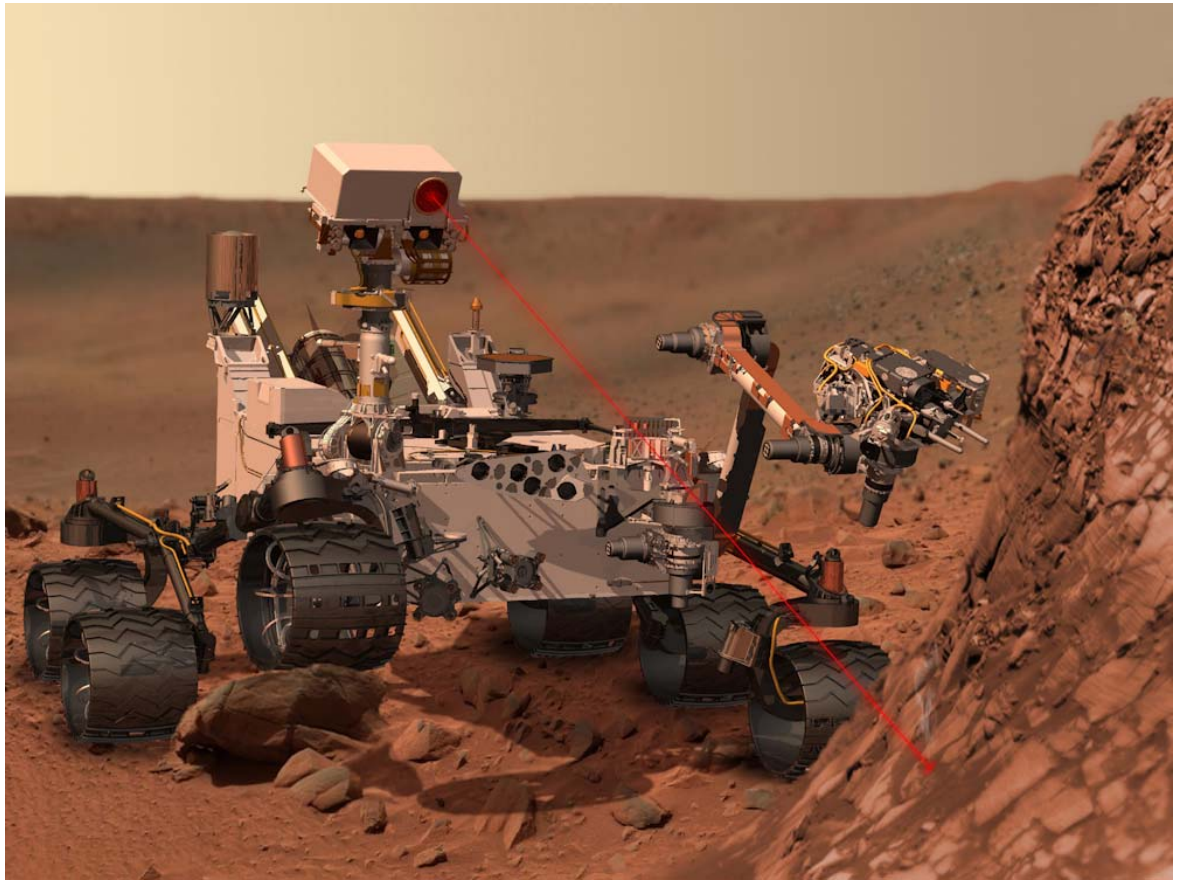
SIEMENS



Nonlinear Analysis Using Femap with NX Nastran

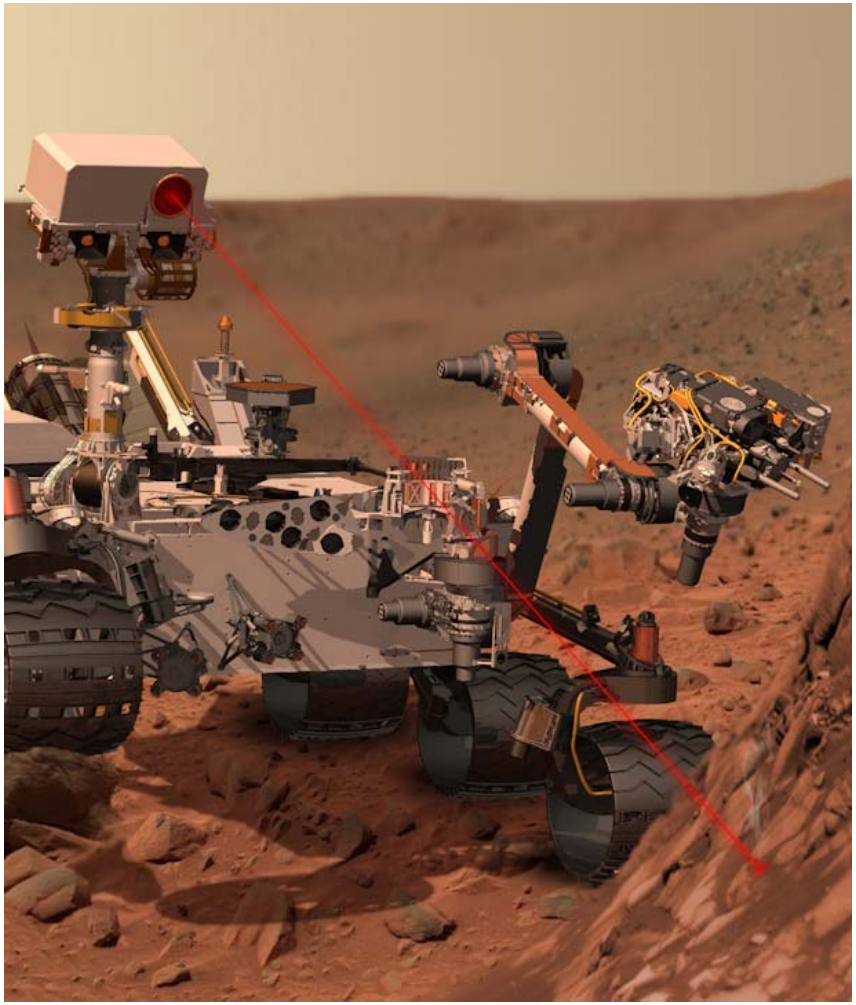
Chip Fricke, Principal Applications Engineer, Siemens PLM Software

FEMAP SYMPOSIUM 2013
Mission Critical Design and Analysis

Unrestricted © Siemens AG 2013 All rights reserved.

Agenda

Nonlinear Analysis Using Femap with NX Nastran



Who am I?

Overview of Nonlinear Analysis

Comparison of the NX Nastran Nonlinear and NX Nastran Advanced Nonlinear Solvers

Nonlinear Material Models

Comparison of Results – Basic vs. Advanced Nonlinear

NX Nastran Basic Nonlinear Analysis

NX Nastran Advanced Nonlinear Analysis

Femap Examples and NX Nastran Technical References

Unrestricted © Siemens AG 2013 All rights reserved.

SIEMENS

About Your Presenter

Principal Applications Engineer with the Femap Product Development Group of Siemens PLM Software

31 years experience in the PLM Software industry

- Pre-sales
- Technical Marketing
- Product Management

Aerospace Industry experience at Boeing

- Lead Engineer on the 757 program

BSCEE, University of Wisconsin – Madison

Email: chip.fricke@siemens.com

Twitter: @ae4mcae

Unrestricted © Siemens AG 2013 All rights reserved.

What Defines Nonlinear Analysis

Material Nonlinearity

Transient (time-dependent) loading

Large displacement

Contact due to

- Closure or opening of large gaps
- Rigid contact bodies
- “Double-sided” contact
- Edge to Edge contact
- Collision or impact

Unrestricted © Siemens AG 2013 All rights reserved.

NX Nastran Basic Nonlinear vs. NX Nastran Advanced Nonlinear

SIEMENS

Capability	NX Nastran Basic Nonlinear	NX Nastran Advanced Nonlinear
Large Strain (> 10%)	-	✓
Linear Elastic Material	✓	✓
Elasto-plastic Material	✓	✓
Plastic Material	✓	✓
Creep Material	✓	✓
Hyperelastic Material	✓(added in NX Nastran 8)	✓
Transient loading	✓	✓
Gap and Slide Line Contact	✓	✓
Rigid Body Contact	-	✓
Double-sided Contact	-	✓
Edge-to-edge Contact	✓	-
Impact	-	✓

Unrestricted © Siemens AG 2013 All rights reserved.

Nonlinear Material Models

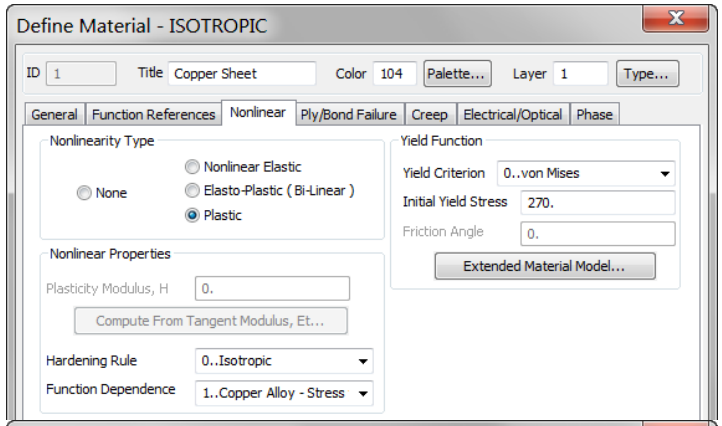
For *Isotropic* materials, select the **Nonlinear** tab in the *Define Material* dialog box

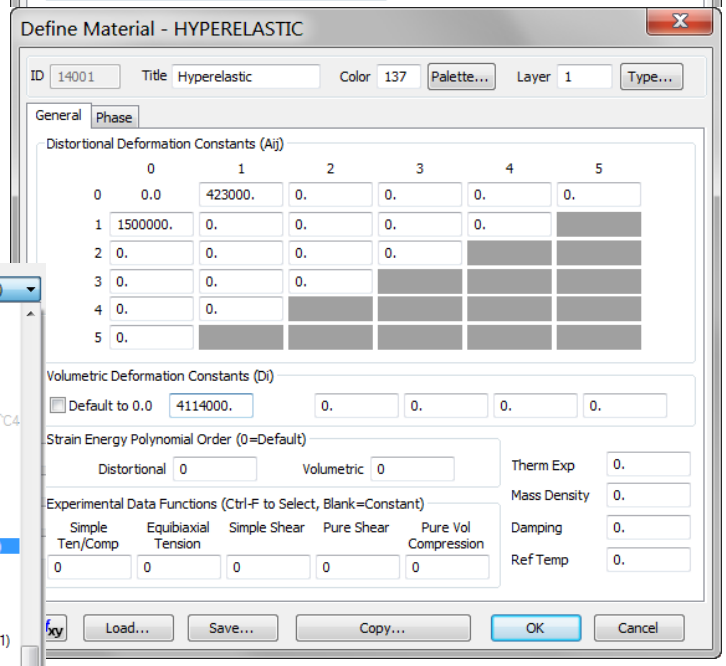
- Specify the **Nonlinearity Type** and their parameters

