Geometric Nonlinear Analysis Using UAI-NASTRAN

April 1st, 1999

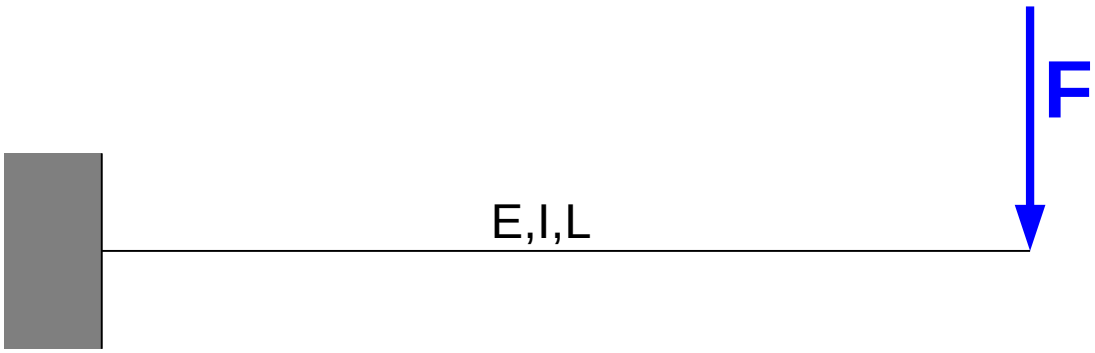
Jonathan L. Kuhn - Jonathan.Kuhn@gsfc.nasa.gov

Sebastien Lienard - slienard@pop500.gsfc.nasa.gov

Plan

- Background
  - Motivation (simple example)
  - Implementation (UAI-NASTRAN cards)
- Example of instability: 10th Scale Model of NGST Sunshield
  - Linear Results
  - Nonlinear Results
  - Modal Analysis With Updated Stiffness
- Example of Large Displacements: Constellation X PUD
  - Nonlinear Geometry Static Analysis
  - Modal Analysis With Updated Stiffness
- Conclusions

Example: Cantilever Beam



Types	**Linear**	**Material nonlinear**	**Geometric nonlinear**	**Material and Geometric nonlinear**
Static solution	F=K.X	F=K(X).X	F=K(X).X	F=K(X).X
Stiffness	$K = \frac{3.E.I}{L^3}$	$K(x) = \frac{3.E(x).I}{L^3}$	$K(x) = \frac{3.E.I(x)}{L^3}$	$K(x) = \frac{3.E(x).I(x)}{L^3}$
Assumption	Small displacements and rotations	Small displacements and rotations	Large displacements and/or rotations OR instability (small dX → large dK)	Large displacements and/or rotations

UAI-NASTRAN

- Reference: *User's Guide, Chapter 17*

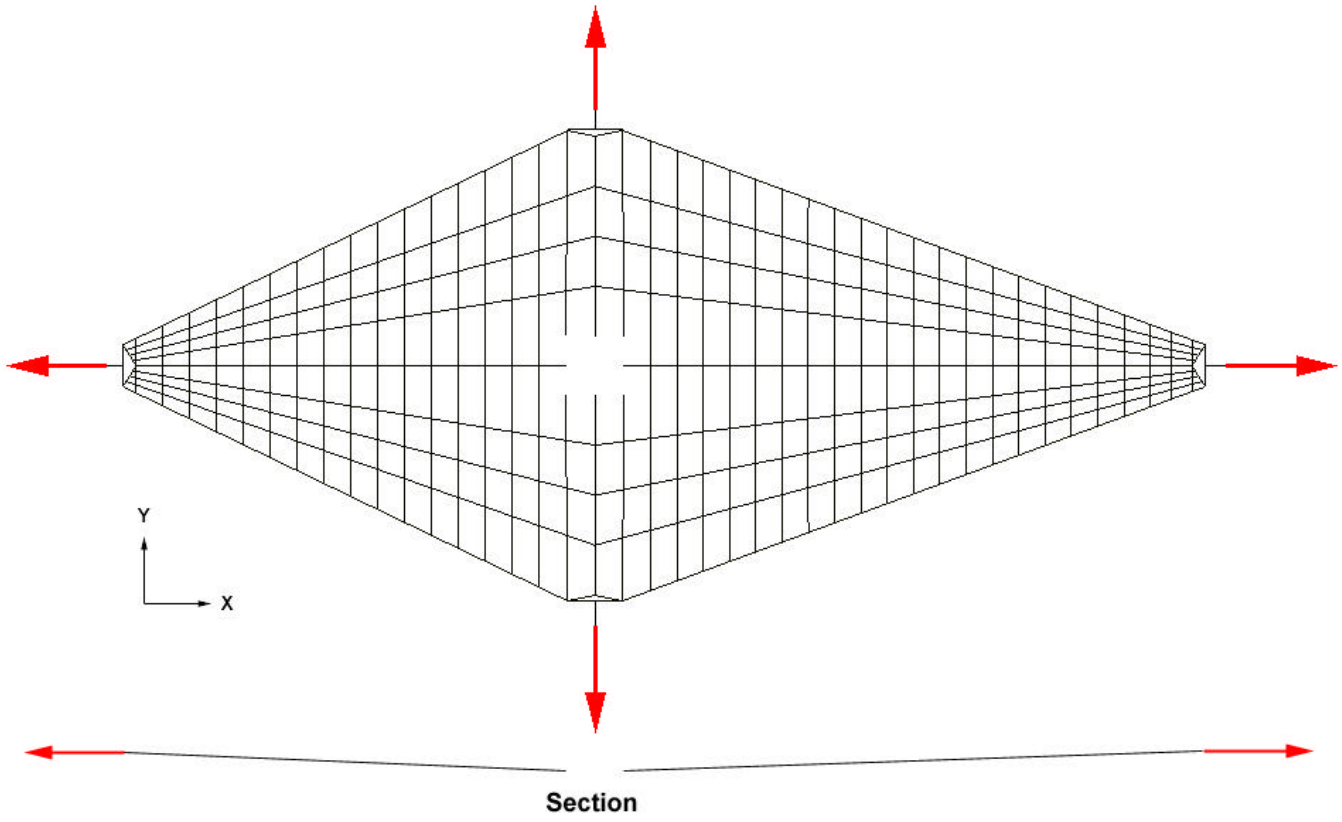
- *Exec control*
  - *APP NONLINEAR*
- *Case control*
  - *Solution: NLTYPE, STEP, NLSOLVE*
  - *Output: NLFORCE, NLSTRESS, NLSTRAIN...*
- *Bulk data*
  - *NLSOLVE*

