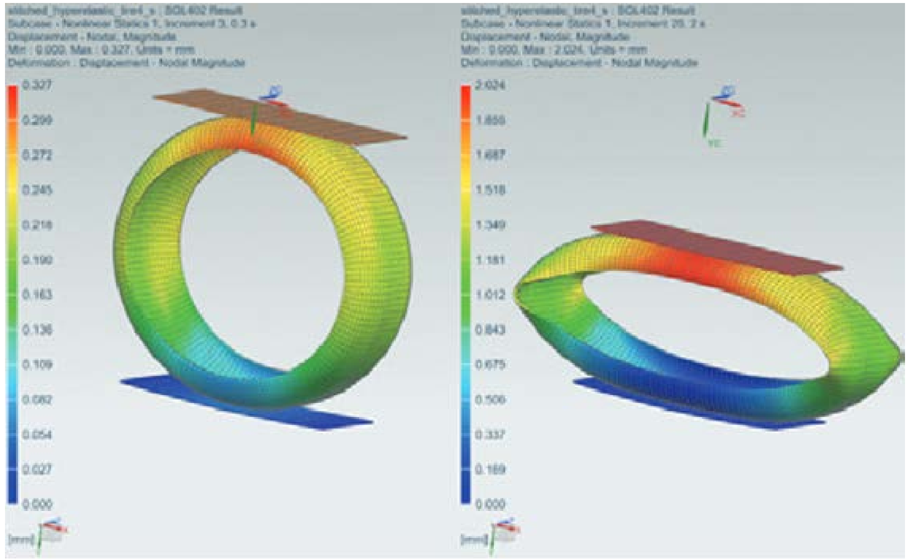
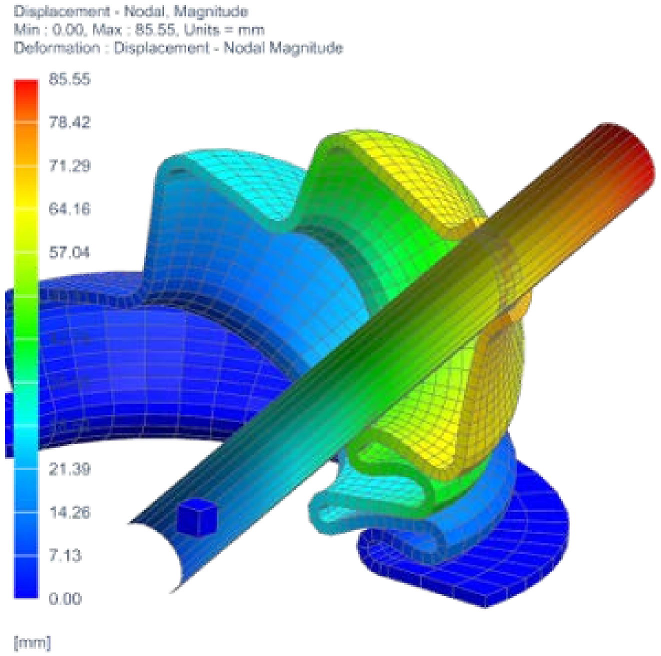
Getting Started with Simcenter Nastran Multistep Nonlinear Solutions

December 1, 2022

Your Host: Scott Thibault, ATA Engineering, Inc.

Your Speaker: Miles Hatem, ATA Engineering, Inc.





SIEMENS

San Diego • Los Angeles • Berkeley • Albuquerque • Denver • Huntsville • Washington D.C.

ata-engineering @ATAEngineering (858) 480-2000 www.ata-e.com

Who Are ATA Engineering?

We are an **employee-owned** small business with a **full-time staff of over 190**, more than 160 of whom are degreed engineers

38 Ph.D.

95 M.S.

28 B.S.

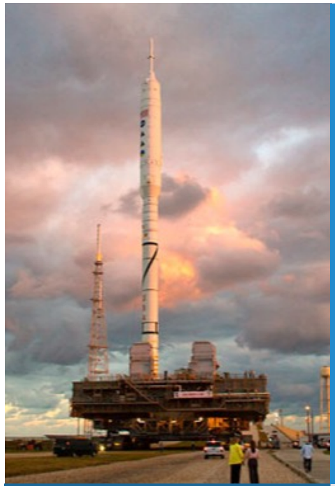
11 Registered Professional Engineers

Subject-matter experts recognized by

- Society of Experimental Mechanics
- American Society of Mechanical Engineers
- American Institute of Aeronautics and Astronautics

15 Average years of experience

ATA Engineering's **high-value engineering services** help solve the toughest product design challenges