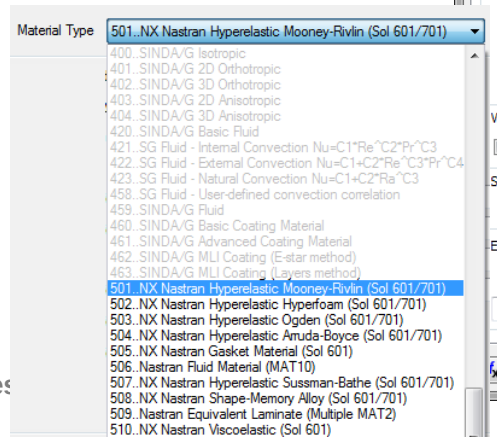
Hyperelastic materials are designated by setting the *Type* to **Hyperelastic**

- Hyperelastic material properties are complex and should be supplied by the material supplier or by testing

Additional NX Nastran Nonlinear Materials are available as “Other Types”





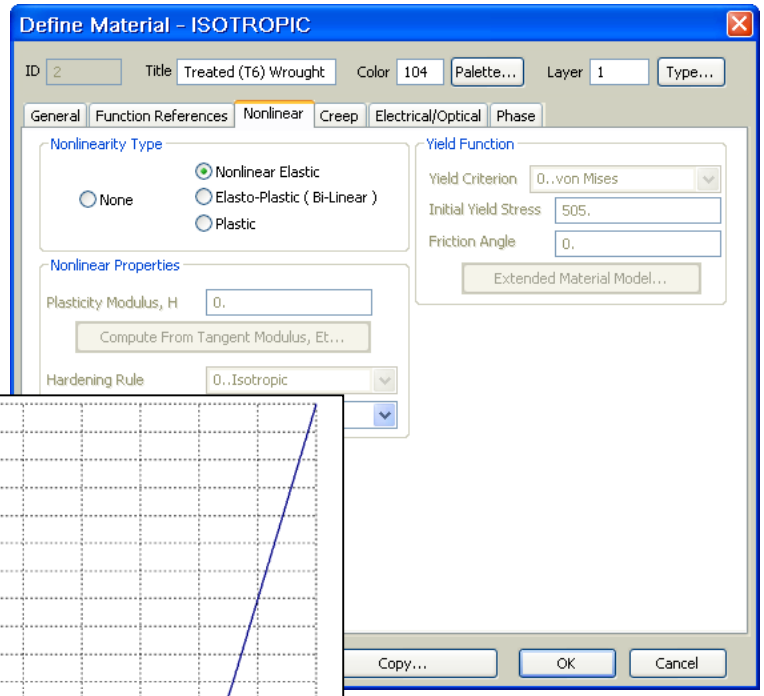


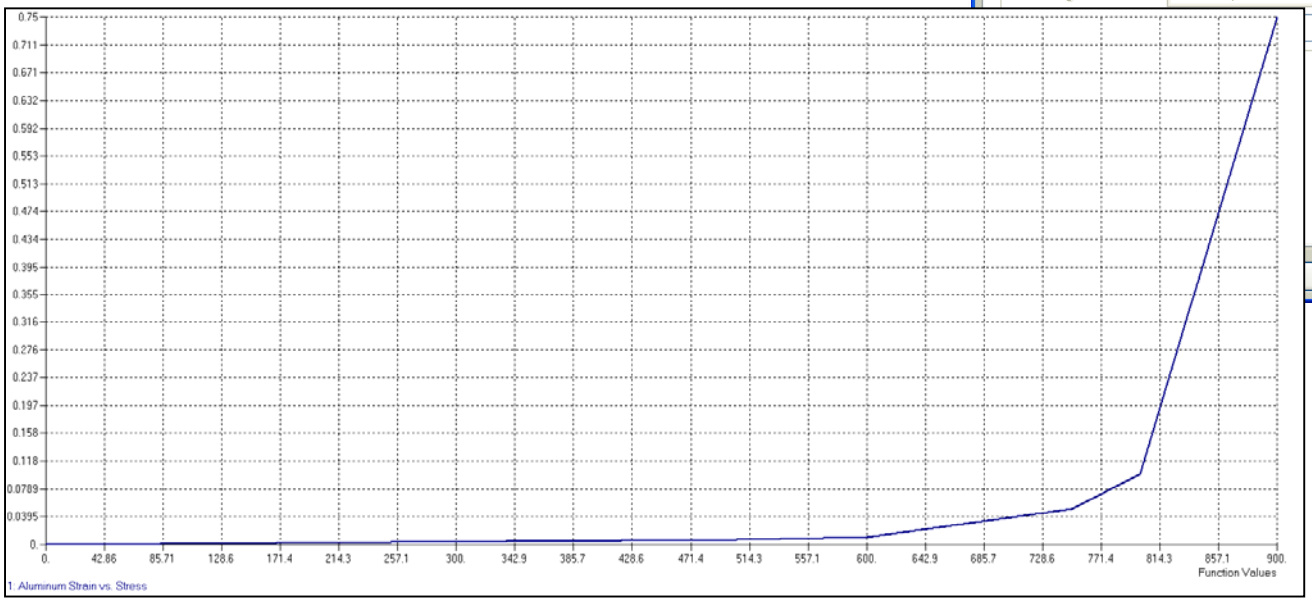
Unrestricted © Siemens AG 2013 All rights res

Linear Elastic Material

No strain hardening

- Used only for *Isotropic* materials.
- Define with a **vs. Stress** OR a **Stress vs. Strain** *Function*





Unrestricted © Siemens AG 2013 All rights reserved.

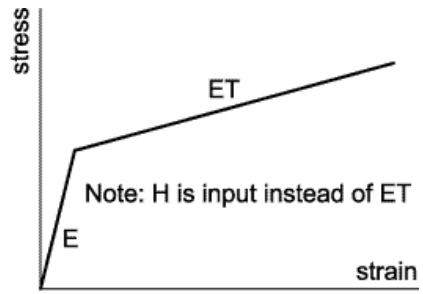
Elasto-Plastic Material

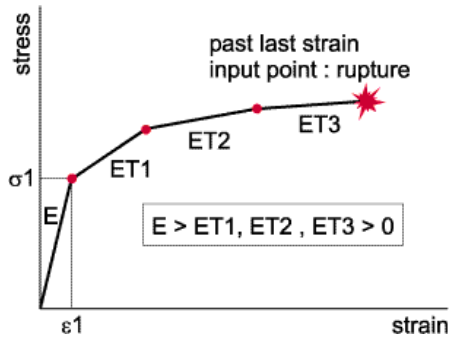
Basic assumptions:

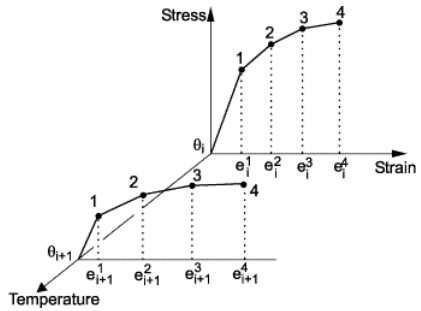
- Von Mises yield condition
- Isotropic, kinematic or mixed hardening
- Bilinear or multi-linear stress-strain curves
- Thermal strains can be included

This material can be used with:

- Small displacements/small strains
- Large displacements/small strains
- Large displacements/large strains
- Temperature dependence





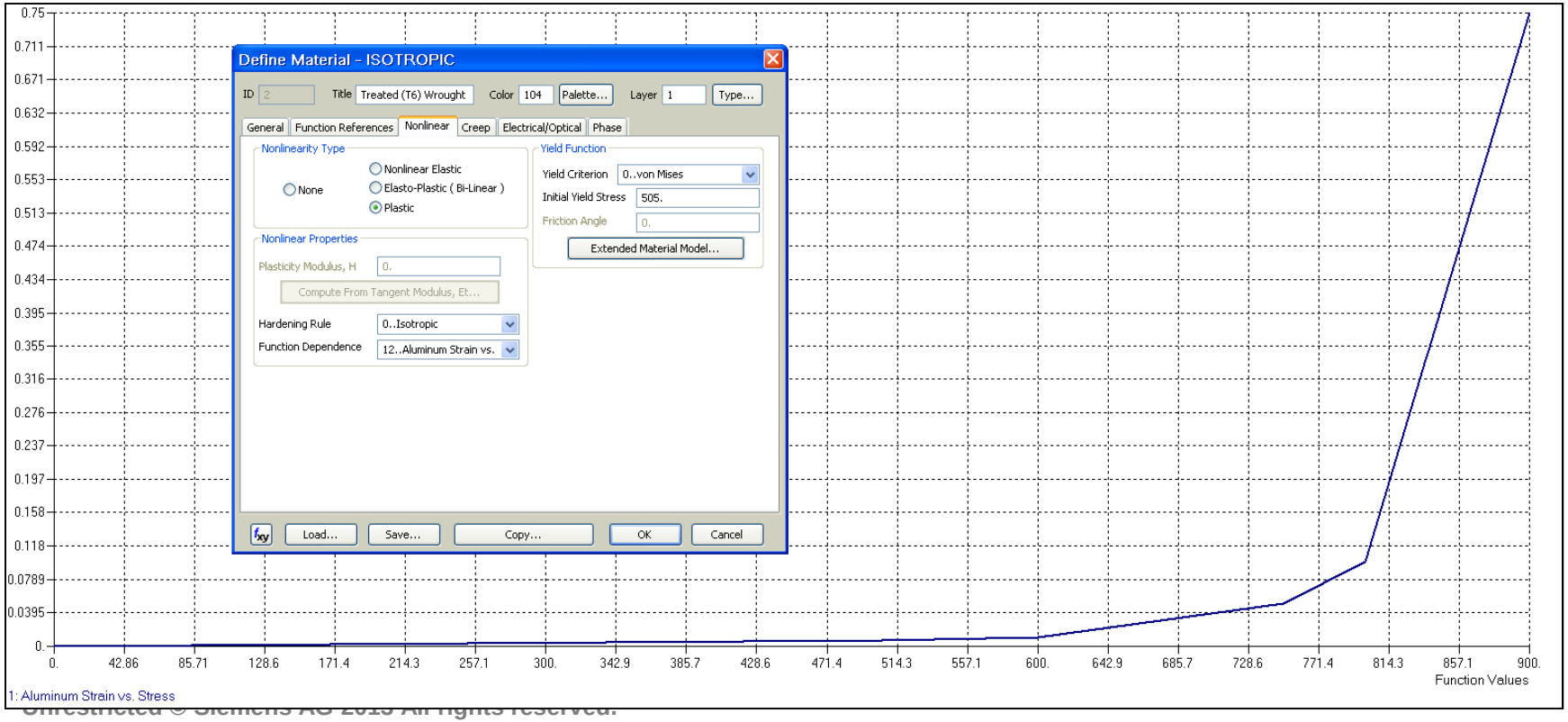


Unrestricted © Siemens AG 2013 All rights reserved.

