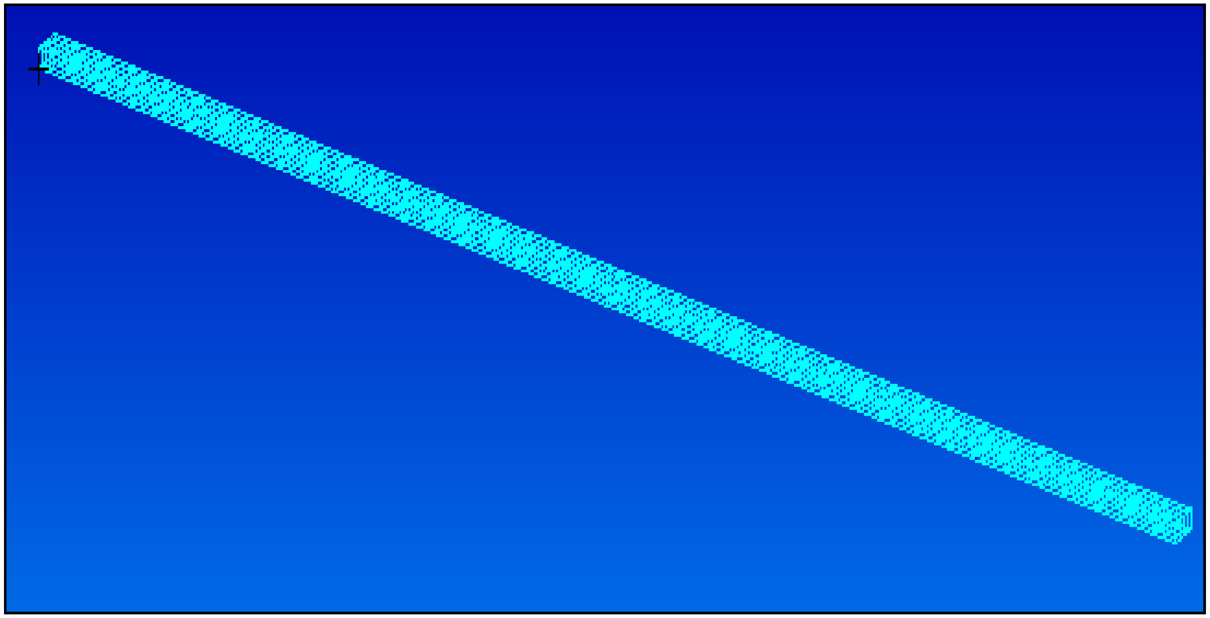


Section A-A

Length, L	100.0 in	2.54 m
a	1.0 in	25.4 mm
b	2.0 in	50.8 mm
Young’s Modulus	30.0 x 10^6 lb/in^3	207 GPa
Poisson’s Ratio	0.3	0.3
P	6000 lb	27200 N

PROBLEM DESCRIPTION (CONT.)

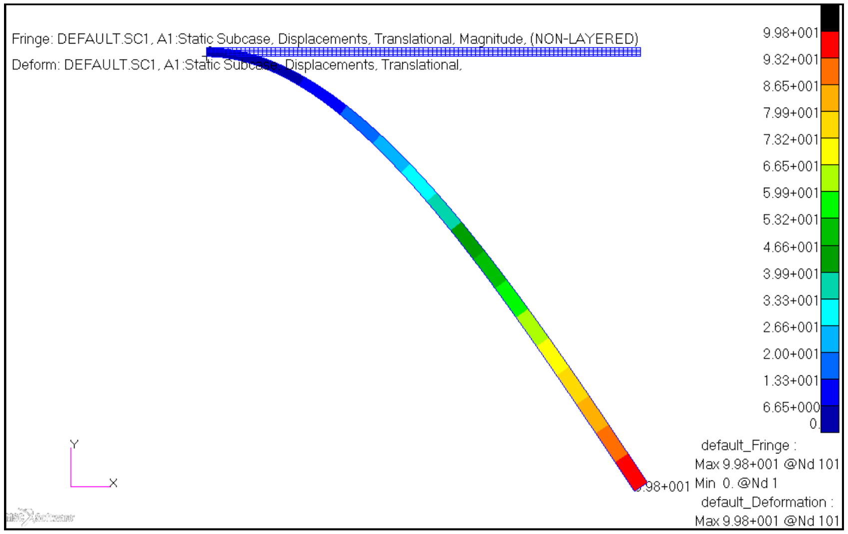
- **The simulation is to use 3D solid finite elements. The elements are uniformly spaced along the length of the beam (i.e. a mesh 100 elements long and four elements deep).**

