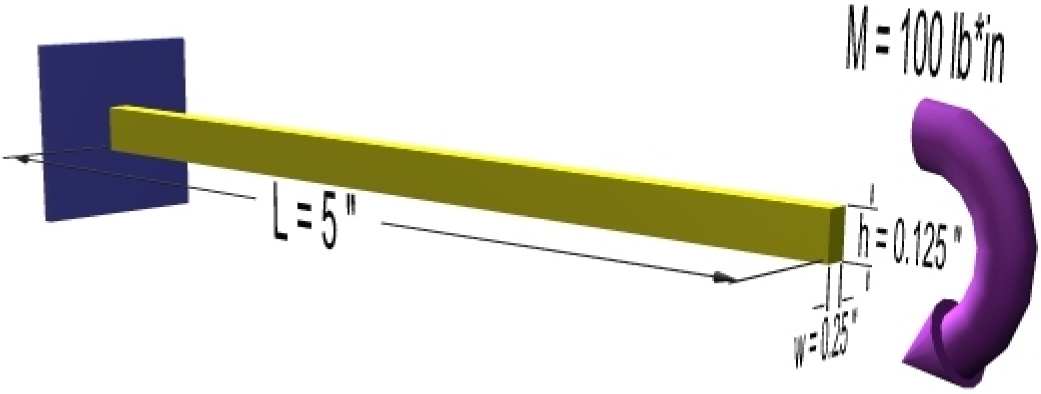
NonLinear Analysis of a Cantilever Beam

Introduction

This tutorial was created using ANSYS 7.0 The purpose of this tutorial is to outline the steps required to do a simple nonlinear analysis of the beam shown below.



There are several causes for nonlinear behaviour such as **Changing Status** (ex. contact elements), **Material Nonlinearities** and **Geometric Nonlinearities** (change in response due to large deformations). This tutorial will deal specifically with Geometric Nonlinearities .

To solve this problem, the load will added incrementally. After each increment, the stiffness matrix will be adjusted before increasing the load.

The solution will be compared to the equivalent solution using a linear response.

Preprocessing: Defining the Problem

1. **Give example a Title**
   Utility Menu > File > Change Title ...
2. **Create Keypoints**
   Preprocessor > Modeling > Create > Keypoints > In Active CS
   We are going to define 2 keypoints (the beam vertices) for this structure to create a beam with a length of 5 inches:

Keypoint	Coordinates (x,y)
1	(0,0)
2	(5,0)

3. **Define Lines**
   Preprocessor > Modeling > Create > Lines > Lines > Straight Line
   Create a line between Keypoint 1 and Keypoint 2.
4. **Define Element Types**
   Preprocessor > Element Type > Add/Edit/Delete...
   For this problem we will use the BEAM3 (Beam 2D elastic) element. This element has 3 degrees of freedom (translation along the X and Y axis's, and rotation about the Z axis). With only 3 degrees of freedom, the BEAM3 element can only be used in 2D analysis.
5. **Define Real Constants**
   Preprocessor > Real Constants... > Add...
   In the 'Real Constants for BEAM3' window, enter the following geometric properties:
   i. Cross-sectional area AREA: 0.03125
   ii. Area Moment of Inertia IZZ: 4.069e-5
   iii. Total beam height HEIGHT: 0.125
   This defines an element with a solid rectangular cross section 0.25 x 0.125 inches.
6. **Define Element Material Properties**
   Preprocessor > Material Props > Material Models > Structural > Linear > Elastic > Isotropic

In the window that appears, enter the following geometric properties for steel:

i. Young's modulus EX: 30e6
ii. Poisson's Ratio PRXY: 0.3

If you are wondering why a 'Linear' model was chosen when this is a non-linear example, it is because this example is for non-linear geometry, not non-linear material properties. If we were considering a block of wood, for example, we would have to consider non-linear material properties.

7. **Define Mesh Size**

   Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > All Lines...

   For this example we will specify an element edge length of 0.1 " (50 element divisions along the line).