Nonlinear Plastic Material

Strain hardening

Specify **Initial Yield Stress, Yield Criterion** and **Function Dependence** (**vs. Stress** or **Stress vs.** Strain Function)



Unrestricted © Siemens AG 2013 All rights reserved.

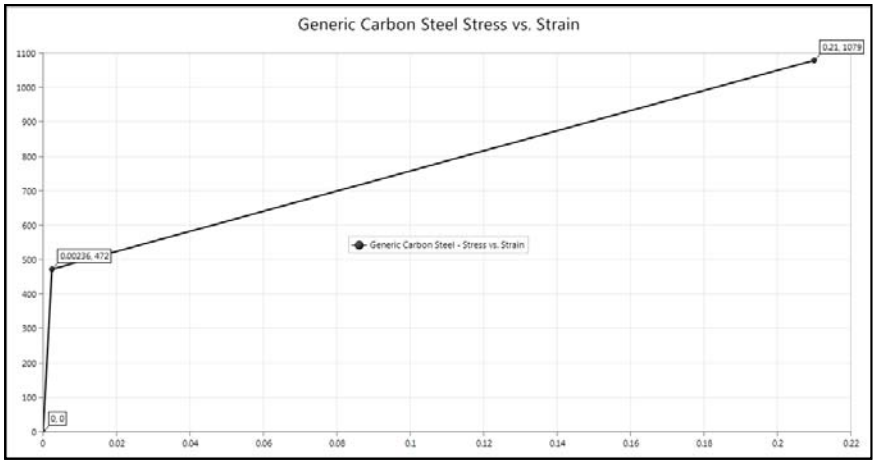
NX Nastran Plasticity Test Model

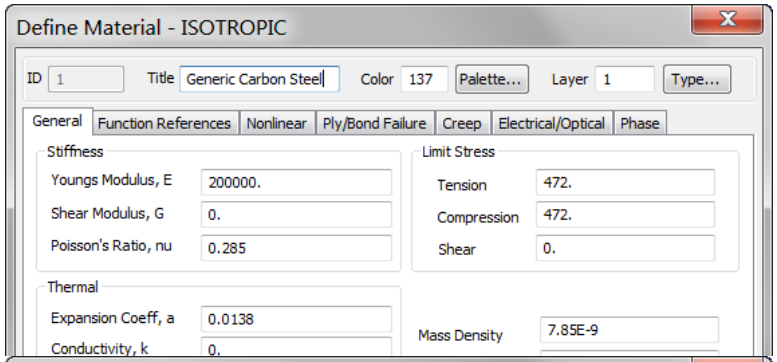
Simple Test Coupon Meshed with Hex Elements

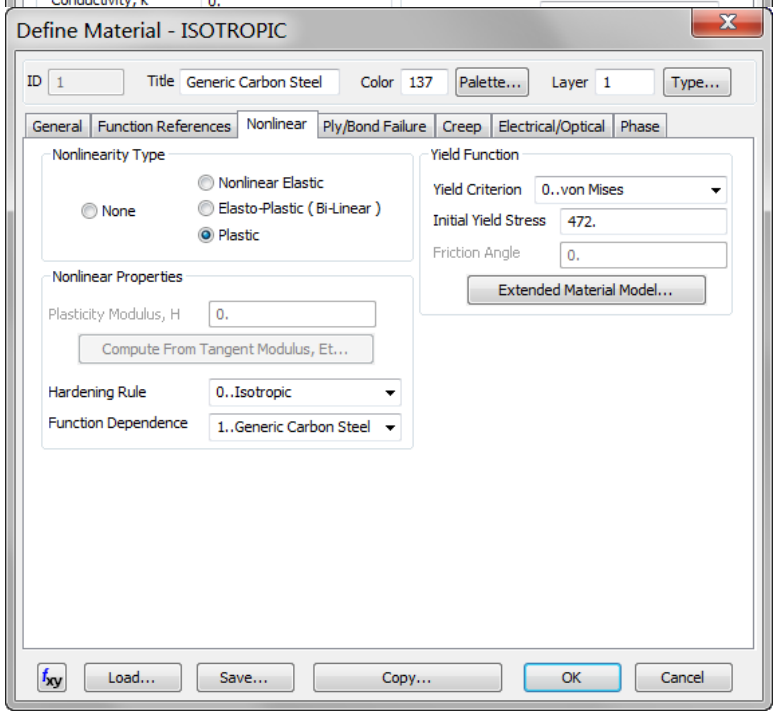
- Applied Tension Load = 2 MegaN

Generic Carbon Steel Material

- E = 200,000 Mpa
- Yield Stress = 472 MPa

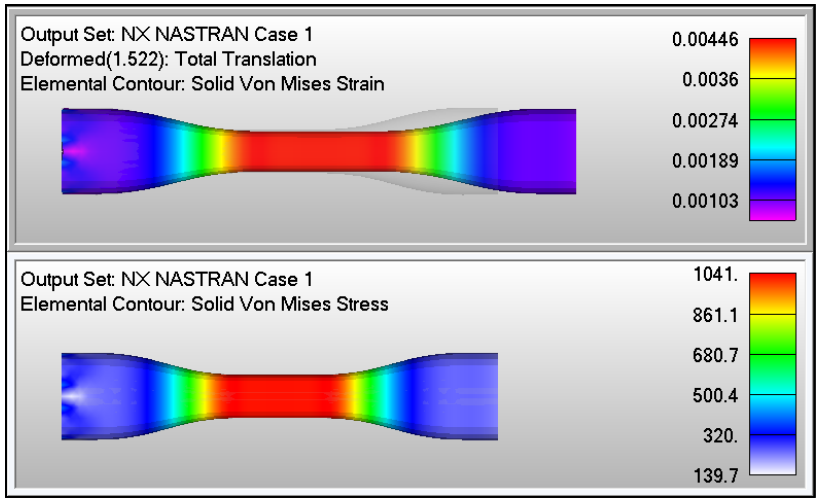


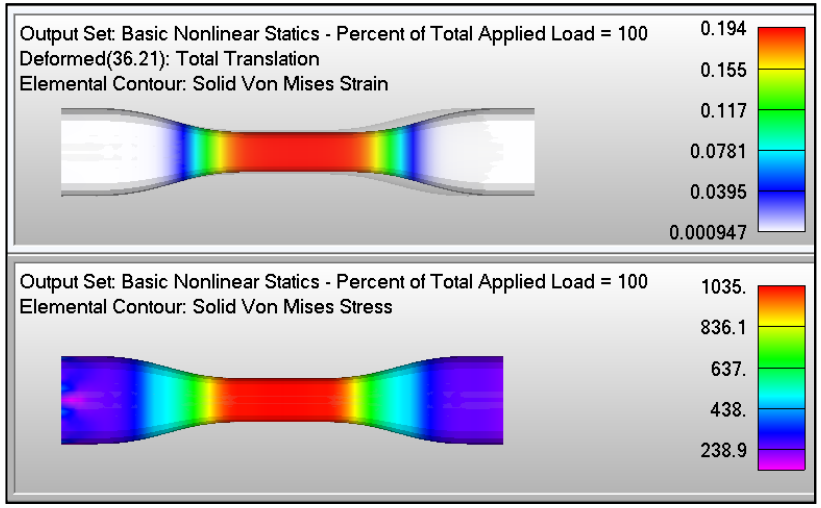




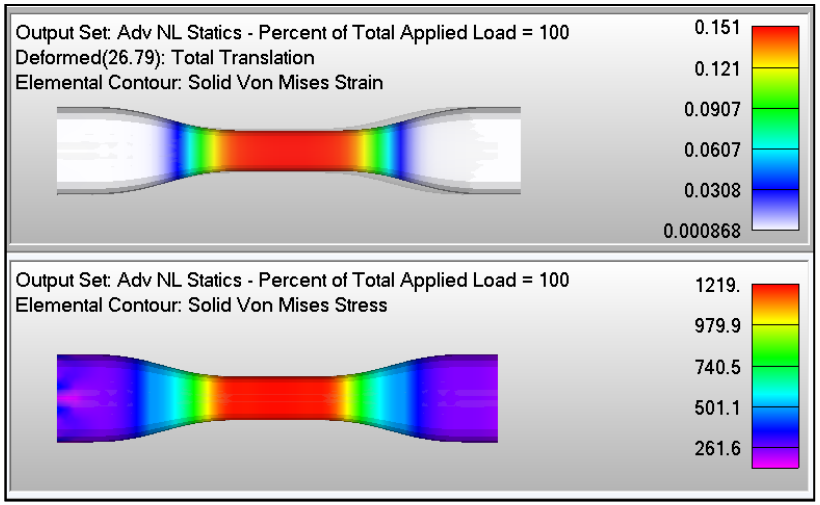
Unrestricted © Siemens AG 2013 All rights reserved.