Example: 10th Scale Model of NGST Sunshield

- NGST Sunshield (scale factor 1)
- ISIS Flight Experiment (scale factor 3)
- 10th Scale Model (scale factor 10)
  - Dimensions: 3.2x1.4x0.1m
  - Components:
    - Support structure
    - Four tubes in an orthogonal cross shape
    - 4 layers of thin film (12.7 microns)
    - Constant force springs applying pretension
  - Objectives:
    - Dynamic testing for model correlation
    - Natural frequencies
    - Damping coefficient
    - Mode shapes
  - *More details during peer review (April 6th, 1999)*

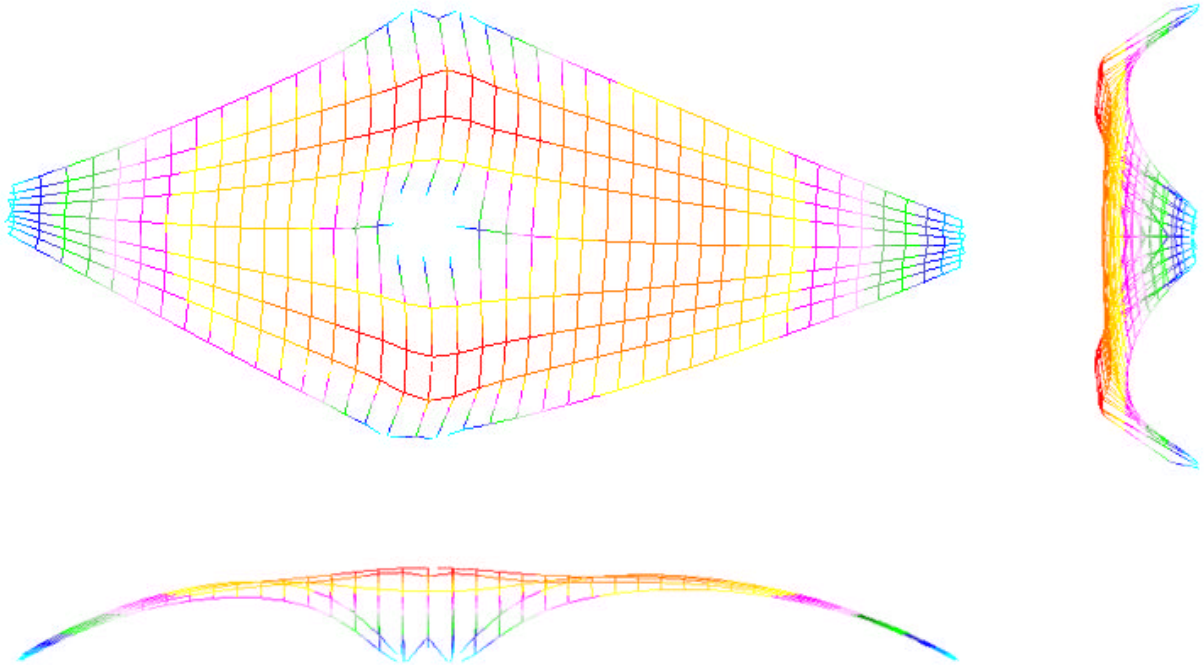
Finite Element Model

- FEM: Network of bar elements with various sections
- Constant forces at each corners (simulating constant force springs)
- Clamp at the base (8 nodes SPC(123))



Static Linear Solution

- Deformation



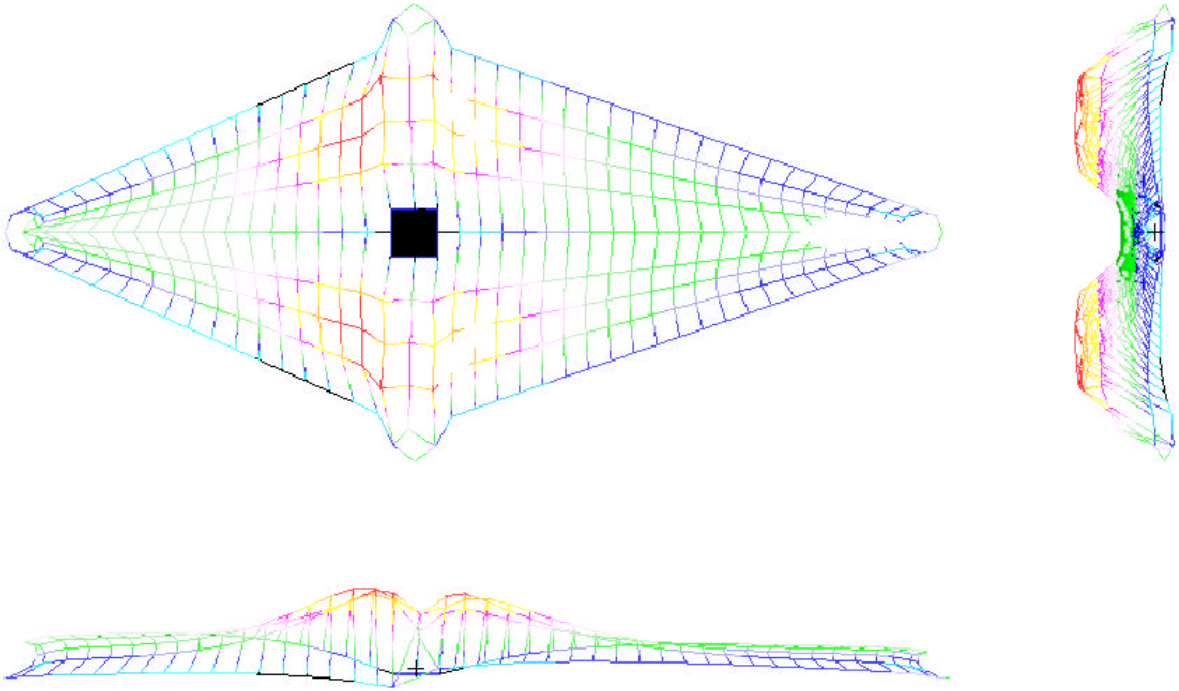
- Maximum displacement = 20.9mm
- *Note:* When the model was assembled the membrane did not touch each other: This result was not correct!