Aerospace

Spacecraft

Hypersonics & Composites

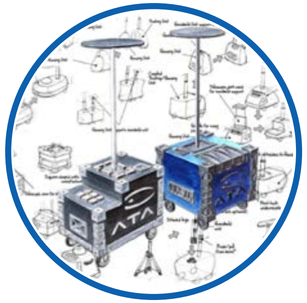
Defense

Industrial & Mining Equipment

Themed Entertainment

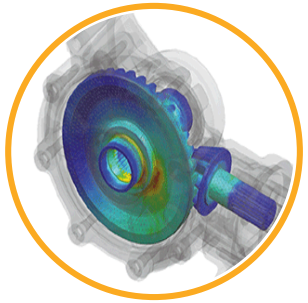
Our Services

We provide our customers with **complete, integrated solutions**



Design

From initial concept development to detailed structural design

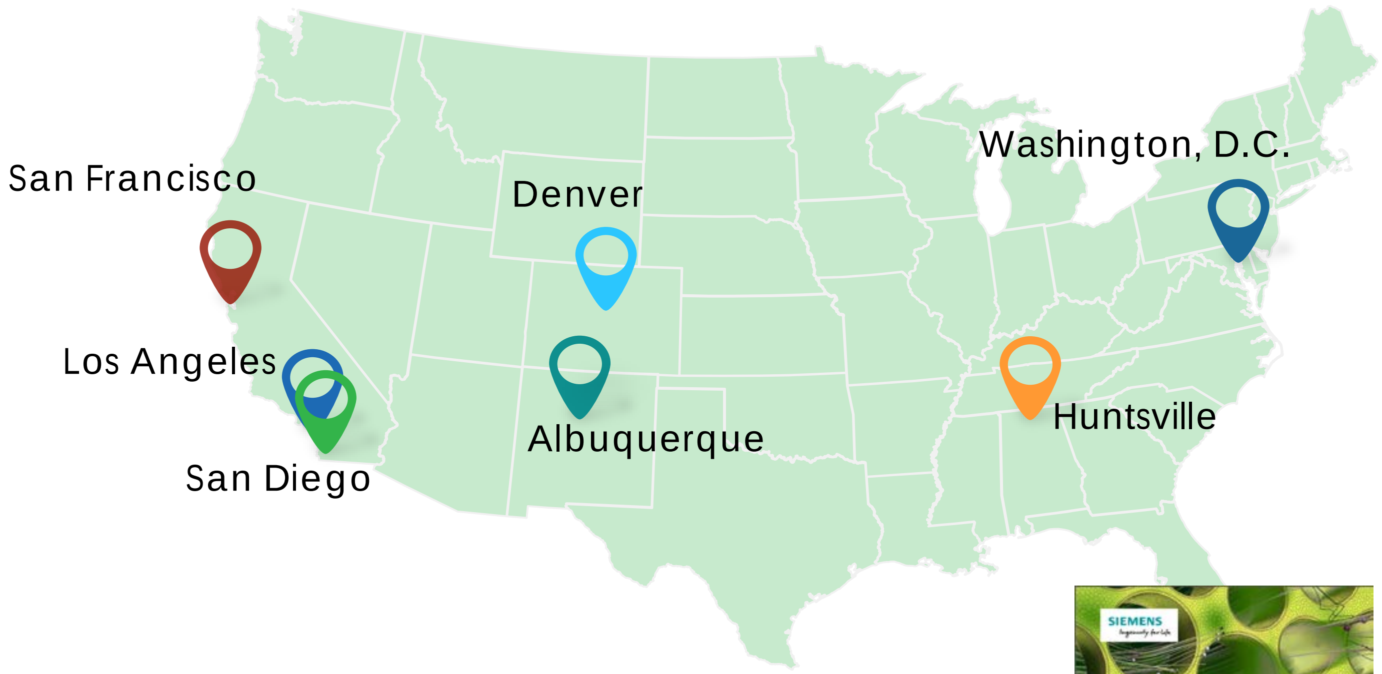


Analysis

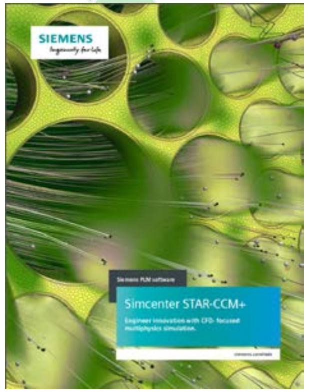
Comprehensive structural, fluid, acoustic, and thermal analysis services

Test

Industry-leading structural test services for extreme loading environments



ATA Engineering is also a **Siemens Platinum Level** Solution Partner.



Our Software Services

ATA is a Platinum-level Solution Partner for **Siemens Digital Industries Software**

Siemens Value Added Reseller

Hotline Support

Nastran Training Material

Software Implementation

Siemens Preferred Training Provider

Software Integration

ATA Engineering is recognized as a Certified Expert Partner with validated expertise in Femap, STAR-CCM+, and Simcenter 3D.