Comparison between NX Nastran Linear Statics, Basic Nonlinear and Advanced Nonlinear Analyses





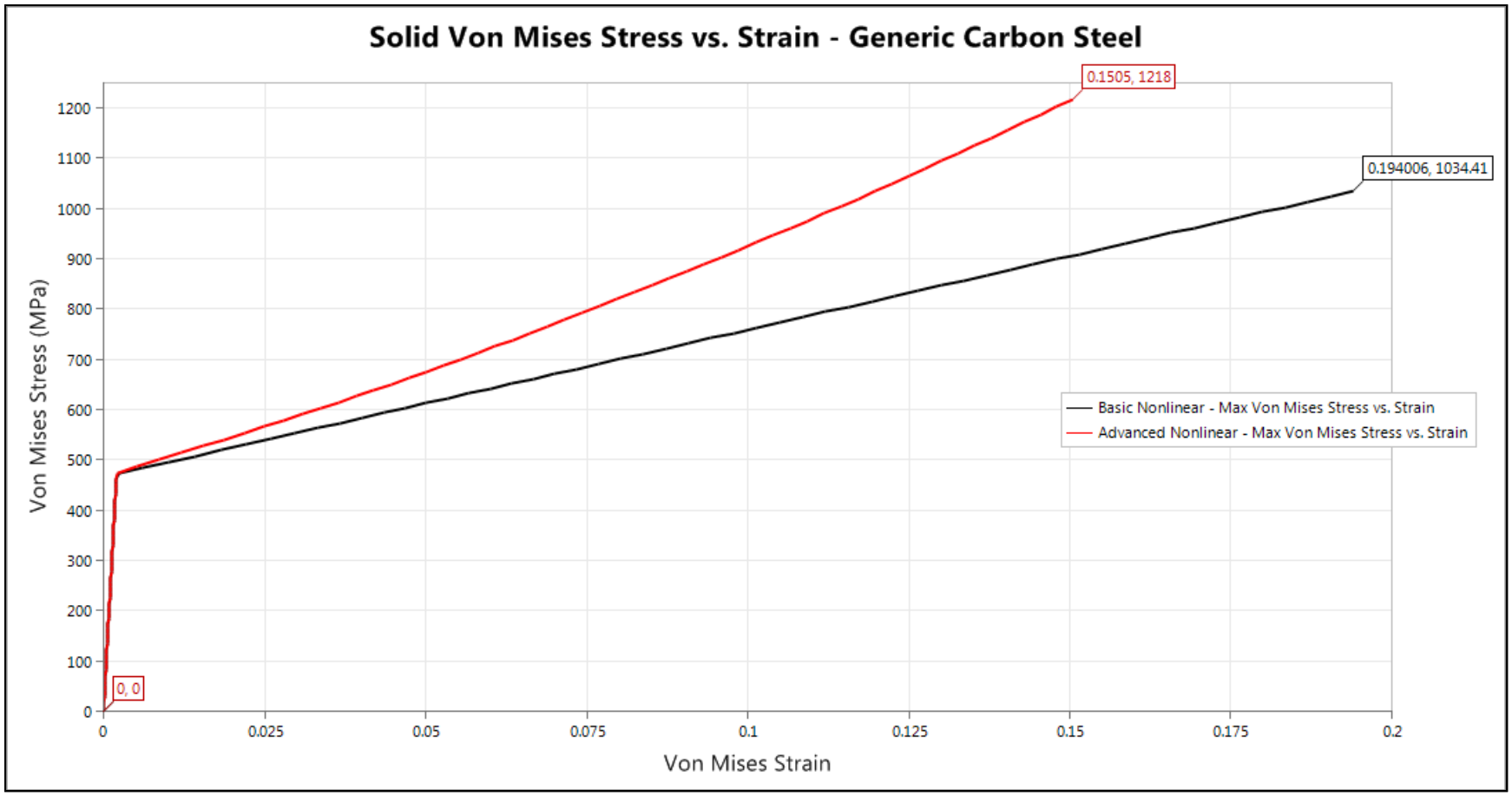
	Maximum Von Mises Stress	Axial Deformation
Linear Statics	1041 MPa	1.52 mm
NL Statics with Plasticity	1035 MPa	36.2 mm
Adv. NL Statics with Plasticity	1218 Mpa	26.8 mm



Unrestricted © Siemens AG 2013 All rights reserved.

Comparison of NX Nastran Basic Nonlinear vs. Advanced Nonlinear Results - Von Mises Stress vs. Strain Curves

SIEMENS



Unrestricted © Siemens AG 2013 All rights reserved.

NX Nastran Basic Nonlinear Analysis Settings

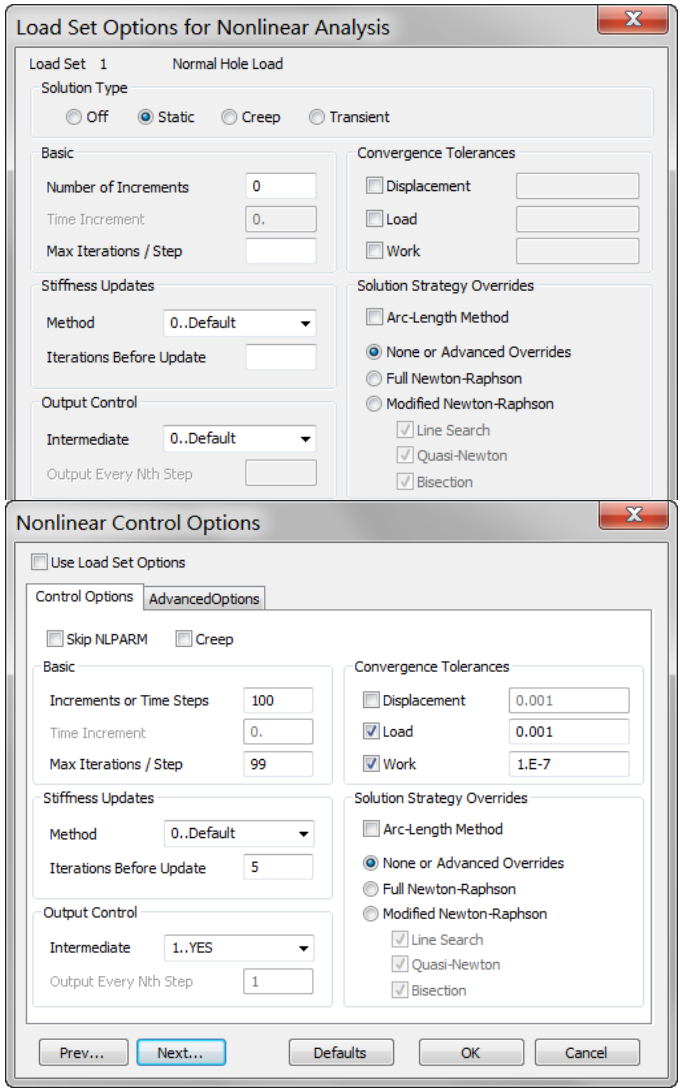
SIEMENS

All NX Nastran Basic Nonlinear analyses require the specification of:

- **Basic** Iteration settings
- **Stiffness Updates**
- **Output Control**
- **Convergence Tolerances**
- Optional **Solution Strategy Overrides**

Define these options in either:

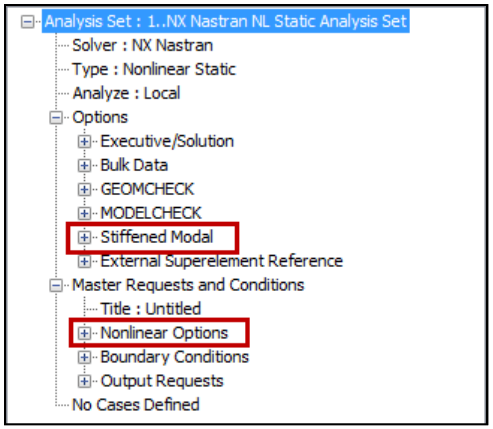
- As part of a *Load Set* in the *Load Set Options for Nonlinear Analysis* dialog box OR
- *Nonlinear Options* in an Analysis Set's *Master Requests and Conditions – Nonlinear Options* dialog box

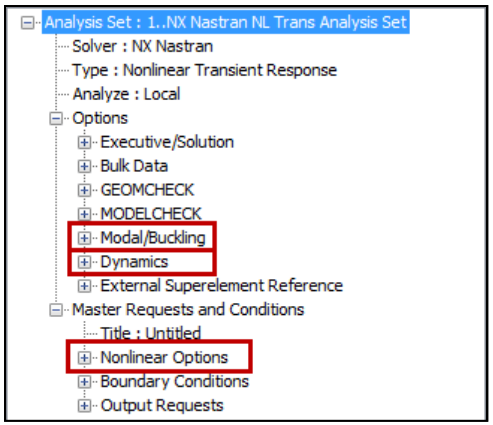


Unrestricted © Siemens AG 2013 All rights reserved.

NX Nastran Basic Nonlinear - Significant Differences in Analysis Options

Analysis Set Manager Options	Nonlinear Statics	Nonlinear Transient
Stiffened Modal – Runs a Stress-stiffened Normal Modes Analysis at the End of each Subcase	✓	
Multi-Case	✓	
Modal/Buckling + **Dynamics** – equivalent to a Direct or Modal Transient Dynamic Response analysis with Nonlinear Materials • Nonlinear damping must be specified as a material attribute • Force and GPForce results not available		✓
Nonlinear Options – Creep	✓	



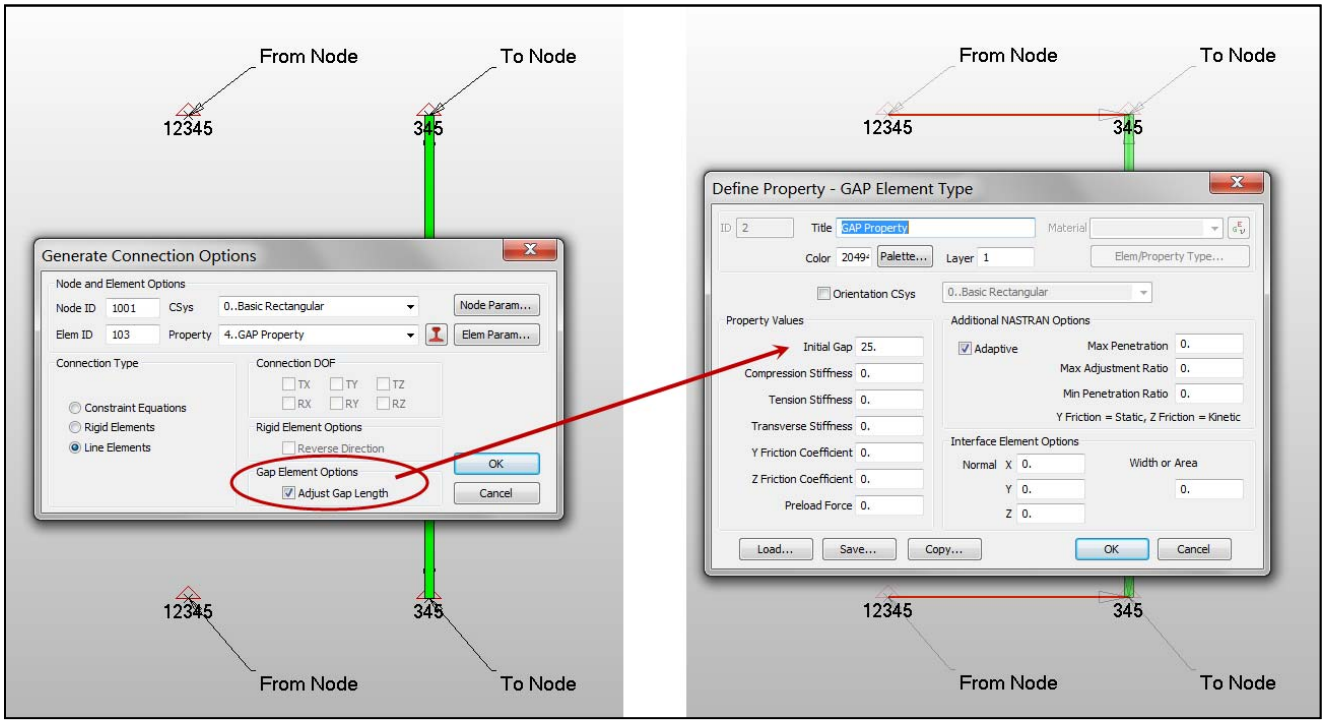


Unrestricted © Siemens AG 2013 All rights reserved.