8. **Mesh the frame**

   Preprocessor > Meshing > Mesh > Lines > click 'Pick All'

   ```
   LMESH,ALL
   ```

Solution: Assigning Loads and Solving

1. **Define Analysis Type**

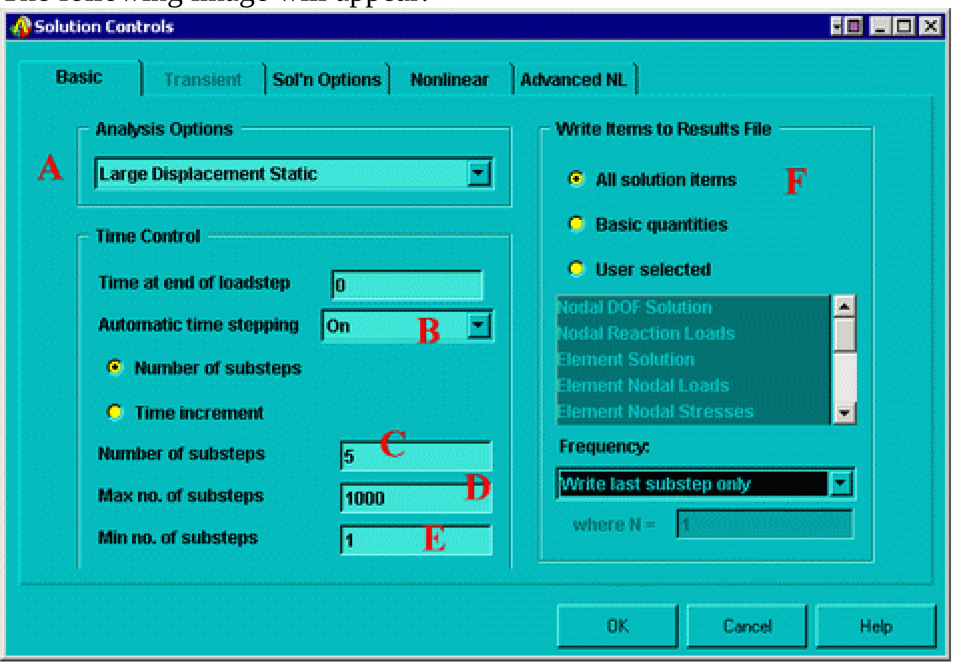
   Solution > New Analysis > Static

   ```
   ANTYPE,0
   ```

2. **Set Solution Controls**

   o Select **Solution > Analysis Type > Sol'n Control...**

   The following image will appear:



   Ensure the following selections are made (as shown above)

   A. Ensure Large Static Displacements are permitted (this will include the effects of large deflection in the results)

   B. Ensure Automatic time stepping is on. Automatic time stepping allows ANSYS to determine appropriate sizes to break the load steps into. Decreasing the step size usually ensures better accuracy, however, this takes time. The Automatic Time Step feature will determine an appropriate balance. This feature also activates the ANSYS bisection feature which will allow recovery if convergence fails.

   C. Enter 5 as the number of substeps. This will set the initial substep to 1/5 th of the total load.

      The following example explains this: Assume that the applied load is 100 lb*in. If the Automatic Time Stepping was off, there would be 5 load steps (each increasing by 1/5 th of the total load):

      ▪ 20 lb*in

- 40 lb*in
- 60 lb*in
- 80 lb*in
- 100 lb*in

Now, with the Automatic Time Stepping is on, the first step size will still be 20 lb*in. However, the remaining substeps will be determined based on the response of the material due to the previous load increment.

D. Enter a maximum number of substeps of 1000. This stops the program if the solution does not converge after 1000 steps.
E. Enter a minimum number of substeps of 1.
F. Ensure all solution items are writen to a results file.

NOTE

There are several options which have not been changed from their default values. For more information about these commands, type `help` followed by the command into the command line.