UAI-NASTRAN Cards

```
APP NONLINEAR                          <-- Non linear module
SOL 1                                  <-- Static solution
TIME 5
MEMORY 20MW
CEND
AUTOSPC( punch, SID=2, SPC, NOPRINT)=YES
POST PDA3 GLOBAL
ECHO = NONE
MAXLINES = 999999999
SPC = 2
DISPLACEMENT(NOPRINT)=ALL
SPCFORCES(NOPRINT)=ALL
STRESS(NOPRINT)=ALL
NLFORCE(NOPRINT)=ALL
NLTYPE = GEOM(STRAIN=SMALL)            <-- Type of non-linear analysis:
SUBCASE 1                                      geometric/material/both
  STEP 1                               <-- At least one step must be defined
    LOAD = 427
    NLSOLVE = 1                        <-- Select the NLSOLVE card in bulk
BEGIN BULK
NLSOLVE 1 ARC AUTO AUTO                <-- Select the algorithm and the controls
…
```

Static Nonlinear Solution

- Deformation



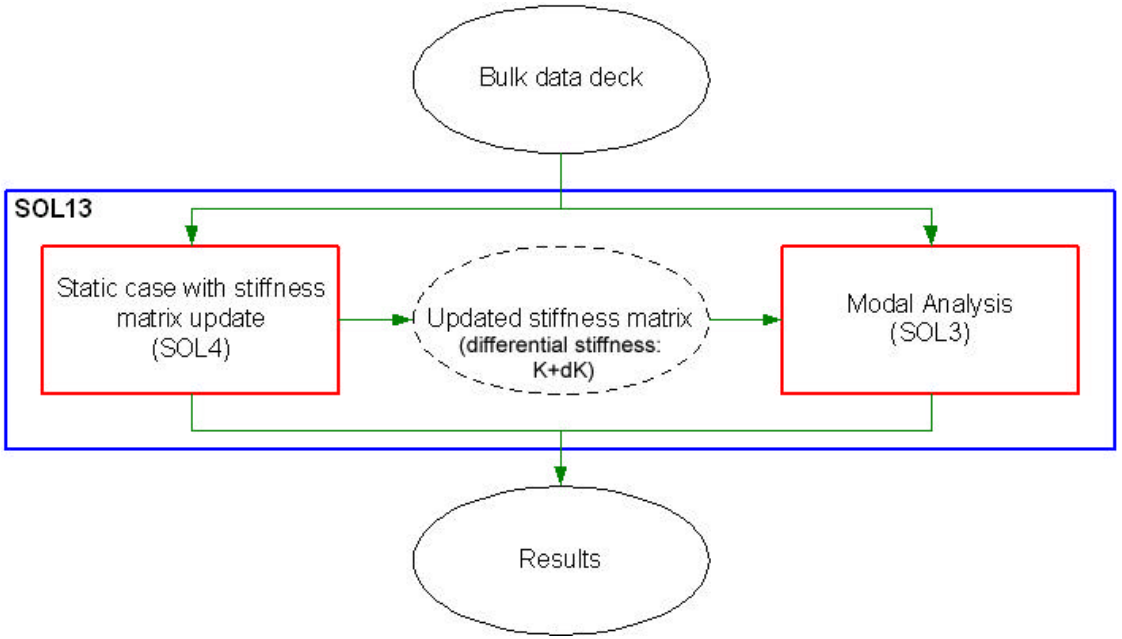
- Maximum displacement = 0.989mm

Modal Analysis With Updated Stiffness

- The purpose of modal analysis is to compute the natural frequencies of an undamped structure. Due to some preloading, the natural frequencies of the structure are modified. Depending on the loads, they can soften or stiffen the structure and then decrease or increase the natural frequencies. Therefore, they can not be ignored.

Modal Analysis With Differential Stiffness Linear Solution

- This solution is implemented in UAI-NASTRAN. Three cases have to be defined in the bulk data deck: the first solves a linear static case, then the second case updates the stiffness matrix to include differential strain energy stiffness terms to the linear (K+dK) and the third calculates the eigenvalues and extract the modes (For more information see Chapter 17 of the *UAI-NASTRAN Guide book*). This block diagram below presents the equivalent solutions preformed:



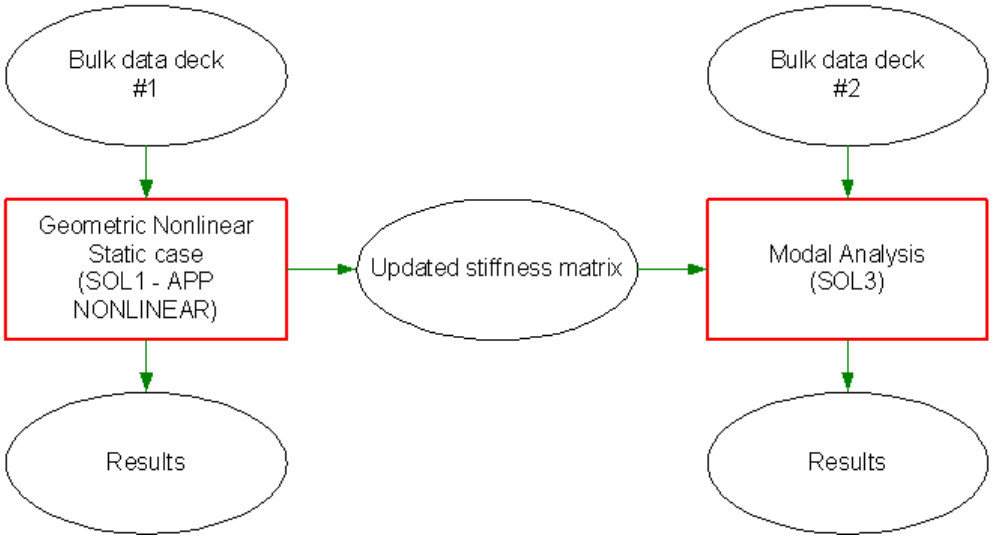
UAI-NASTRAN Cards

```
– SOL 13                                  <-- Modal analysis with
  TIME 5                                   differential stiffness solution
  MEMORY 20MW
  CEND
  AUTOSPC( punch, SID=2, SPC, NOPRINT)=YES
  POST PDA3 GLOBAL
  ECHO = NONE
  MAXLINES = 999999999
  SPC = 2                                 <-- Must be defined before
  DISPLACEMENT(NOPRINT)=ALL               the subcases
  SUBCASE 1                               <-- Static case
    LOAD = 427                            <-- Load card
  SUBCASE 2                               <-- Update of the stiff. matrix
    DSCOEFFICIENT=DEFAULT                 <-- Load coefficient
  SUBCASE 3                               <-- Modal analysis
    METHOD=1000                           <-- Select the method
  BEGIN BULK
  EIGR, 1000, LANCZ,0., , ,10             <-- Define the mode
                                          extraction method
  …
```

- *Note:* Here the DSCOEFFICIENT card is set to use the default value of the load factor (1.0). To change this value, a DSFACT card must be defined in the bulk section (see UAI-NASTRAN Reference).

Modal Analysis With Updated Stiffness Nonlinear Solution

- This solution is similar to UAI-NASTRAN SOL13. However, it is not implemented in UAI-NASTRAN as a solution by itself.
  The first step is to solve a geometric nonlinear solution and extract the last updated stiffness matrix. Then, during the modal analysis process, the stiffness matrix built has to be replaced by the nonlinear one recorded before. To do so, some *DMAP* cards have to be added in the bulk data decks. Indeed, two different files have to be generated and run independently. During each solving process, a matrix is either output or input and then replaced.



UAI-NASTRAN Cards: NL Static Solution

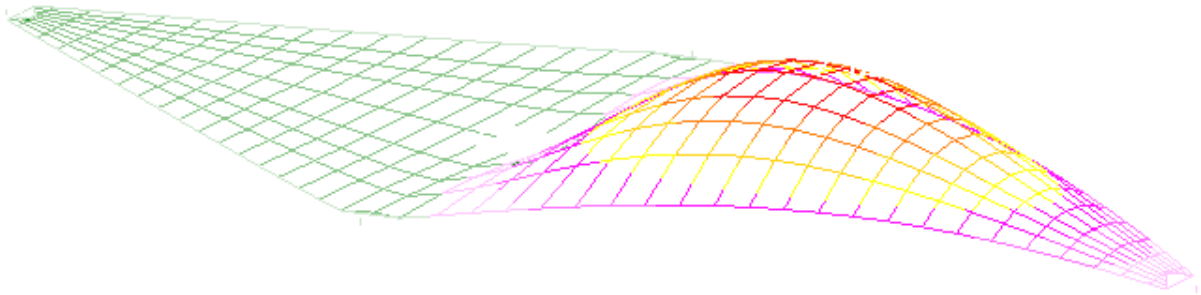
```
APP NONLINEAR
SOL 1
TIME 5
MEMORY 20MW
$ ------------------------------------------------------------
ASSIGN USER1=Matrix,NEW,USE=OUTPUT1,reallocate
ALTER 205
OUTPUT1 KGGNLX,,,,//-1/*USER1* $
Endalter
$ ------------------------------------------------------------
CEND
AUTOSPC( punch, SID=2, SPC, NOPRINT)=YES
POST PDA3 GLOBAL
ECHO = NONE
MAXLINES = 999999999
SPC = 2
DISPLACEMENT(NOPRINT)=ALL
SPCFORCES(NOPRINT)=ALL
STRESS(NOPRINT)=ALL
NLFORCE(NOPRINT)=ALL
NLTYPE = GEOM(STRAIN=SMALL)
SUBCASE 1
STEP 1
LOAD = 427
NLSOLVE = 1
BEGIN BULK
NLSOLVE 1 ARC SEMI AUTO            <-- SEMI updates the stiffness matrix after
                                       the last force iteration
…
```