Contact Modeling –
NX Nastran Basic Nonlinear

Contact can be modeled with two (2) element types:

- **SLIDE LINE** – element node connectivity must be defined correctly
- **GAP** – use the **Mesh > Connect > Closest Link** tool to create multiple Gap element connections

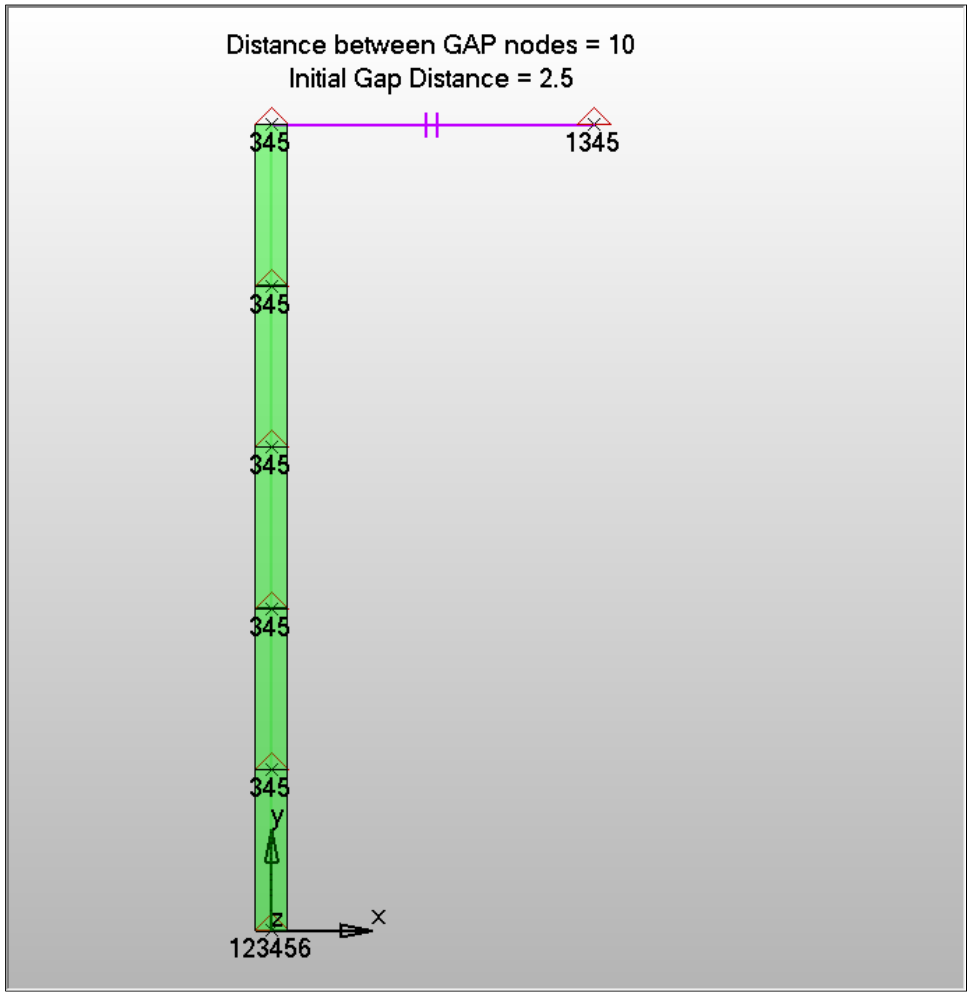


Initial Gap can be used to define a gap other than the physical gap between the ends of the GAP element

Unrestricted © Siemens AG 2013 All rights reserved.

GAP Element Example –
Use of Orientation Vector and Initial Gap Distance

The Orientation vector of a gap element defines the direction of positive direction of a GAP element.

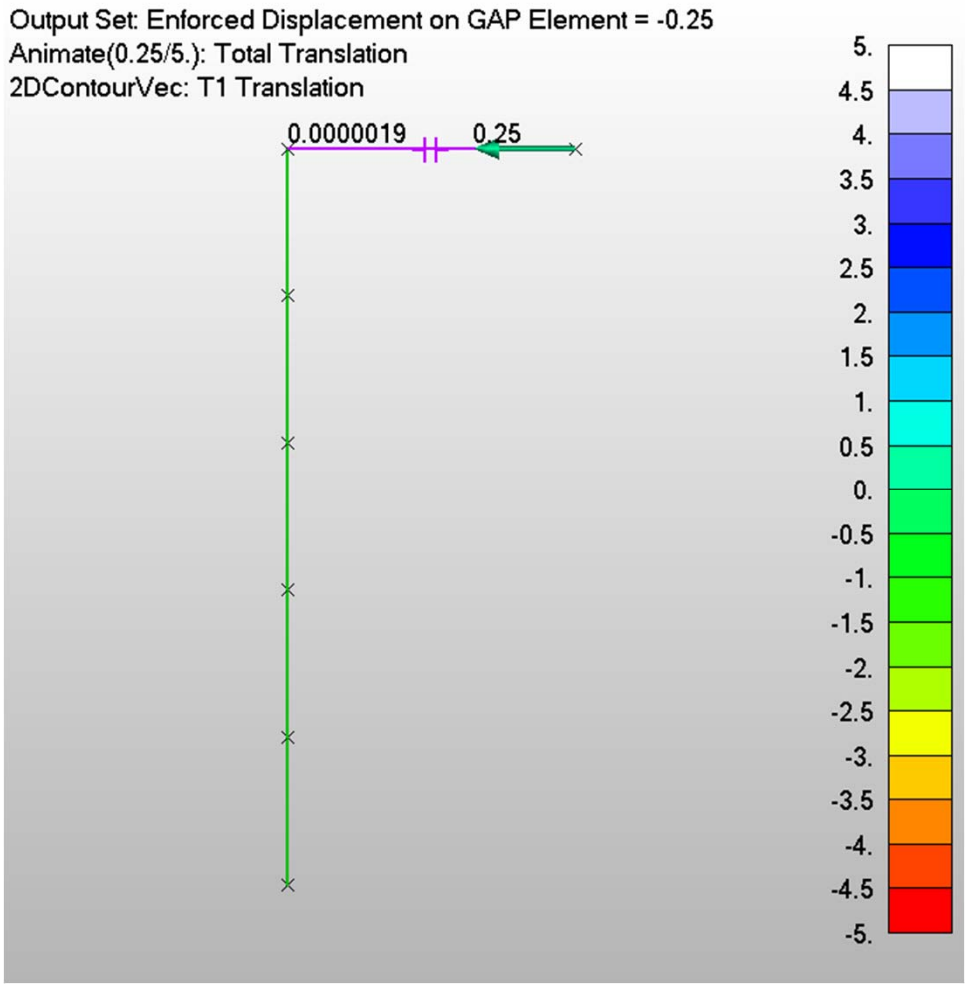


Unrestricted © Siemens AG 2013 All rights reserved.

GAP Element Example – Use of Orientation Vector and Initial Gap Distance

Applied Displacement of 0 to -5 to 0 applied to the right end of the GAP element

- GAP element is oriented with the X-Axis of the Global Rectangular CSYS
- **Initial Gap** is set to 2.5
- Note how the top of the beam starts to displace after the GAP displacement = 2.5

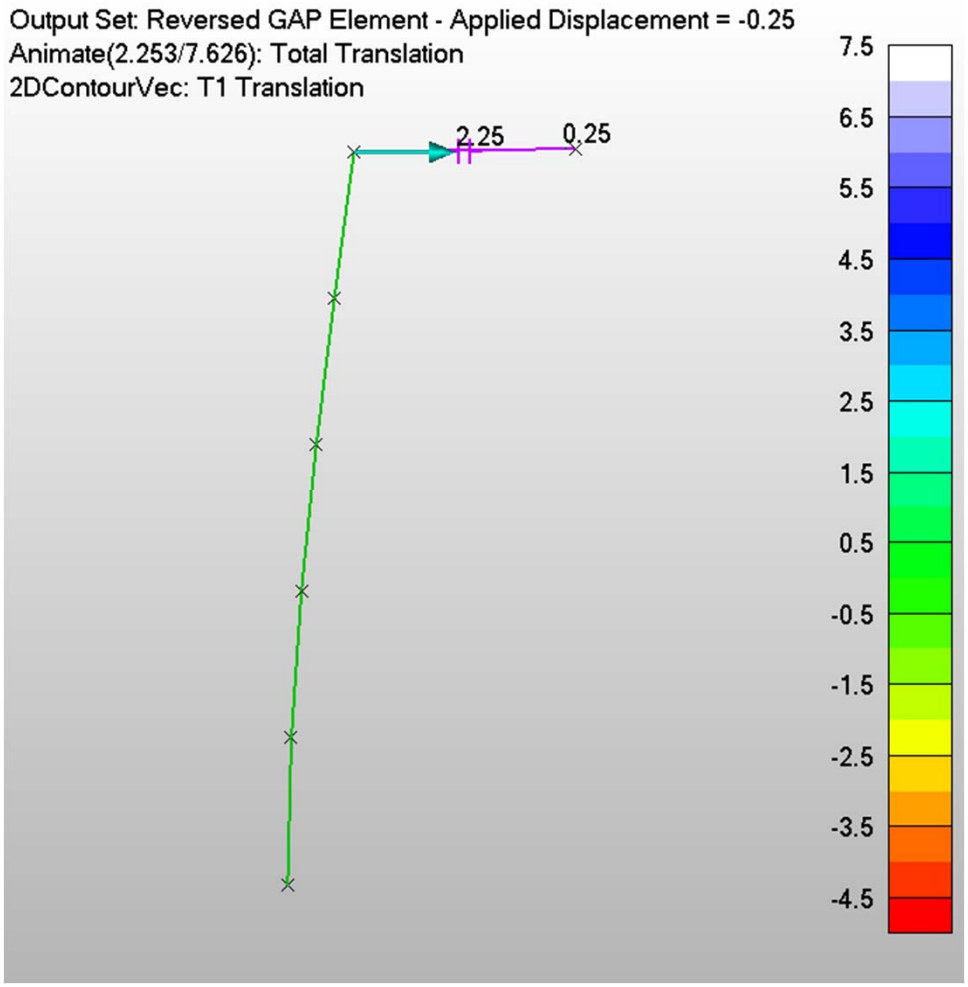


Unrestricted © Siemens AG 2013 All rights reserved.