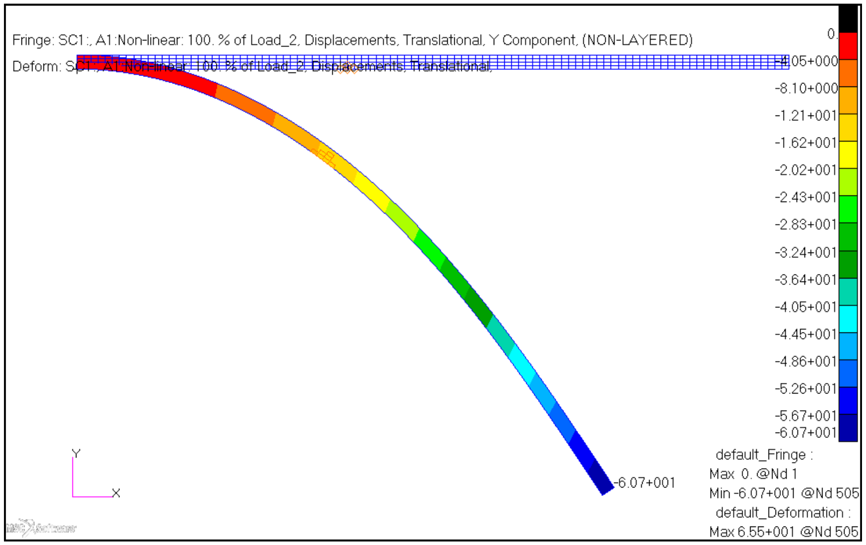


- **Objective:**
  - Nonlinear Static vs. Nonlinear Transient Dynamic Analysis.