UAI-NASTRAN Cards: Modal Solution

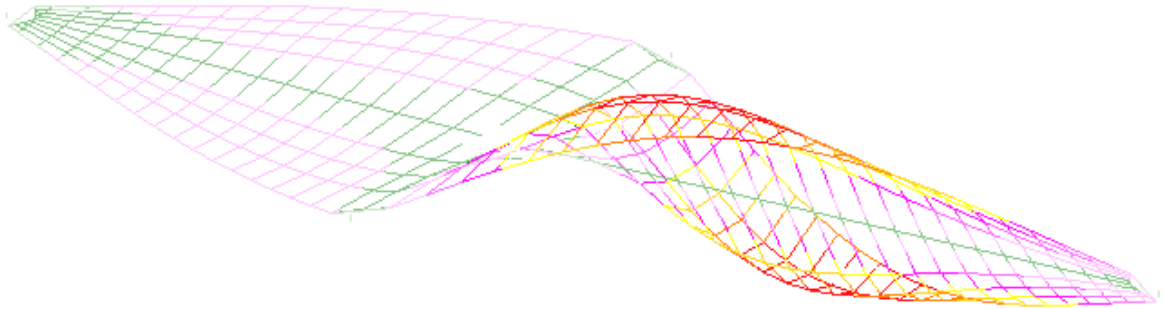
```
SOL 3
TIME 5
MEMORY 20MW
$ -------------------------------------------------------
ASSIGN USER1=Matrix,OLD,USE=INPUTT1
Alter 37
$ Load the matrix
INPUTT1 /KGGNLX,,,,/-1/*USER1* $
$ Replace the matrix KGGX by KGGNLX before any reduction
Add KGGX, KGGNLX/K1X $
Equiv K1X,KGGX/Always $
Endalter
$ -------------------------------------------------------
CEND
AUTOSPC( punch, SID=2, SPC, NOPRINT)=YES
POST PDA3 GLOBAL
TITLE = MSC/NASTRAN job created on 22-Jan-99 at 17:30:06
ECHO = NONE
MAXLINES = 999999999
SPC = 2
DISPLACEMENT(NOPRINT)=ALL
METHOD=1000
BEGIN BULK
EIGR, 1000, LANCZ,0., , ,10
…
```

Mode Shapes

Mode 1



Mode 2



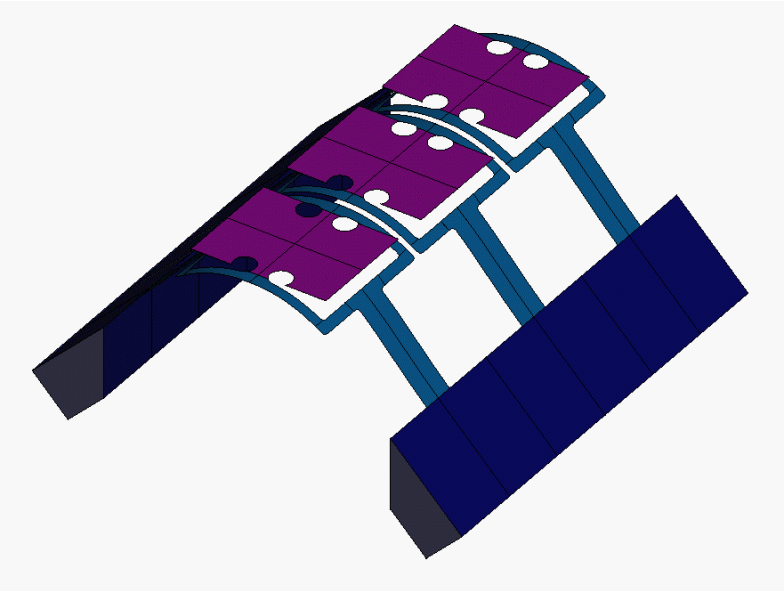
Conclusion

- 10th Scale Model
  - The membrane must be preloaded for dynamic analysis to represent the state of strain energy introduced by the constant force springs.
  - Tests in vacuum chamber have been performed and the results seam to match FEM predictions.
  - Correlation needs to be performed.

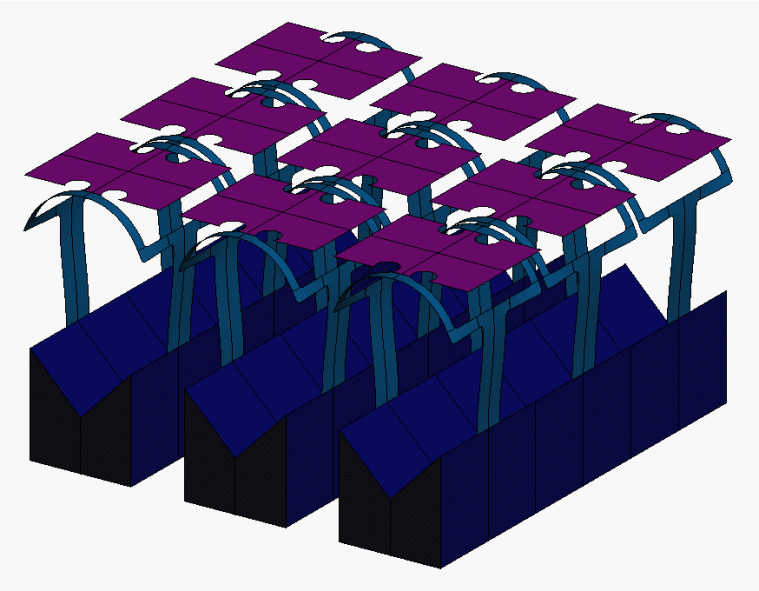
- Geometric Nonlinear Solution
  - This solution is easy to implement.
  - Updated stiffness solution can be implemented for frequency response analysis as well.

Example: Constellation-X Pop-Up Detector

- Detector array of calorimeter spectrometer for Constellation-X
- Close-packed arrays of 200x200μm pixels
- Constructed from silicon using anisotropic etch and photolithography techniques
- Pixel surfaces coated with Bismuth