Function	Command	Comments
Load Step	**KBC**	Loads are either linearly interpolated (ramped) from the one substep to another (ie - the load will increase from 10 lbs to 20 lbs in a linear fashion) or they are step functions (ie. the load steps directly from 10 lbs to 20 lbs). By default, the load is ramped. You may wish to use the stepped loading for rate-dependent behaviour or transient load steps.
Output	**OUTRES**	This command controls the solution data written to the database. By default, all of the solution items are written at the end of each load step. You may select only a specific iten (ie Nodal DOF solution) to decrease processing time.
Stress Stiffness	**SSTIF**	This command activates stress stiffness effects in nonlinear analyses. When large static deformations are permitted (as they are in this case), stress stiffening is automatically included. For some special nonlinear cases, this can cause divergence because some elements do not provide a complete consistent tangent.
Newton Raphson	**NROPT**	By default, the program will automatically choose the Newton-Raphson options. Options include the full Newton-Raphson, the modified Newton-Raphson, the previously computed matrix, and the full Newton-Raphson with unsymmetric matrices of elements.
Convergence Values	**CNVTOL**	By default, the program checks the out-of-balance load for any active DOF.

3. **Apply Constraints**

   Solution > Define Loads > Apply > Structural > Displacement > On Keypoints

   Fix Keypoint 1 (ie all DOFs constrained).

4. **Apply Loads**

   Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

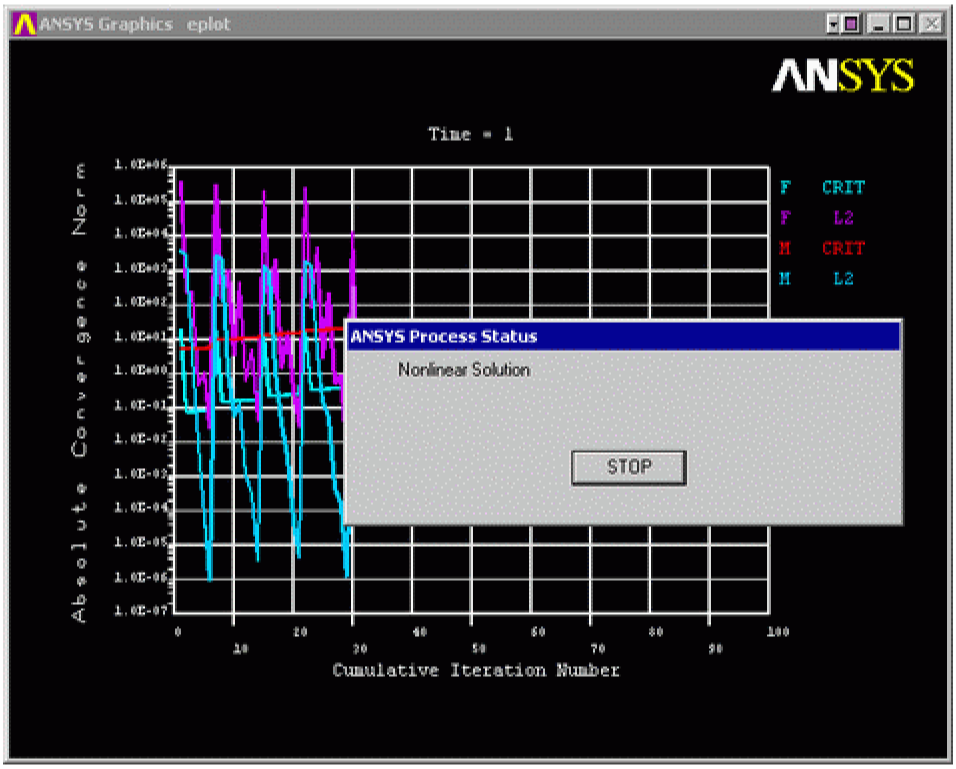
   Place a -100 lb*in moment in the MZ direction at the right end of the beam (Keypoint 2)