GAP Element Example – Use of Orientation Vector and Initial Gap Distance

Applied Displacement of 0 to -5 to +5 to 0 applied to the right end of the GAP element

- GAP element is oriented with the X-Axis of the Global Rectangular CSYS
- GAP element is reversed and the **Initial Gap** is set to -2.5
- Note how the top of the beam starts to displace after the GAP displacement = 2.5



Unrestricted © Siemens AG 2013 All rights reserved.

NX Nastran Advanced Nonlinear Statics and Transient vs. Advanced Nonlinear Explicit

NX Nastran Advanced Nonlinear Static / Advanced Nonlinear Transient (SOL 601)	NX Nastran Advanced Nonlinear Explicit (SOL 701)
For statics and low frequencies dominate the dynamic response of the structure. Examples: crush analysis, earthquake response	For problems of highly dynamic, short duration events. Examples: wave propagation, high speed impact
Unconditionally stable, so larger time steps can be used	Conditionally stable; need small time steps
Equilibrium satisfied; greater confidence in solution	More difficult to assess quality of solution
Assembly and solution of matrix with equilibrium iterations; computational effort per step is high	No matrix assembly and no iterations; computational effort per step is low
Requires more memory	Less memory for same mesh
All supported elements can be used	Certain elements and materials not supported
Solution may fail to converge in equilibrium iterations	Analysis may fail due to diverging solution

Unrestricted © Siemens AG 2013 All rights reserved.

NX Nastran Advanced Nonlinear Solver - Range of Analyses

	Linear	Nonlinear Analysis			
Displacement	Small	Small	Small	Large	Large
Strain	Small	Small	Small	Small	Large

Default is small displacement, small strain

Large strain (PARAM,LGSTRN,1) automatically implies large displacement (PARAM,LGDISP,1)

- Set in the Analysis Set's *NASTRAN Bulk Data Options* dialog box.

Large strain formulation is used only for 3D solid, plane strain, axisymmetric and shell elements

Hyperelastic material models (MATHP and MATHE) automatically use large strain formulation

Unrestricted © Siemens AG 2013 All rights reserved.

Which Advanced Nonlinear Solution Should You Use?

SIEMENS

In general, use Advanced Nonlinear Statics or Advanced Nonlinear Transient

- Use Advanced Nonlinear Transient if inertia effects are important – otherwise use Advanced Nonlinear Statics
- If SOL 601 fails to converge, use SOL 701 and compare results at the last SOL 601 converged step
- You can also do a restart of a Sol601 solution in Sol701

Note that for certain classes of problems, such as drop tests and metal forming, both solutions may be comparable in performance

Unrestricted © Siemens AG 2013 All rights reserved.

SIEMENS

Advanced Nonlinear Explicit Notes

Several NX Nastran elements supported for Sol601 are not supported in Sol701:

- Plane strain and axisymmetric elements
- Higher-order elements parabolic elements are also not supported such as 10-node tetrahedrals, 20-node bricks, and 8-node shells

Reduced integration elements with hour-glassing should not be used. Using such element formulations can have adverse effects on the accuracy of the solution.

Gasket and creep materials are not supported. For hyperlastic materials, Mooney-Rivlin and Ogden models are supported but not Arruda-Boyce and hyperfoam models.

Nearly incompressible material will significantly reduce the stable time step size. The compressibility of the material can be increased (e.g., reducing bulk modulus of hyperelastic materials) to achieve a reasonable time step size

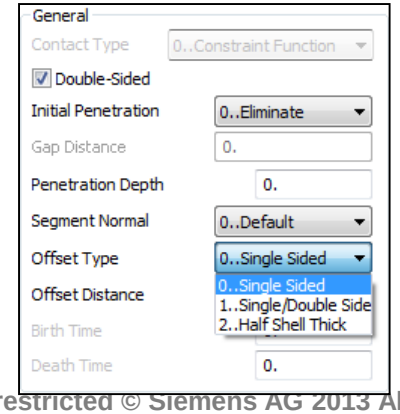
Glued contact and pre-load bolt features are not available

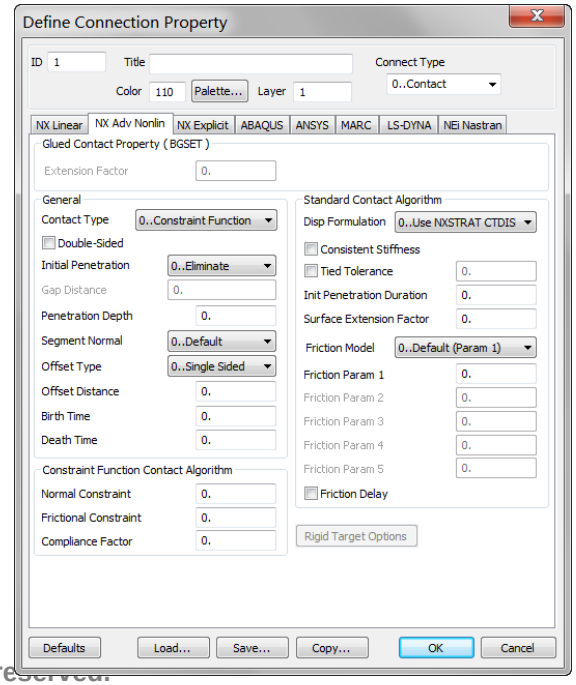
Unrestricted © Siemens AG 2013 All rights reserved.

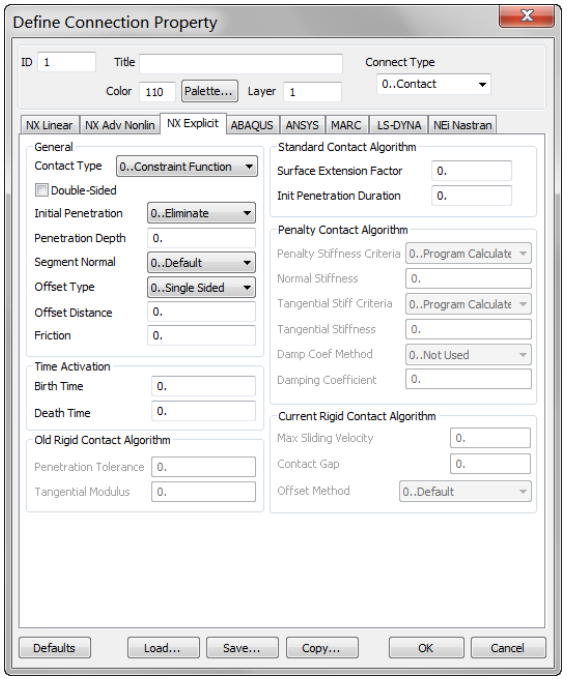
NX Nastran Advanced Nonlinear - Setting Connection Properties

To set the *Connection Property* for Advanced Nonlinear problems involving contact, you will need to select either the **NX Adv Nonlin** tab or the **NX Explicit** tab in the *Define Connection Property* dialog box.

- Specify the **Contact Type** as **Contact** or **Glued** for Sol601.
- For details on these settings, see Sections 4.4.3.2 and 4.4.3.3 of the *Femap Commands* manual.
- Set **Double-Sided** and **Offset Type** options for planar element contact





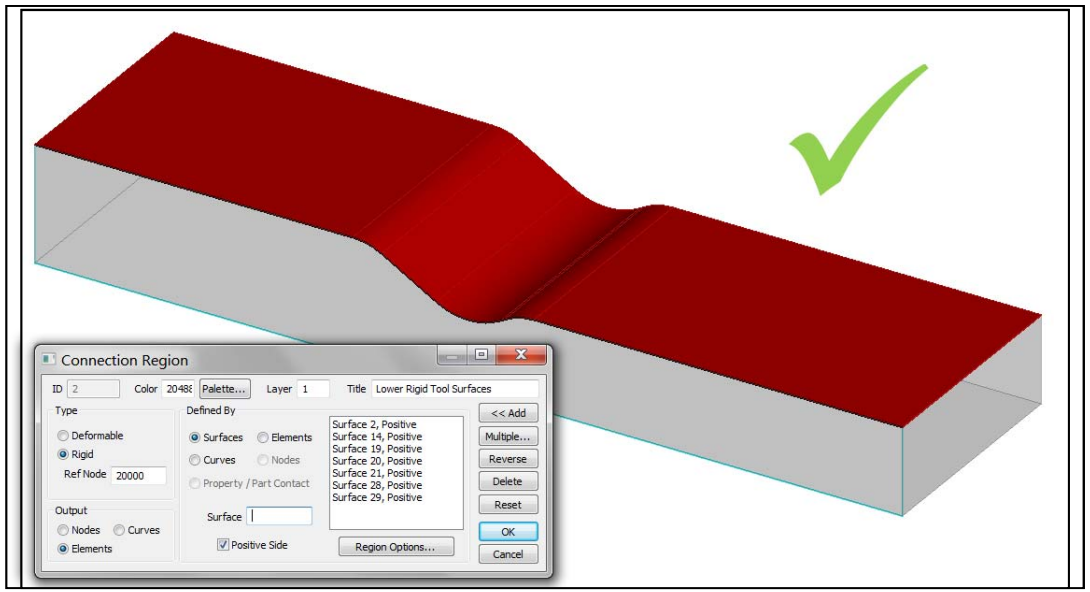


Unrestricted © Siemens AG 2013 All rights reserved.

Advanced Nonlinear Connection Regions

Advanced Nonlinear supports both flexible and RIGID Connection Regions:

- Both regions can be modeled with either Surfaces or Element Faces
  - Correct specification of positive or negative directions are critical
- A single Connection Region should be tangent continuous within a reasonable tolerance – i.e, no sharp discontinuities



Unrestricted © Siemens AG 2013 All rights reserved.

