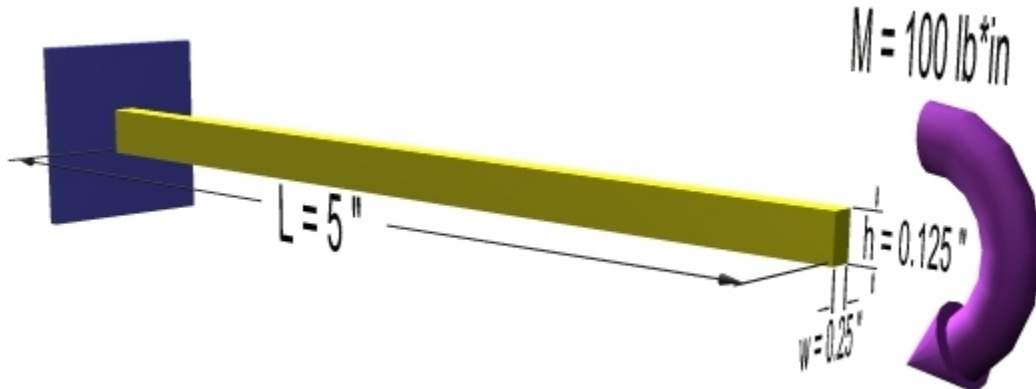


NonLinear Analysis of a Cantilever Beam

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**, **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 be 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.

ANSYS Command Listing

```

/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=10
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
```