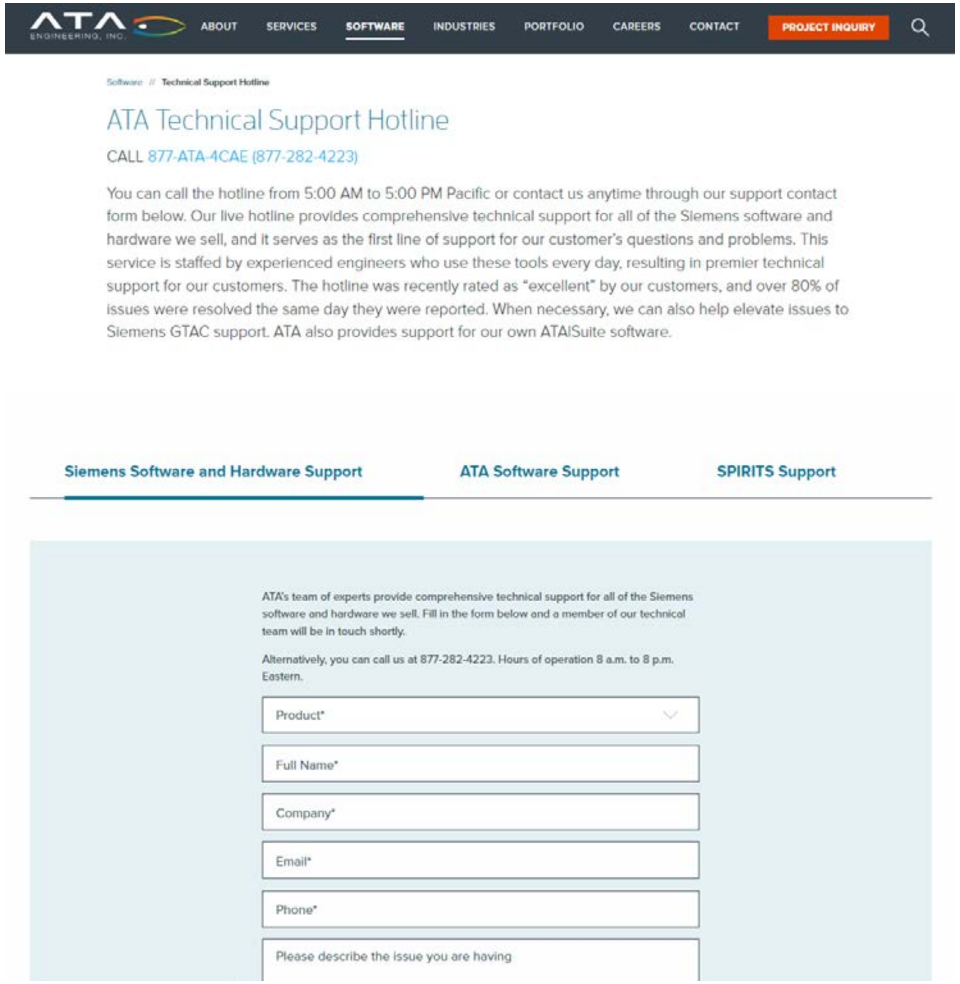
Expert Partner

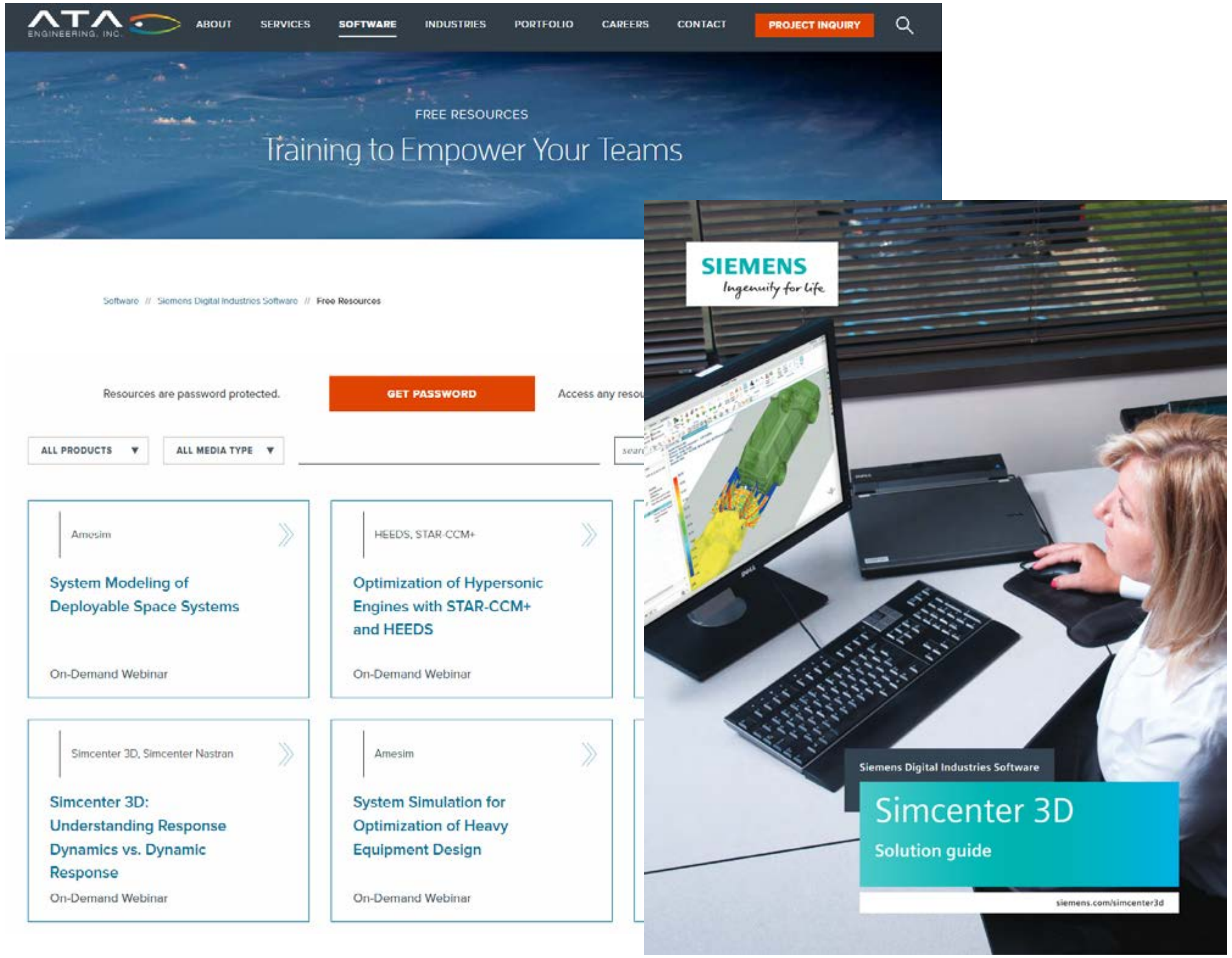
Digital Industries Software

SIEMENS

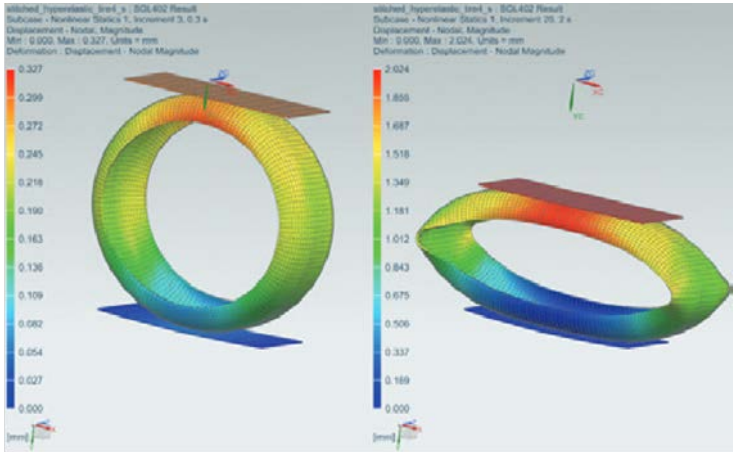
Visit Our Website for Product Information and Free Resources