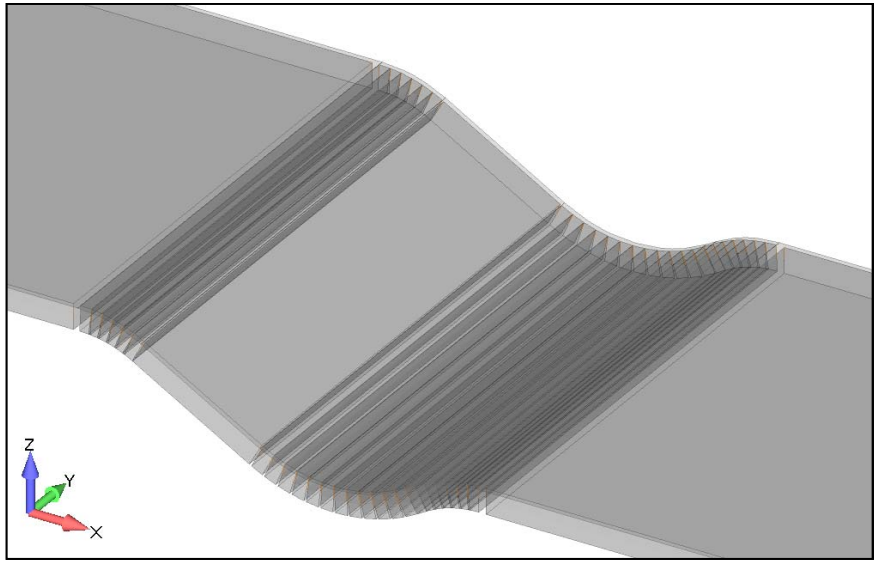
Rigid Connection Regions

RIGID regions require a Reference Node and can only have Enforced Displacement or Enforced Rotation loads applied to the Reference Noden

RIGID regions internally are treated as surfaces, therefore, mesh size, aspect ratio and distortion on planar regions is not critical – e.g. a planar surface can be meshed as a single element

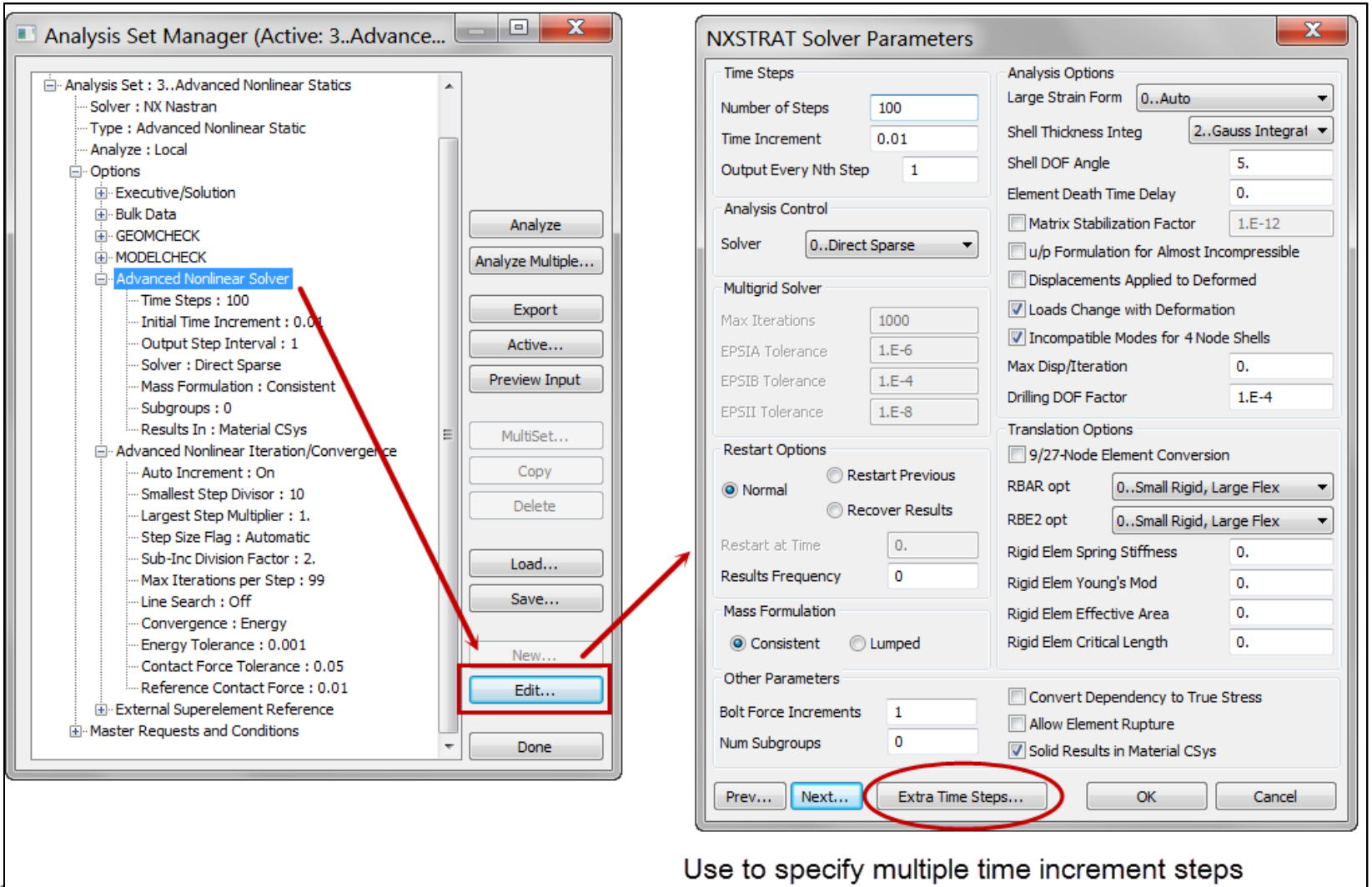
Offsets and shell thicknesses are ignored, use for display purpose

- Femap requires the assignment of a material to the solid or shell element faces and a positive thickness attribute for shell elements



Unrestricted © Siemens AG 2013 All rights reserved.

Advanced Nonlinear Static/Transient Analysis
NXSTRAT Solver Parameters



Use to specify multiple time increment steps

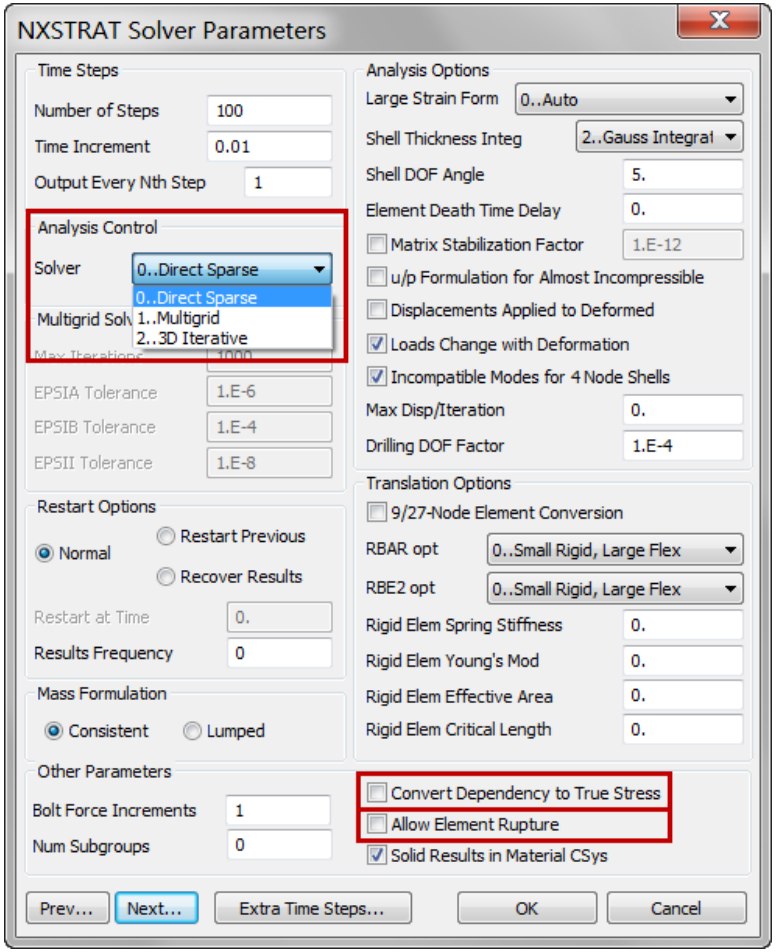
Advanced Nonlinear Static/Transient Analysis
NXSTRAT Solver Parameters

Specify the **Solver** as one of:

- **Direct Sparse** – default solver
- **Multigrid** – use for very large models with Solid Tetrahedral elements only
- **3D Iterative** – use for models with mainly higher order solid elements

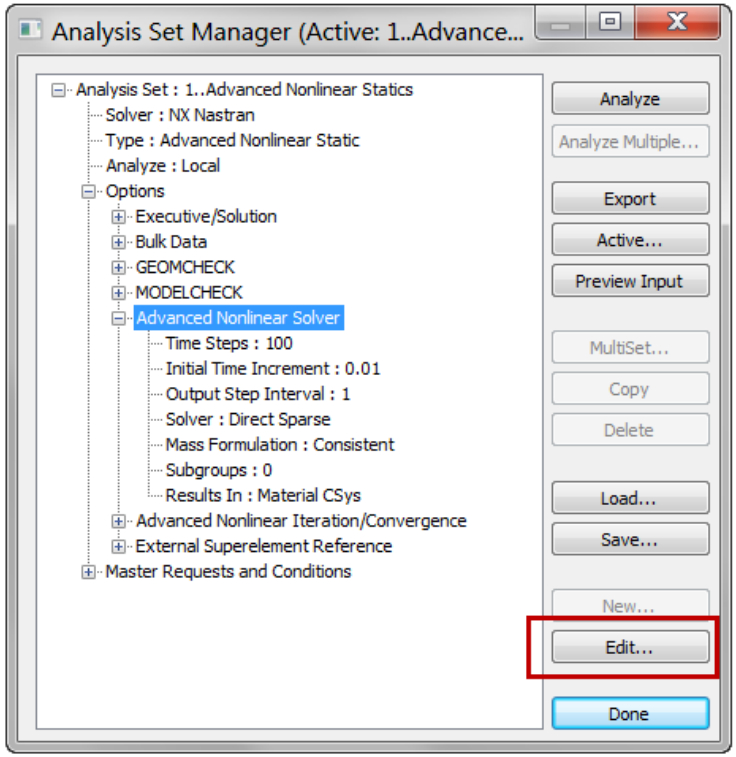
Convert Dependency to True Stress can be used when Material attributes are defined using true Stress/Strain values