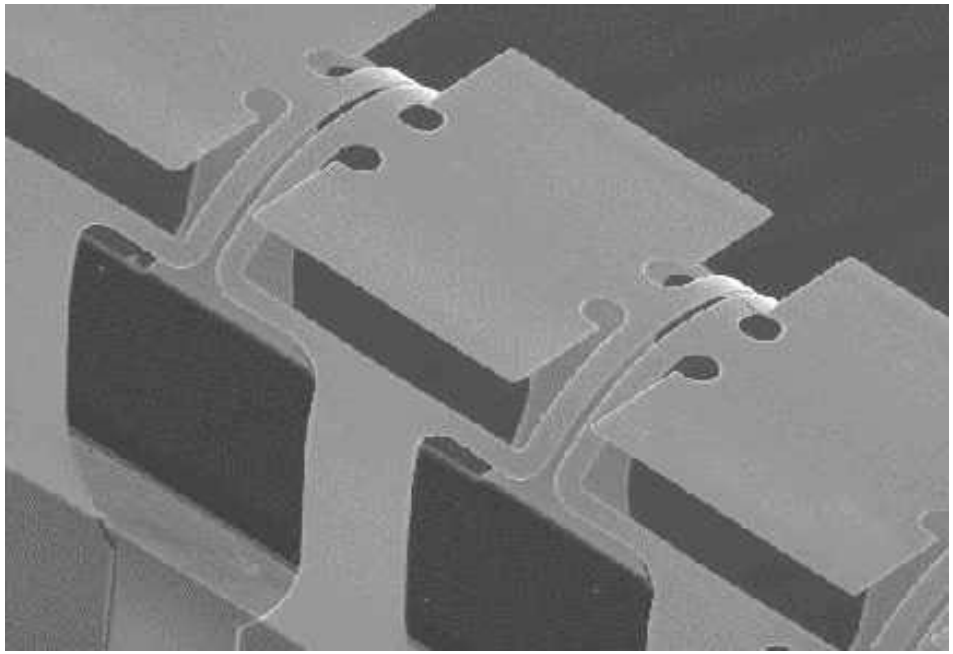
Folding Process



3x3 Array of Pop-Up Pixels

Constellation Pop-Up Detector: Mechanical Test Structures

- » Path length = 150 μm
- » Torsion bar length = 55 μm, width = 15 μm
- » Strain relief cantilever length = 75 μm



300X

Finite Element Analysis

- Nonlinear Geometry Static Analysis
  - Folding involves large displacements and rotations
  - Use UAI/NASTRAN “app nonlinear”
  - Linear material response (constructed from brittle Si)
- Preloaded Normal Modes Analysis
  - Update stiffness: import from static analysis
  - Update geometry/mass: transform nodes

Non-linear Geometry Static Analysis: Finite Element Model

- Geometry
  - Focus on a single pixel
  - QUADR elements
  - PSHELL and PCOMP physical properties
- Material
  - Single crystal silicon (1μm membrane)
    - E = 175 GPa ν = 0.27 ρ = 2300 Kg/m^3 = 2.3E-9 mg/μm^3
  - Bismuth (10μm coating)
    - E = 32 GPa ν = 0.33 ρ = 9800 Kg/m^3 = 9.8E-9 mg/μm^3
- Boundary Conditions
  - Enforced rotation
  - Enforced lateral displacement
- Units
  - Length = μm, Mass = mg ===> Force = pN, Stress = Pa

Solution Procedure and Convergence

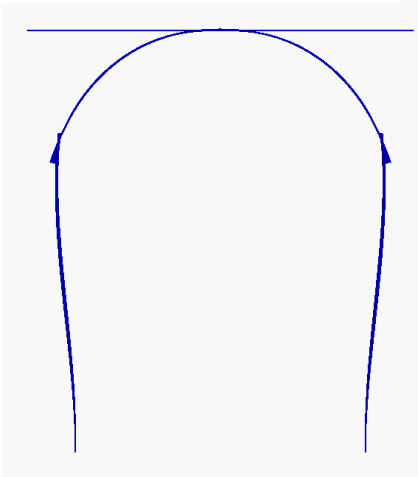
- Requires incremental and iterative solution approach
- Solution procedure:
  - Apply loads in **steps**
  - Divide steps into load **increments**
    - Automatic
    - Manual
  - Numerically **iterate** to convergent solution for each increment
    - Stiffness update (accurate, fewer iterations, more computation)
    - Unbalanced force update (approximate, more iterations, less computation)
- Primary challenge: get convergence to correct solution
- Vary steps, increments, iterations, and other parameters to obtain and improve convergence
- Non-linear significantly more sensitive than linear