https://www.ata-e.com/software/





Getting Started with Simcenter Nastran Multistep Nonlinear Solutions



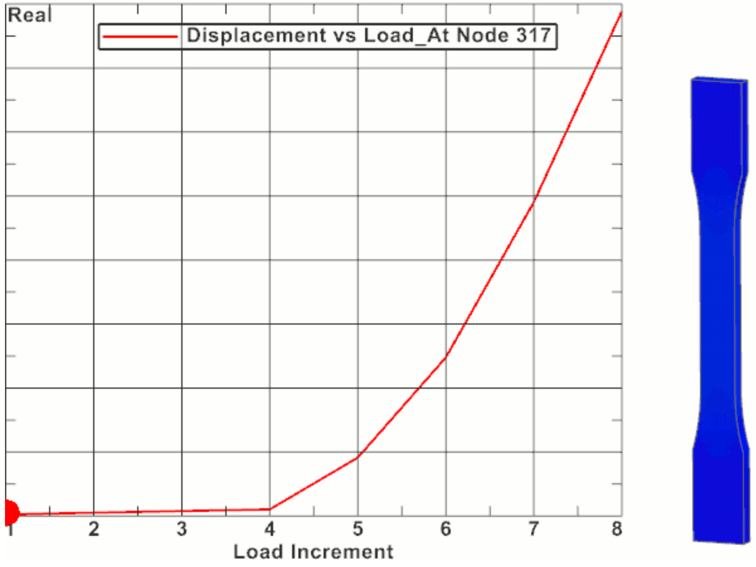
SPEAKER: Miles Hatem, Senior Project Engineer, ATA Engineering, Inc.