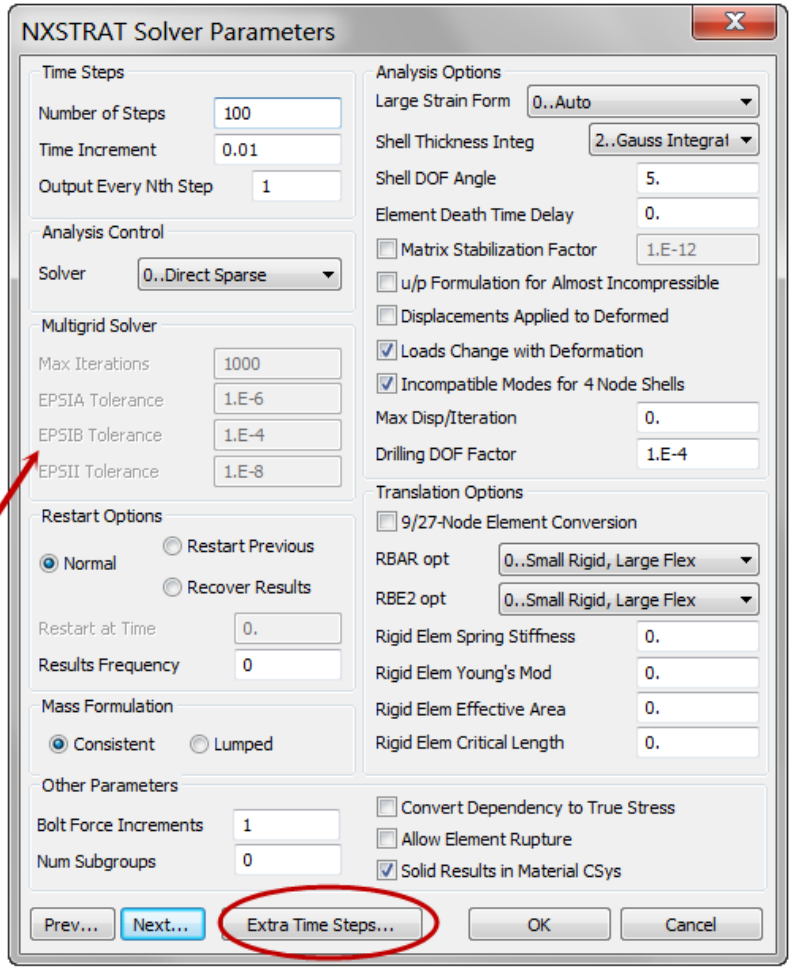
Allow Element Rupture – when element stress exceeds max allowable, element is removed from subsequent solution time steps (including Stiffness, Mass, etc.)



Unrestricted © Siemens AG 2013 All rights reserved.

Advanced Nonlinear Analysis Convergence Control – SOL601

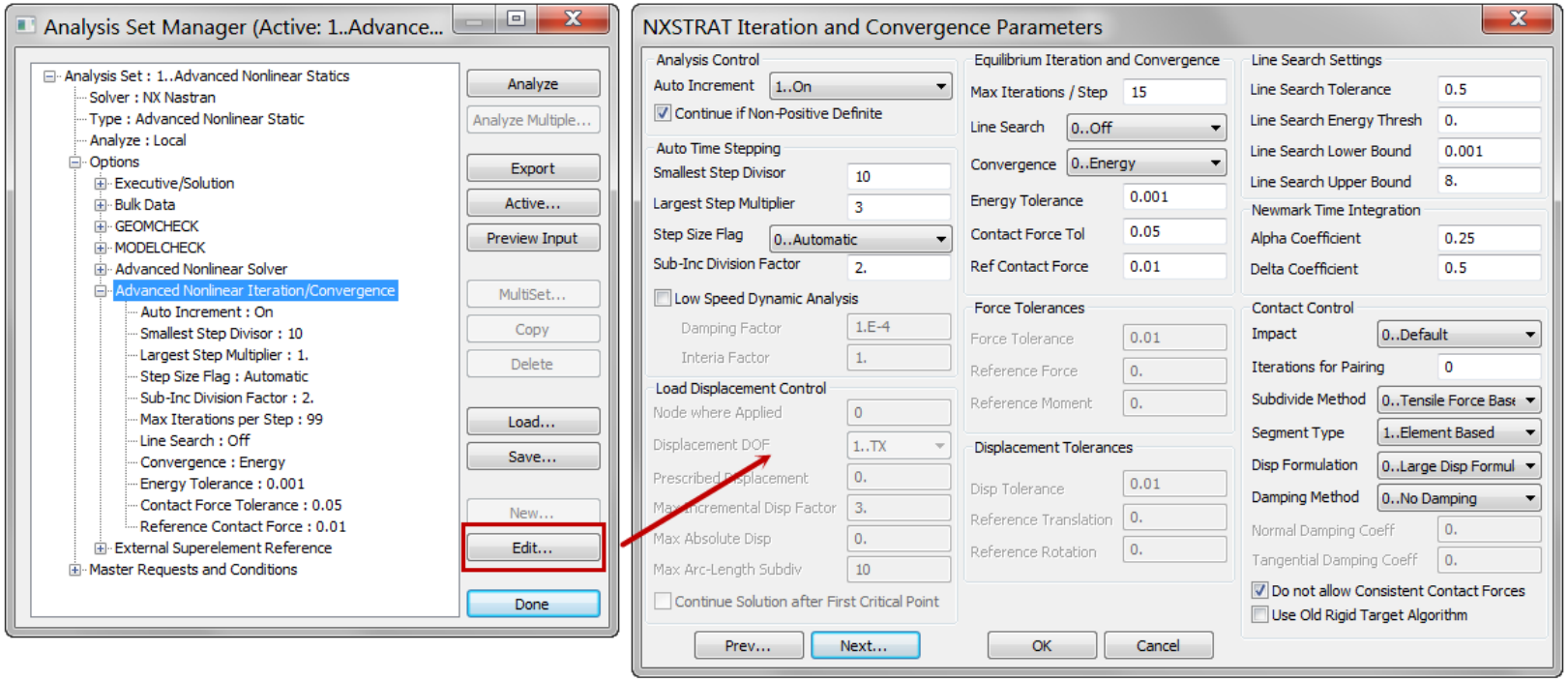




Use to specify time steps with differing **Time Increments**

Unrestricted © Siemens AG 2013 All rights reserved.

NX Nastran Advanced Nonlinear Analysis Control – SOL701



Unrestricted © Siemens AG 2013 All rights reserved.

SIEMENS

Examples and References

Femap Examples for NX Nastran Nonlinear:

- Example 25: Plastic Deformation of Rod - Nonlinear Material
- Example 26: Gap Contact - Cantilever Beam
- Example 27: Large Deformation – Cantilever Beam
- Example 28: Slide Line Contact – Hyperelastic Seals
- Example 29: 3-D Contact - Plastic Clip and Base

Femap Examples for NX Nastran Advanced Nonlinear:

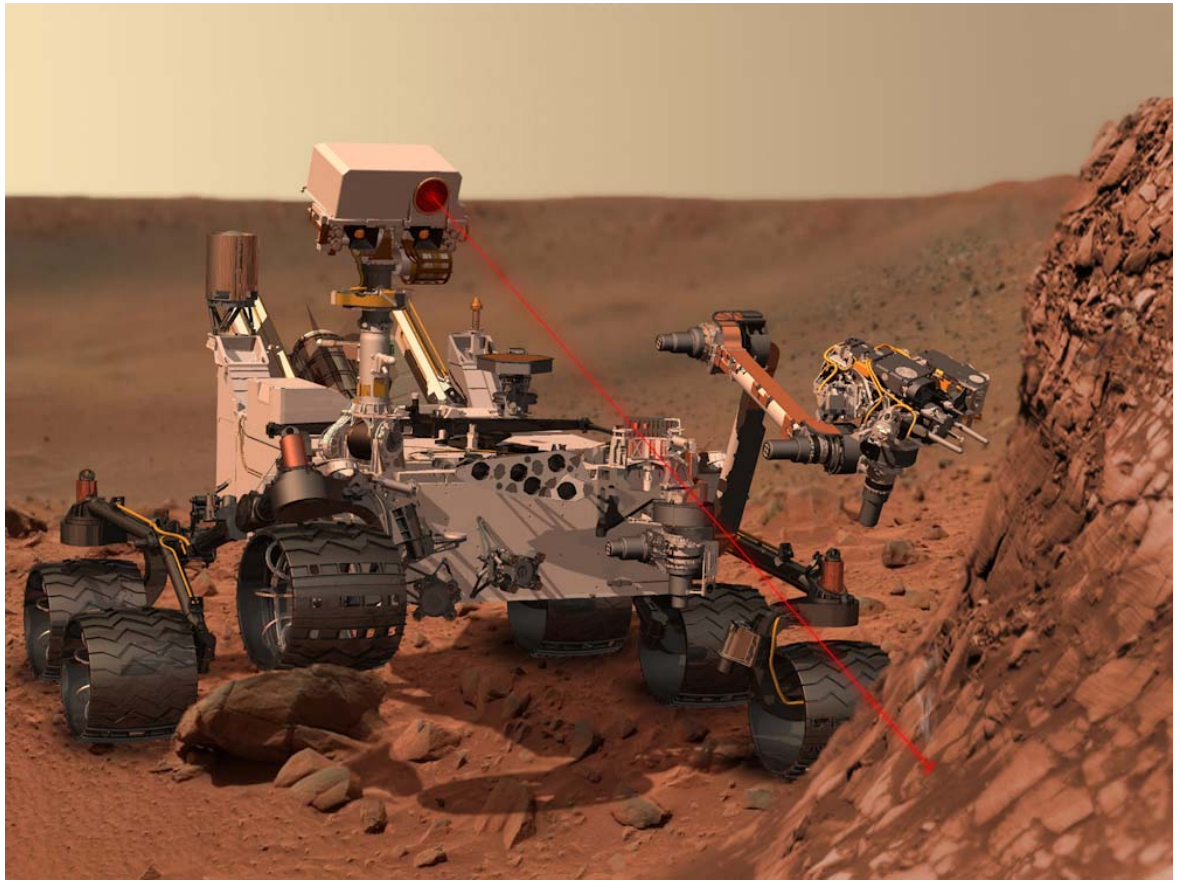
- Example 30: Large Deformation – Advanced Nonlinear (SOL 601)
- Example 31: Surface to Surface Contact - Advanced Nonlinear (SOL 601)
  - Same as NX Nastran Example 29, except uses Advanced Nonlinear solver

NX Nastran User Guides

- Handbook of Nonlinear Analysis (SOL 106)
- NX Nastran Basic Nonlinear Analysis User's Guide
- Advanced Nonlinear Theory and Modeling Guide

Unrestricted © Siemens AG 2013 All rights reserved.

SIEMENS



Q&A

Thank you!

FEMAP SYMPOSIUM 2013
Mission Critical Design and Analysis

Unrestricted © Siemens AG 2013 All rights reserved.