5. **Solve the System**

   Solution > Solve > Current LS

   `SOLVE`

The following will appear on your screan for NonLinear Analyses

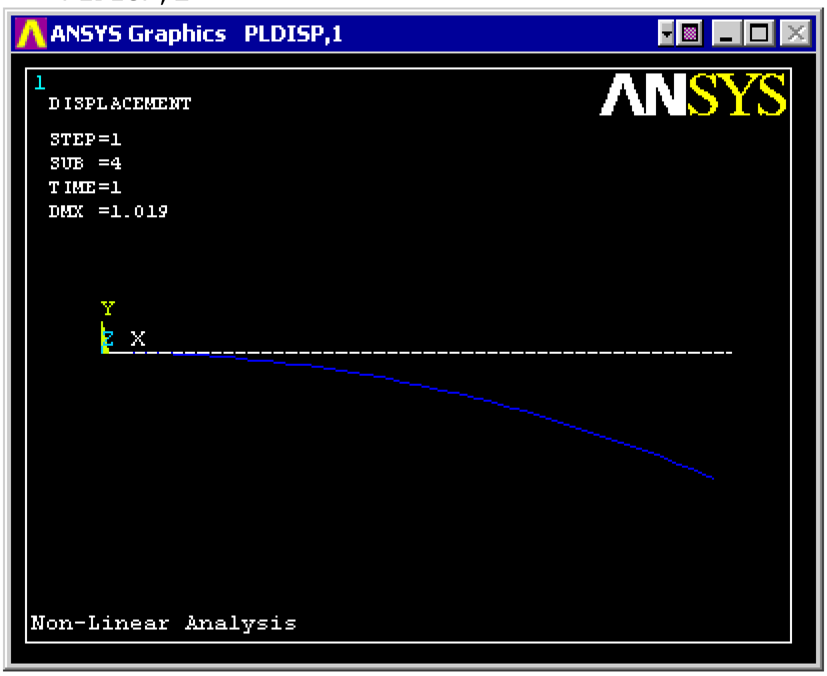


This shows the convergence of the solution.

General Postprocessing: Viewing the Results

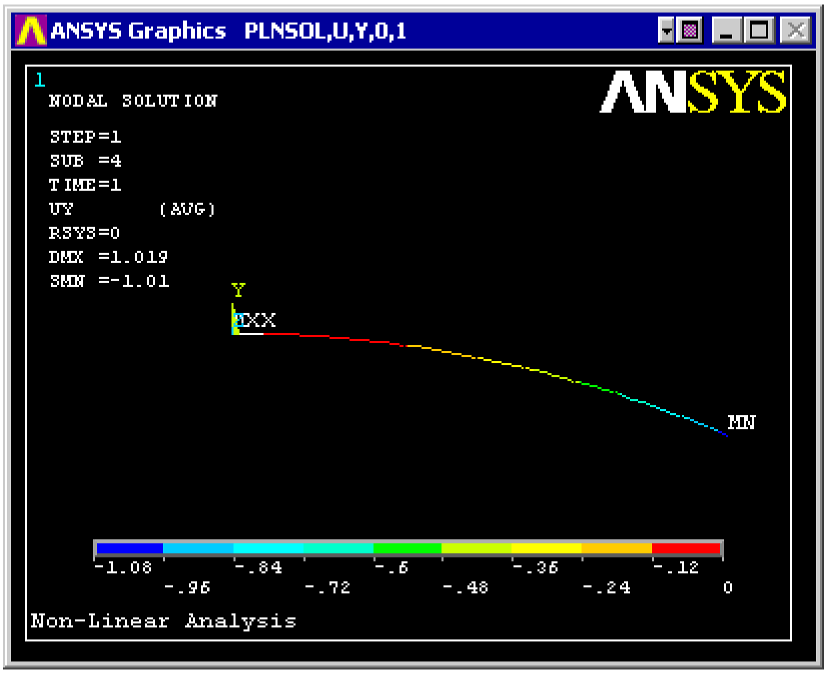
1. **View the deformed shape**

   General Postproc > Plot Results > Deformed Shape... > Def + undeformed
   `PLDISP,1`



2. **View the deflection contour plot**

   General Postproc > Plot Results > Contour Plot > Nodal Solu... > DOF solution, UY
   `PLNSOL,U,Y,0,1`



3. **List Horizontal Displacement**

   If this example is performed as a linear model there will be no nodal deflection in the horizontal direction due to the small deflections assumptions. However, this is not realistic for large deflections. Modeling the system non-linearly, these horizontal deflections are calculated by ANSYS.