UAI/NASTRAN Cards

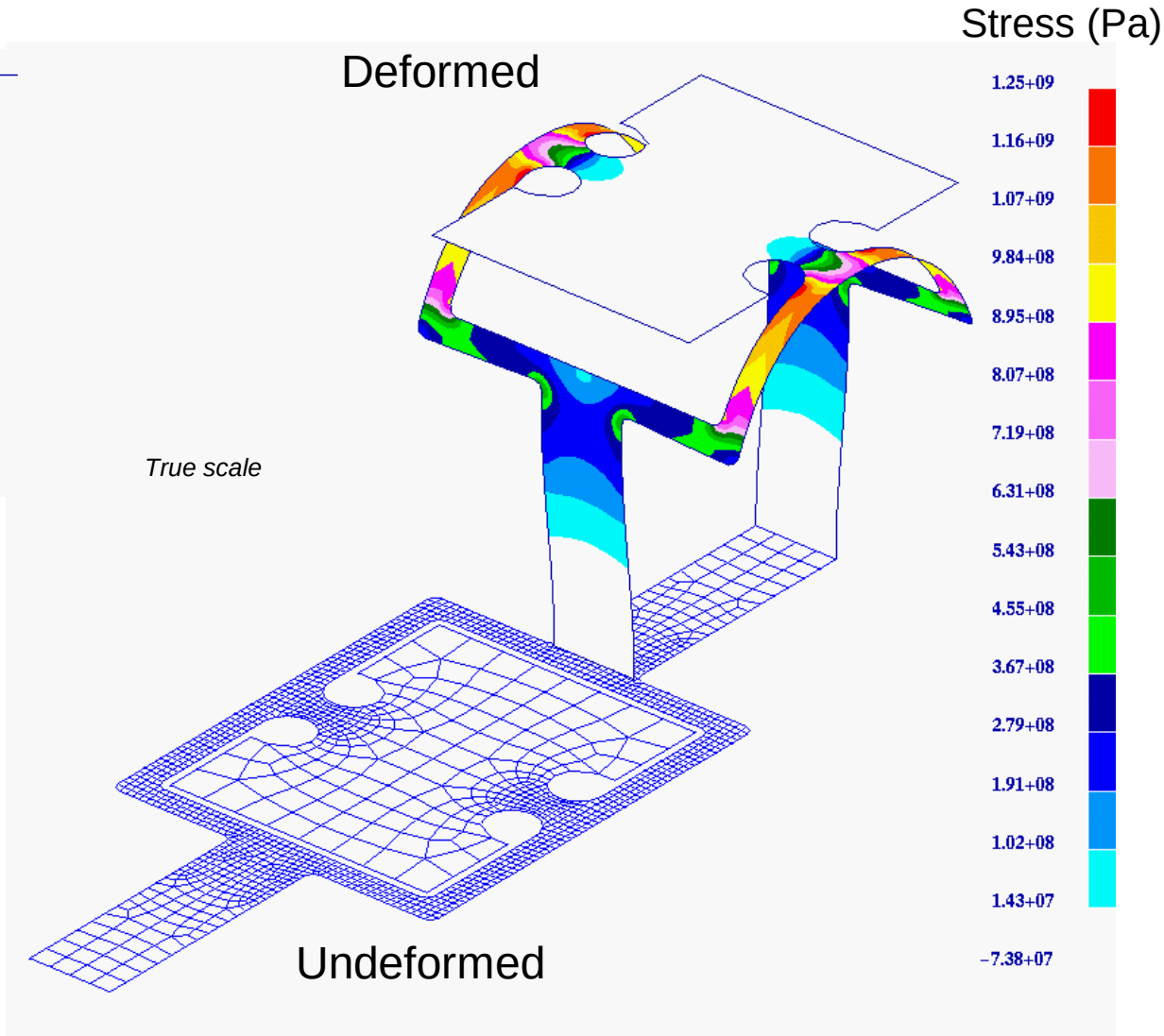
```
APP NONLINEAR
SOL STATICS
$-----------------------------------------------
$ Extract the last stiff. matrix before reduction
ASSIGN USER1=matrix.mtx,NEW,USE=OUTPUT1,reallocate
ALTER 205
OUTPUT1 KGGNLX,,,,//-1/*USER1* $
ENDALTER
$-----------------------------------------------
CEND
   ...
   DISP(PRINT) = ALL
   NLFORCE(PRINT) = ALL
   NLSTRES(PRINT,VONMISES) = ALL
   ...
SUBCASE 1
   SPC = 1
   NLTYPE = GEOM(STRAIN=GREEN)
   STEP 1
      LOAD = 301
      NLSOLVE = 1
   ...
   STEP 9
      LOAD = 309
      NLSOLVE = 1
   STEP 10
      LOAD = 310
      NLSOLVE = 2
   ...
   STEP 20
      LOAD = 320
      NLSOLVE = 2
```

```
BEGIN BULK

NLSOLVE,1,ARC,FIX,AUTO,,,,,
   200,1,100
NLSOLVE,2,ARC,FIX,FIX,,,,,
   200,1,10
   ...
MOMENT,1, 10001,,0.0,0.0,1.0,0.0
SPCD,  1,  10001,  5, 1.5708
SPC1,  1,   5, 10001
   ...
LOAD,301,1.0,   0.05,1
LOAD,302,1.0,   0.10,1
   ...
INCLUDE 'model.bdf'

ENDDATA
```