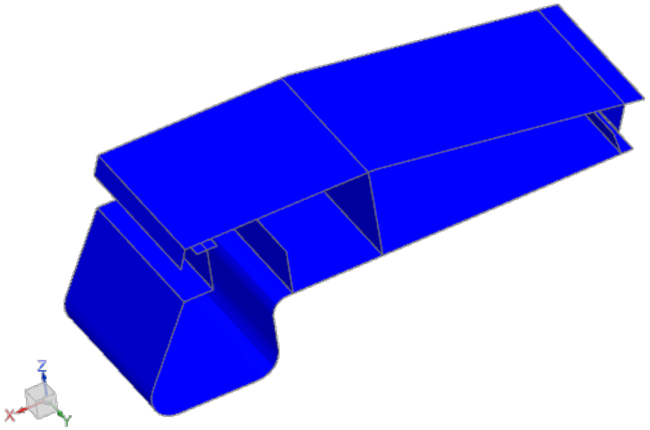
Mr. Hatem is an expert Simcenter 3D and Nastran user and the author and instructor for Siemens' official course on Simcenter 3D Multistep Nonlinear Analysis. As an ATA project engineer, he uses Nastran on a daily basis to deliver analysis-driven solutions to customers' most challenging engineering problems. He specializes in linear and nonlinear structural analysis as well as structural dynamics analysis. Before joining ATA, Mr. Hatem received his BS and MS from Purdue University, where he studied structural analysis of aeronautical and aerospace systems.

What does “Nonlinear Analysis” Mean?

The term ‘nonlinear analysis’ can be used to describe a number of different types of structural analysis

The one thing they have in common is that, as the name indicates, the structure does not respond in a linear manner to applied boundary conditions





How do linear and nonlinear solutions differ?

- A **linear analysis** is, generally speaking, one based on linear elastic theory:

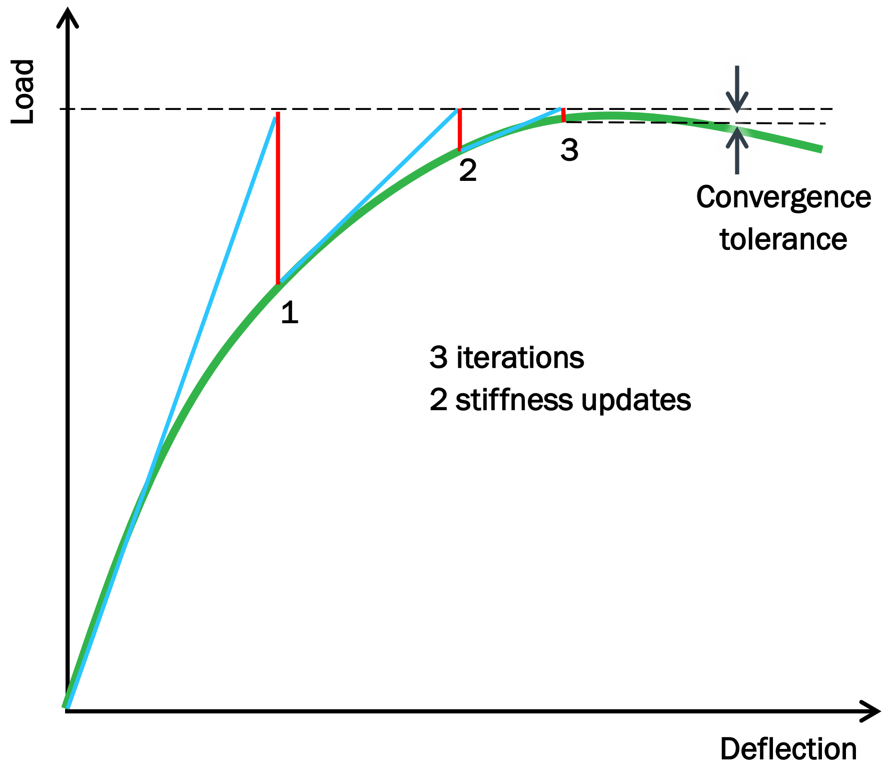
- Materials have a linear stress-strain relationship
- Displacements and strains are assumed to be small
- Stiffness is not altered by deflection
- Loads are independent of deformation

- **Nonlinear analysis** is used when a structure's deformation will not be directly proportional to applied load:

- Materials may have complex stress-strain relationships
- Displacements and strains can be very large
- Displacement can reduce or increase stiffness
- Loads can depend on deformation

Another major difference, nonlinear solutions are iterative

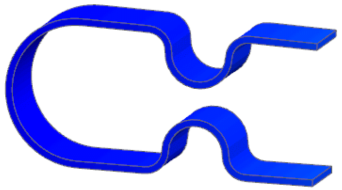
- Nonlinear solutions also differ from their linear counterparts in that they **iterate** to reach a **converged** state
- While iterating the solution may also be updating
  - Nodal positions
  - Stiffnesses
  - Load directions
  - Material properties
  - Etc.