   General Postproc > **List Results** > **Nodal Solution...**> **DOF solution, UX**

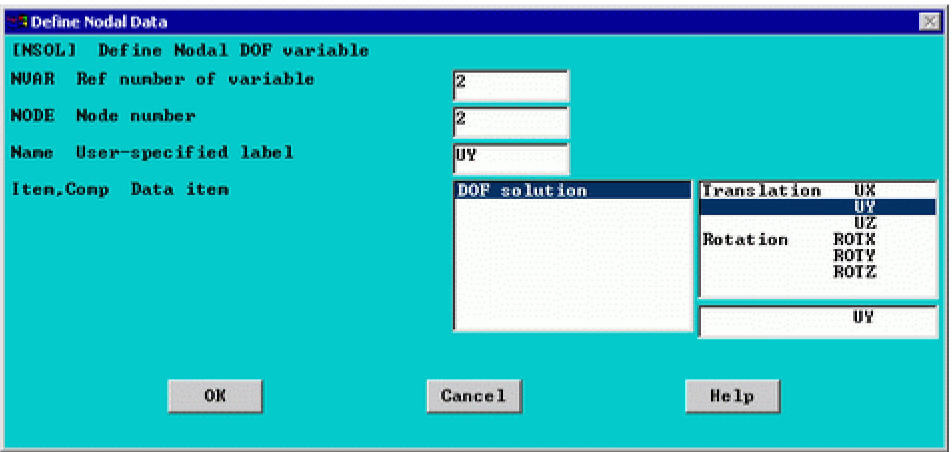
Other results can be obtained as shown in previous linear static analyses.

Time History Postprocessing: Viewing the Results

As shown, you can obtain the results (such as deflection, stress and bending moment diagrams) the same way you did in previous examples using the General Postprocessor. However, you may wish to view time history results such as the deflection of the object and the step sizes of the load.

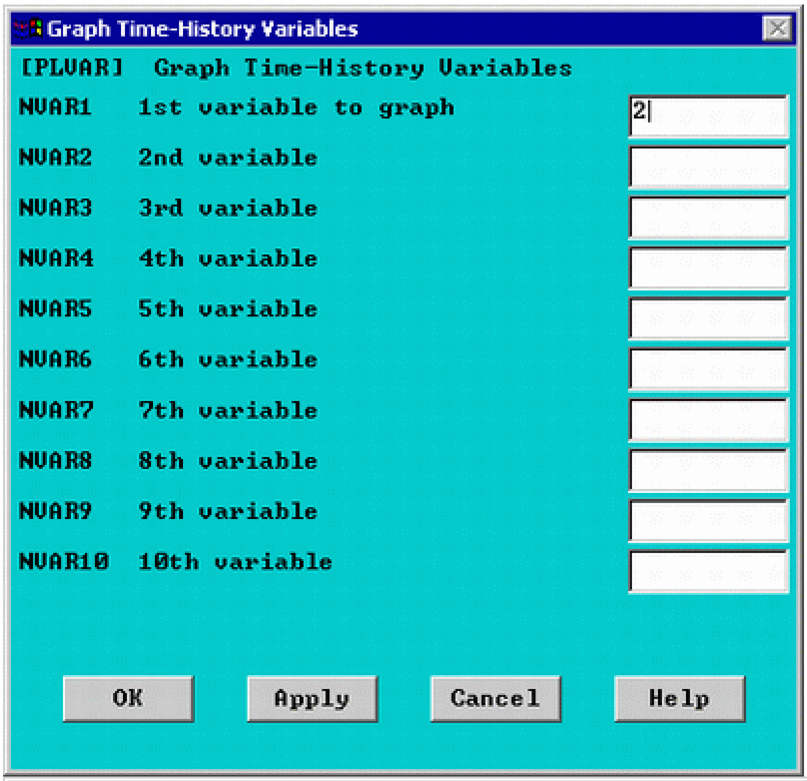
As you recall, the load was applied in steps. The step size was automatically determined in ANSYS