COMPARE YOUR RESULTS

Y-Displacement at Tip

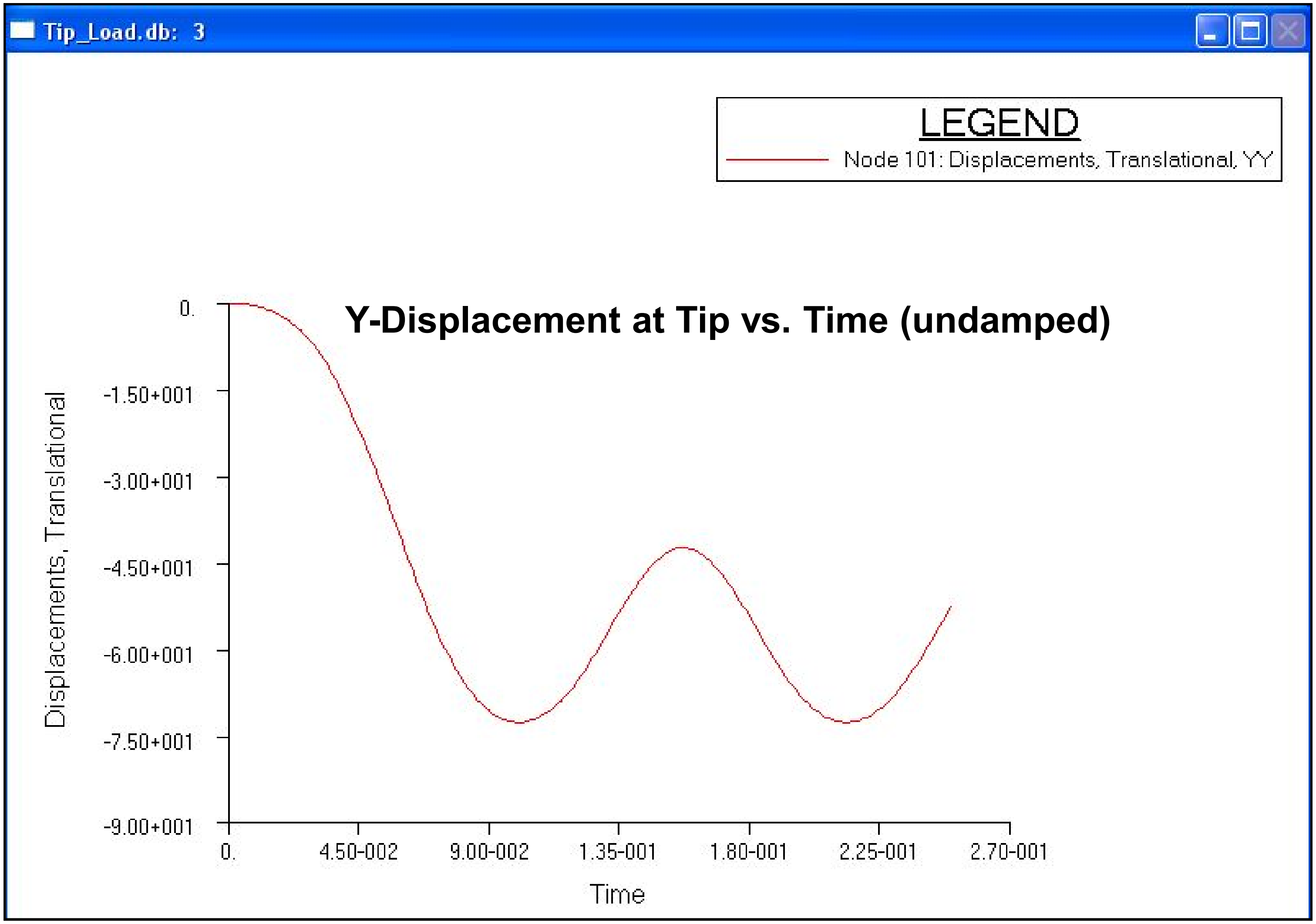




Nonlinear Static

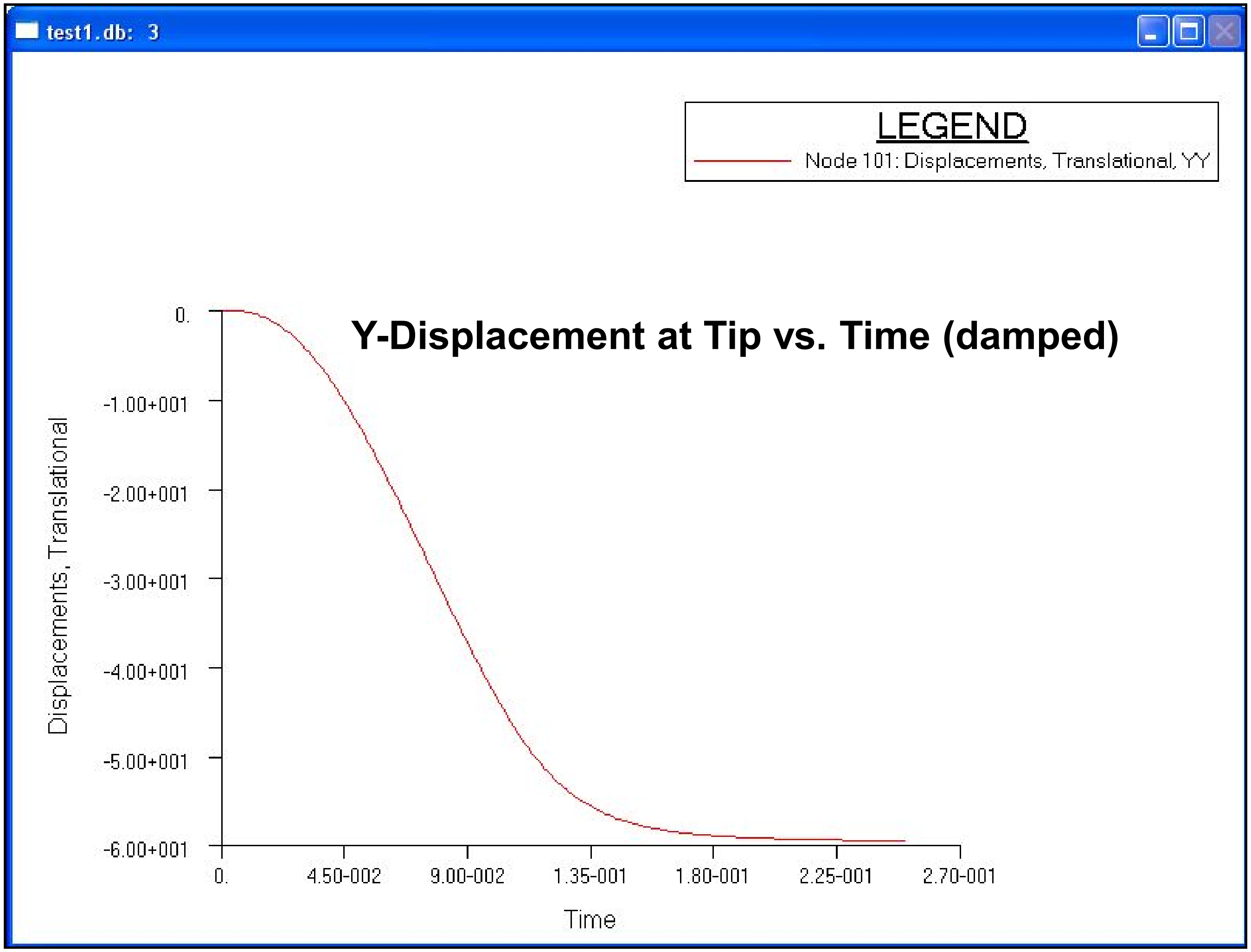
Nonlinear Transient Dynamic (100% of load)