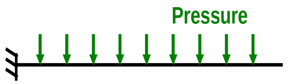


Reasons to consider using a nonlinear solution

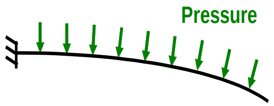
- Structure is loaded highly enough to push materials past yield

- Contact conditions

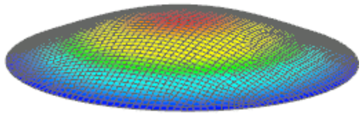




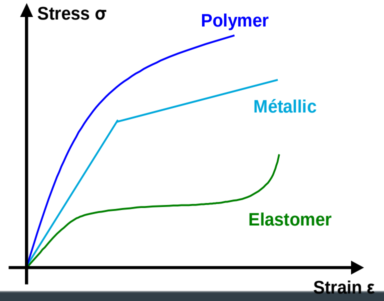
- Deflections will be large enough that load directions should be updated



- Structure will stiffen with deformation (e.g. drum-head)



- Materials behave nonlinearly (large strain expected, hyperelastic, etc.)



Categories of Nonlinearity

1. Geometric (aka Large Displacement)

- Geometric nonlinearity describes conditions in which motion from the initial state is no longer negligible

2. Contact

- Contact is exactly what it sounds like, two or more bodies exerting compressive and/or frictional forces on each other when touching, but able to separate under tension

3. Material

- Material nonlinearity exists when any material does not respond proportionally to load

Brief comparison of Simcenter Nastran nonlinear capabilities

SOL	101	106/129	401	402	601	701
Solver base	DMAP	DMAP	DMAP	Samcef	Adina	Adina
Contact	Linear	Gap El.	✓	✓	✓	✓
Bolt Preloads	✓		✓	✓	✓	✓
Large Displacements		✓	✓	✓	✓	✓
Large Strain			Solids	✓	✓	✓
NL Elasticity (small strain)		✓	✓		✓	✓
Hyperelasticity (large strain)		Limited		✓	✓	✓
Plasticity		✓	✓	✓	✓	✓
Creep		Limited	✓	✓	✓	✓
Implicit statics/dynamics		✓	✓	✓	✓	
Explicit dynamics						✓
Subcase chaining		Required	User Choice	User Choice		
Tangent stiffness modal			✓	✓		

DEPRICATED

There are two Multistep Nonlinear solutions, which should I use?

	SOL 401	SOL 402
Strongest Capabilities	• Thermal / mechanical / flow cosimulation	• Large rotations • Large strains
Currently recommend for problems containing	• Turbomachinery • Multi-physics • Multi-step nonlinear	• Complex contact • Complex dynamics • Formability • Rubbers • Kinematic Joints

- Analysis types not explicitly mentioned are generally good in either solution sequence
- Keep in mind that these are only suggestions; you should make the engineering choice of which solver to use based on what your problem requires

Creating a SOL401 run from SOL101 is easy

- Step 1: 1 → 4
- Step 2: Add subcase and specify type (static / dynamic) and TSTEP card
    - Subcase not strictly required (ANALYSIS=x statement can be set global for single load)
    - Recommended for organizational purposes
- Step 3: Add TSTEP1 card somewhere in BULK

```
SOL 101
CEND
DISPLACEMENT(PLOT) = ALL
SPC = 1
LOAD = 1
BEGIN BULK
GRID,1,0,0.,0.,0.,0
…etc.
```

```
SOL 401
CEND
DISPLACEMENT(PLOT) = ALL
SUBCASE 1
  ANALYSIS = STATICS
  TSTEP = 1
  SPC = 1
  LOAD = 1
BEGIN BULK
TSTEP1,1,1.,10,YES *
GRID,1,0,0.,0.,0.,0
…etc.
```

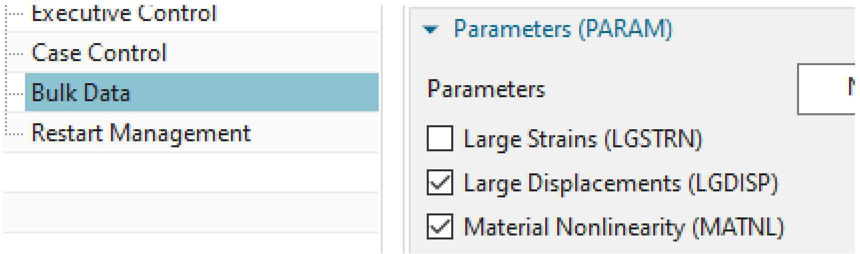
*(1.0 sec subcase, with 10 increments, and output for each increment called out in this TSTEP1 card)