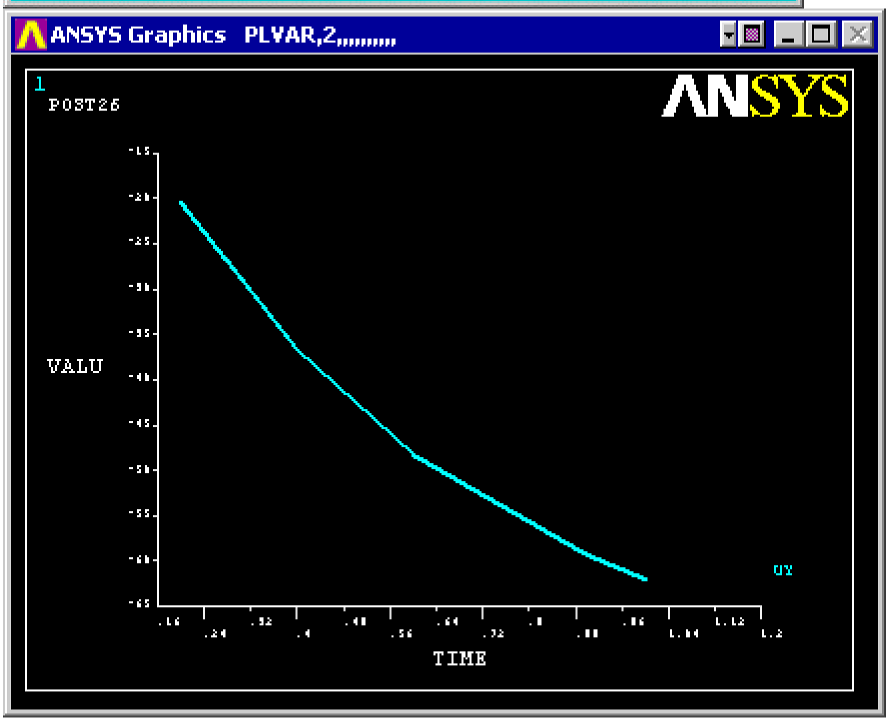
1. **Define Variables**
   - Select: **TimeHist Postpro** > **Define Variables** > **Add...** > **Nodal DOF results**
   - Select Keypoint 2 (Node 2) when prompted
   - Complete the following window as shown to define the translational displacement in the y direction.



Translational displacement of node 2 is now stored as variable 2 (variable 1 being time)

2. **Graph Results over time**
   - Select **TimeHist Postpro** > **Graph Variables...**
   - Enter 2 (UY) as the 1st variable to graph (shown below)





Command File Mode of Solution

The above example was solved using a mixture of the Graphical User Interface (or GUI) and the command language interface of ANSYS. This problem has also been solved using the ANSYS command language interface that you may want to browse. Copy and paste following code into Notepad or a similar text editor and save it to your computer. Now go to **'File > Read input from...'** and select the file.

```
/prep7                      ! start preprocessor
/title,NonLinear Analysis of Cantilever Beam

k,1,0,0,0                   ! define keypoints
k,2,5,0,0                   ! 5" beam (length)
```

```
l,1,2                     ! define line

et,1,beam3                     ! Beam
r,1,0.03125,4.069e-5,0.125     ! area, izz, height of beam
mp,ex,1,30.0e6                 ! Young's Modulus
mp,prxy,1,0.3                  ! Poisson's ratio

esize,0.1                 ! element size of 0.1"
lmesh,all                 ! mesh the line

finish                    ! stop preprocessor
/solu                     ! start solution phase

antype,static             ! static analysis
nlgeom,on                 ! turn on non-linear geometry analysis

autots,on                 ! auto time stepping
nsubst,5,1000,1          ! Size of first substep=1/5 of the total load, max #
substeps=1000, min # substeps=1
outres,all,all            ! save results of all iterations

dk,1,all                  ! constrain all DOF on ground

fk,2,mz,-100              ! applied moment

solve

/post1
pldisp,1                  ! display deformed mesh
PRNSOL,U,X               ! lists horizontal deflections
```

Reference

http://www.mece.ualberta.ca/tutorials/ansys