Elasto-plastic bending of 3D beam.
Residual deformations and stresses

Structural nonlinearities cause the response of a structure or component to vary disproportionately with the applied forces. Realistically, all structures are nonlinear in nature but not always to a degree that the nonlinearities have a significant effect on an analysis. However, if the engineer determines that nonlinearities affect the behavior of a structure to the extent that they cannot be ignored, a nonlinear analysis is required.

In a nonlinear analysis, the structure’s stiffness matrix and load vector may depend on the solution and are, therefore, unknown. To solve the problem, the ANSYS program uses an iterative procedure based on the Newton-Raphson method, in which a series of linear approximations (iterations) converges to the actual nonlinear solution. In many nonlinear static analyses, the loading must be applied in the load steps, representing the history of the load. Each load step is ramped starting from the initial load up to the final load value of interest and can be divided into the substeps.

In this approach the nonlinear static analysis problem is analysed by dividing the load into the load steps and the substeps, performing at each of the substeps a succession of linear approximations to obtain equilibrium. Each linear approximation requires one pass through the equation solver (known as an equilibrium iteration).

The nonlinearities in mechanics of structures may be classified into three categories: material, geometric, and element nonlinearities

Material Nonlinearities
A material nonlinearity exists when stress is not proportional to strain. The ANSYS program simulates various types of nonlinear material behavior. Plasticity, multilinear elasticity, and hyperelasticity are characterized by a nonlinear stress-strain relationship. Viscoplasticity, creep, and viscoelasticity are behaviors in which strain may depend on other factors such as time, temperature, or stress.

To fully account for plastic material behavior in an analysis, three important concepts must be considered: the yield criterion, the flow rule, and the hardening law. The yield criterion measures the 3D stress state by computing a single-valued equivalent stress that is compared against the yield strength to determine when the material will yield. The flow rule predicts the direction in which strain will occur. The hardening law, which is applicable to materials that strain harden, describes how the yield surface expands or changes as the material strains.

Classical Bilinear Kinematic Hardening describes general metallic materials that are considered to be bilinear; having one elastic and one plastic slope. This option is applicable to most common, initially isotropic, engineering metals in the small strain region. Von Mises yield criterion is used with an associative flow rule. Kinematic hardening accounts for the Bauschinger effect.
EXAMPLE
Residual deformations and stresses in elasto-plastic beam.

The simple supported beam with the rectangular crosssection \( b=10\text{mm}, \, h=20\text{mm} \) has the length \( l=200\text{mm} \). The beam made of steel (\( E=2\times 10^5 \text{MPa}, \, \nu=0.3 \), yield stress \( S_y=250\text{MPa} \), hardening tangent modulus \( E_t=100\text{MPa} \)) is loaded by the uniform pressure \( q=5\text{MPa} \) and then unloaded \( (q=0 \text{MPa}) \). Find the distribution of \( \sigma_z \) stresses and the deflection \( u_y \) after the loading and unloading.
1/2 of the analysed beam

\[
\begin{align*}
\sigma & = S_y + \varepsilon \cdot E_y \\
\varepsilon & = \frac{U}{A} \\
\tan \beta & = \frac{S_y}{E_y} \\
\tan \alpha & = \frac{S_y}{E_y}
\end{align*}
\]

Elasto-plastic model of material (BKin)
Summary of steps of numerical analysis (3D):

**Preprocessor**

a) Build the geometrical model of the beam (*modeling > create > volume > block*)
b) Define the mechanical properties of the material (linear and nonlinear- bilinear kinematic hardening model)

c) Mesh the beam using mapped mesh with 20-node brick elements
   Apply the displacement boundary conditions.
SOLUTION

a) Define the analysis type (static)
b) Define the analysis options (large deformations)

c) Apply the load (pressure 5MPa at upper area)

d) Define the LOAD STEP OPTIONS corresponding to the first loading phase: time=5, number of substeps 10, ramped b.c., automatic time stepping off, Require the results for every substep
e) Write the loads as the load step 1 of the analysis (Write LS File)

f) Delete the load (pressure at the upper area)

g) Define the LOAD STEP OPTIONS corresponding to the unloading: time=10, number of substeps 1, ramped b.c., automatic time stepping off,

h) Write the loads as the load step 2 of the analysis (Write LS File)

i) Solve the problem (LSSOLVE)

**GENERAL POSTPROCESSOR**

Check the results.
Find the interesting results and present them in the required form.
Results for full loading: a) $U_y$ displacement, b) $\sigma_z$ stress, c) equivalent plastic strain
Results for unloading: a) $U_y$ displacement, b) $\sigma_z$ stress

The distribution of $\sigma_z$ stress along the vertical path in the centre of the beam corresponding to $q=5$ MPa, and $q=0$ MPa.
The Time History Postprocessor may be also used to plot some time history results

![Image of Time History Postprocessor]

*The deflection of the beam during the loading and unloading*

**Tasks to be done:**

**Task 1** Solve the 3D model as shown in this instruction  
**Task 2** Solve the problem using 2D Plane stress modelling

**The report should contain:**

The description of the problem and the FE model, the most interesting results corresponding to the maximum load and to the final residual state. Close the report by the discussion of the results with the comparison to simple analytical estimations.