PLOT RESULTS



Nonlinear Transient Dynamic

PLOT RESULTS



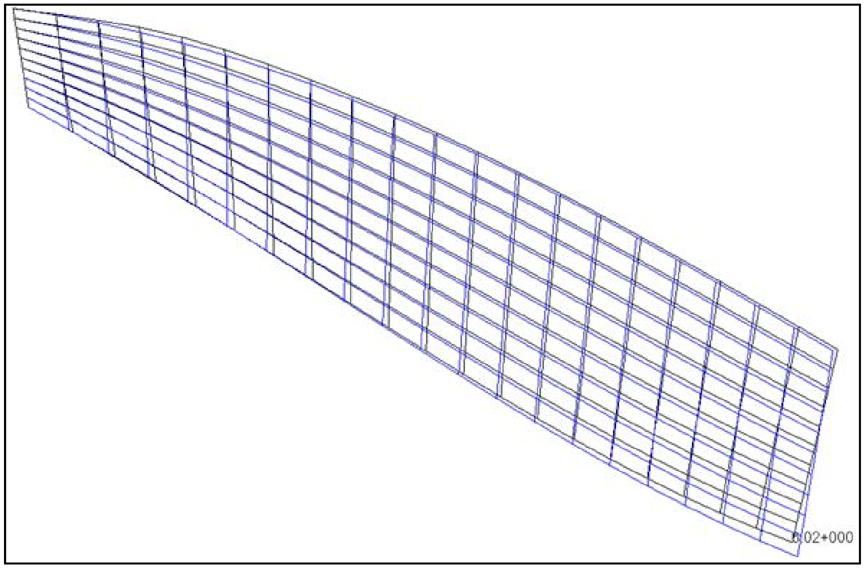
Nonlinear Transient Dynamic

HINTS AND RECOMMENDATIONS

- Identify the type of nonlinearity.
- Localize nonlinear region.
- Divide time history by subcases for convenience.
- Select default values to start - TSTEPNL.
- Pick time step size based on highest frequency of interest. Twelve or more steps per cycle.
- Some damping is desirable for numerical stability.
- Avoid massless degrees of freedom.
- Use PFNT for highly nonlinear problems.

EXERCISE

- **Perform Workshop B1: Normal Modes Analysis of a Pre-stiffened Blade**



EXERCISE

- **Perform Workshop B2: Dynamic Collapse of a Cylinder**