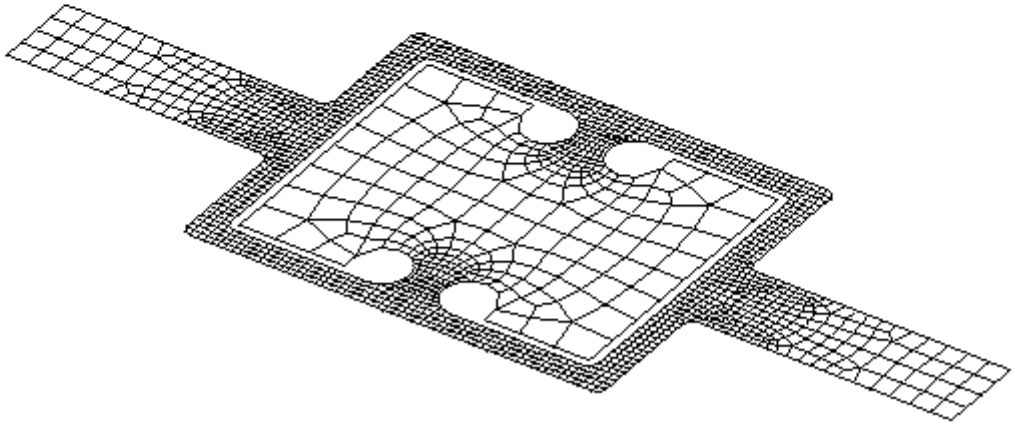
Static Analysis Results



Side View

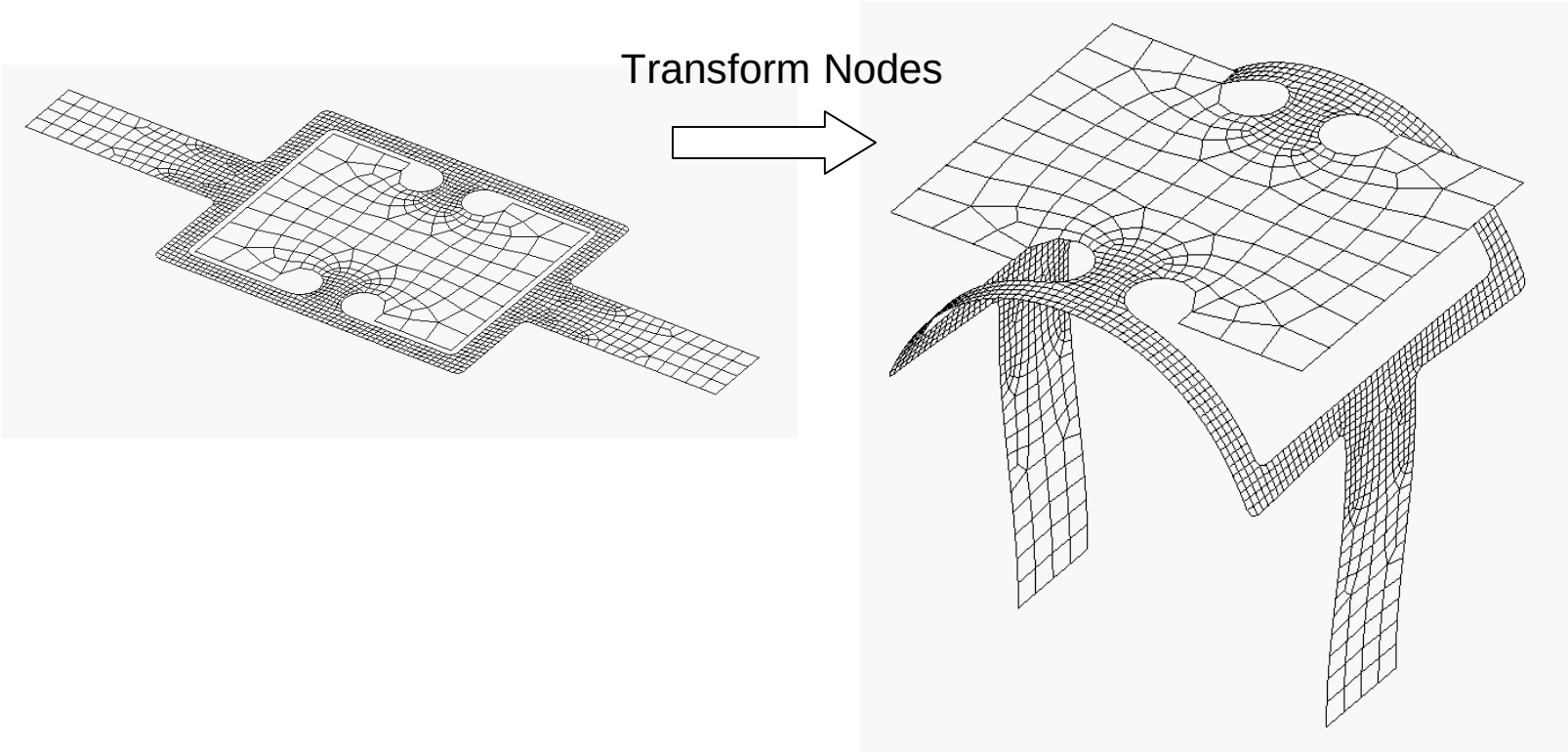


Non-linear Deformation Sequence

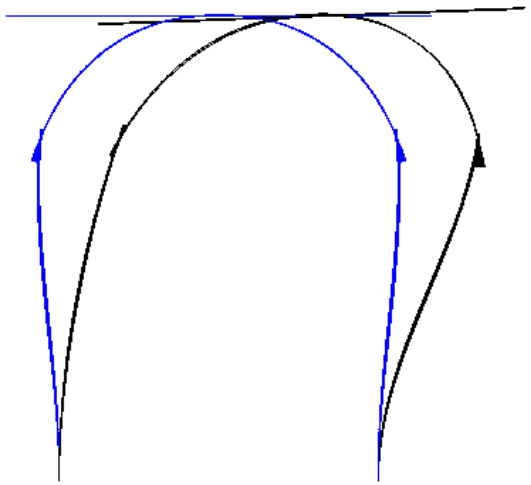


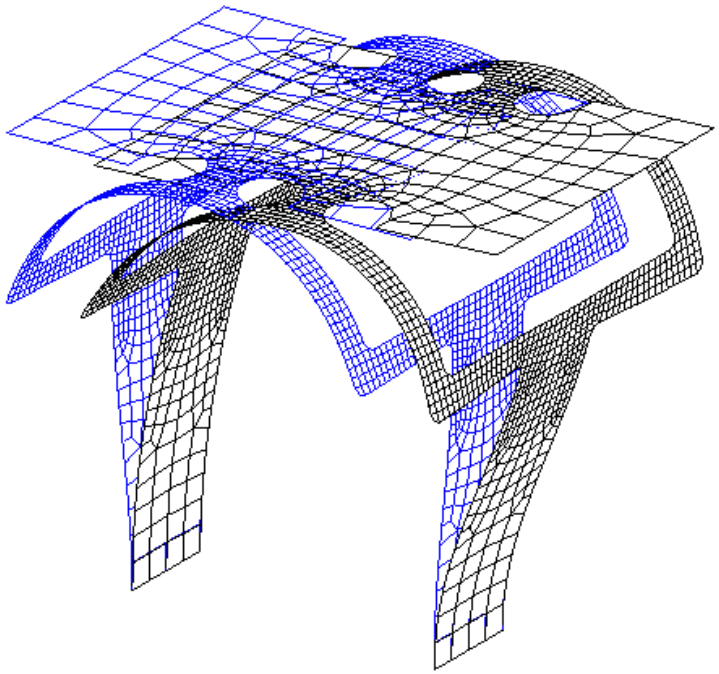
Preloaded Normal Modes Analysis

- Prior to eigenvalue solution:
  - Update stiffness: import from static analysis
  - Update geometry/mass: transform nodes



First Mode (Freq = 5400 Hz)





Modal Analysis Comparison

Transformed Nodes	Updated Stiffness	1st Mode Freq. (Hz)	1st Mode Shape
Yes		3400	Pendulum
	Yes	20800	Flat Rotate
Yes	Yes	5400	Pendulum

Conclusions

- Pros:
  - New APP NONLINEAR works well for both large displacement and instability types of non-linear geometry problems
  - Can do dynamic analysis using DMAP
  - Offers a large amount of solution control
- Cons:
  - As with all non-linear analyses, convergence can be difficult and time consuming (but not impossible!)
  - Documentation lacking and sometimes incorrect or misleading
  - First release: a few bugs and idiosyncrasies
  - Bulk temperature change not implemented for non-linear elements!