There are three primary types of nonlinear subcases

- Static (ANALYSIS = STATICS)
  - Nonlinear static solution
  - Solves nonlinear $\{F\} = [K_T]\{u\}$ solution (no velocity or acceleration)
  - Time is only used as scale factor on applied loads or contact interreference
- Preload (ANALYSIS = PRELOAD)
  - Really a subset of static
  - Zero-time subcase (must end at end time of previous subcase, or 0 if first)
  - Loads in are propagated through rest of solution
- Dynamic (ANALYSIS = DYNAMICS)
  - Nonlinear dynamics solution (implicit)
  - Solves nonlinear $\{F\} = [K_T]\{u\} + [C]\{u'\} + [M]\{u''\}$ solution
  - Time is real-world time
- 401 & 402 also have modal subcases which can be added anywhere in the solution to calculate “snapshot” modes (or buckling for 402) during the nonlinear subcase sequence

Adding nonlinearities to your nonlinear model

- Some global parameters are needed to activate certain nonlinearities:
  - MATNL = material nonlinearity
  - LGDISP = large displacements
  - LGSTRN = large strains (> ~3%)
- Nonlinear material curve cards
  - MATS1 – older formulation (but still good for basic metals)
  - MPLAS – newer option with more material options
  - **Note: Even if you give Nastran a stress-strain curve, without PARAM,MATNL it won’t use it**



1	2	3	4	5	6	7	8	9	10
MATS1	MID	TID	TYPE	H	YF	HR	LIMIT1	LIMIT2	
	STRMEAS								

1	2	3	4	5	6	7	8	9	10
MPLAS	MID	Name							
	"YIELD"	Y-TYPE	Y-PAR1	Y-PAR2	...				
	"ISOH"	I-TYPE	I-PAR1	I-PAR2	...				
	"KINH"	K-TYPE	K-PAR1	K-PAR2	...				

ATA ENGINEERING, INC.

Many, many options for controlling solutions

- Solution 401 and 402 have a large number of control parameters which can be changed to affect the behavior of your solution. These will be set in the bulk data section of your model using the following cards:
    - NLCNTLG – Global nonlinear control parameters (401 and 402)
    - NLCNTL – Subcase nonlinear control parameters (401 only)
    - NLCNTL2 – Subcase nonlinear control parameters (402 only)
    - BCTPARM – Contact control parameters (401 only)
    - BCTPAR2 – Contact control parameters (402 only)
    - BCRPARA – Contact region parameters (401 and 402)
- All of these cards have *numerous* parameters which may be set, and will be referred to throughout this presentation, but all follow the general format as follows:

CARD	ID	PARAM1	VALUE1	PARAM2	VALUE2	PARAM3	VALUE3	
	PARAM4	VALUE4	PARAM5	VALUE5	--etc--			

Commonly used control parameters
Common to both SOL 401 and SOL 402

- Contact (BCTPARM or BCTPAR2)
  - INIPENE – Change initial penetration behavior (remove interference/gaps)
  - FRICMOD – Change friction model
- Solution control (NLCNTL or NLCNTL2)
  - DTINIT – Manually set initial time step size (doesn't work for ANALYSIS=PRELOAD as they are a zero-time subcase)
  - PLASTIC – Disable plasticity for a subcase (I often do this for bolt preload subcases)
- A note on rigid behavior
  - 401 has additional options for the RIGID case control option so it can be made to handle large displacements better
    - LINEAR – 101-type behavior
    - STIFF – Forces internal conversion to stiff beams
    - NONLIN – Elimination method (includes large rotations)
    - AUTO – Software chooses (default) based on LGDISP setting between LINEAR and STIFF
  - 402 always converts rigids to MPCs internally to the Samcef solution with a formulation that handles large displacements well

Commonly used control parameters
SOL 401 Only Parameters

- Contact Parameters (BCTPARM)
  - AUTOSCAL – Normal contact stiffness/compliance scale factor
    - Solution auto-calculates contact stiffness/compliance
    - This is a scale factor on that auto-calculated value, e.g. AUTOSCAL, 0.1 = 10% of auto-calc. stiffness
  - PENTYP – Change contact formulation
    - Compliance (units = [1/length], default)
    - Stiffness (units = [force/(length x area)])
- Solution Parameters (NLCNTL)
  - MSTAB – Enables matrix stabilization
    - Default factor $1E^{-10}$, set with MSFAC
  - PLLIM – Limits step size by an amount of allowed plasticity change
  - EPSBOLT – Bolt preload convergence tolerance (defaults to 0.001 or 0.1%)

Commonly used control parameters
SOL 402 Only Parameters

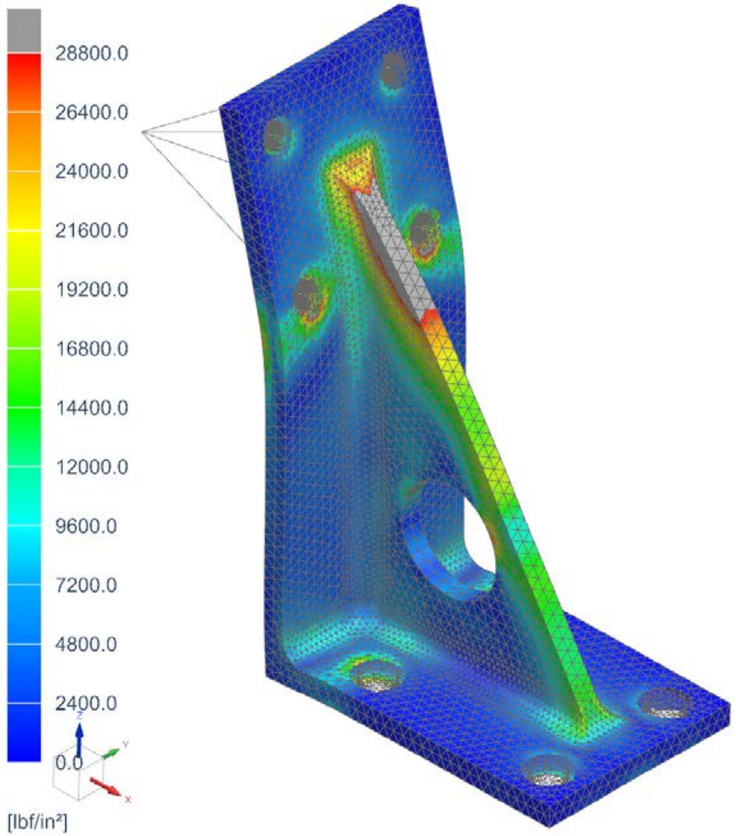
- Contact (NLCNTL2)
  - NSIDE – Can toggle on double-sided contact for shells
  - PDEPTH – Basically a search distance (but really an anti-search distance)
- Solution control (BCTPAR2)
  - INLS – activate line search algorithm (my go to for first thing for an unstable solution)
    - Positive values use energy method, negative values use force method
    - Integer value is how often it is activated (# of iterations)
  - ITMA – Max iterations per time step (default 10, a bit low in my opinion)

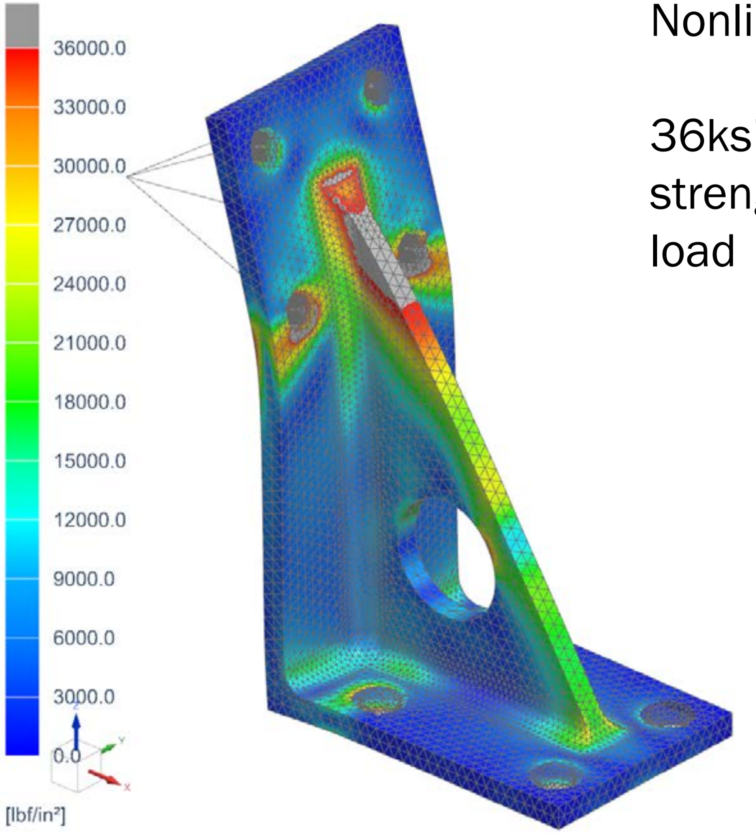
Demo

Converting a linear solution showing above-yield stress to a nonlinear plasticity solution in NX

Linear Analysis

36ksi yield strength with 1.25 FS





Nonlinear Analysis

36ksi yield strength, FS in load

Q&A

For More Information, Contact:

Scott Thibault
Manager, Business Development

ATA Engineering Inc.
Southeast Regional Office
308 Voyager Way NE, Ste 102
Huntsville, AL 35806

Phone: (256) 850-3856
Mobile: (802) 296-1617

E-mail: scott.thibault@ata-e.com

Smarter decisions. Better products. Faster.

Contact Us

13290 Evening Creek Drive
San Diego, CA 92128

(858) 480-2000

sales@ata-e.com

www.ata-e.com

@ATAEngineering

ata-engineering