Ansys Lab — Material nonlinearity

Purpose

- To learn how to use ANSYS to solve elastic-plastic structural problem.

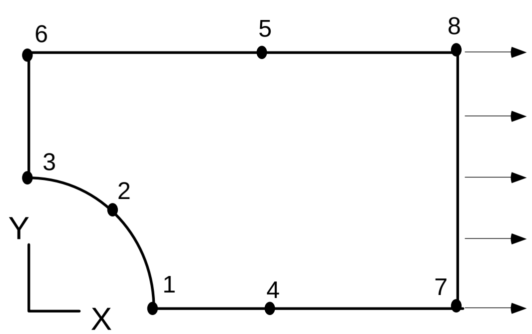
Background

When ductile metals are loaded beyond elastic range, the initial linear stress response will give way to a complicated nonlinear response, characterized by a much-reduced modulus and different stress behavior along load and unloading path. An idealized stress-strain curve in uniaxial tension is depicted in the figure below. The turning point is called elastic limit, the corresponding stress σ_y is called the yield stress. In 3D case, the elastic limit is characterized by some measures of stress, for example, if the von Mises stress reaches a critical value.

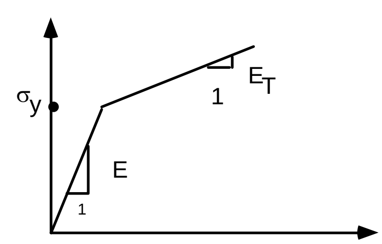
Finite element elastic-plastic analysis is much more delicate than simple elastic analysis. For one, the FEM equation is no longer linear, but a set of nonlinear equations that needs to be solved iteratively. Typically, we divide the applied load into small increments so as to have a better numerical performance.

Lab exercise

A 36x20 rectangular plate with a circular hole in its center is subjected to a tensile pressure force along two end sides into the plastic range. Bilinear isotropic hardening is used. Small strain plane stress condition is assumed. Note that only 1/4th domain is needed for analysis because of symmetry.



$E = 7E4$ $\nu = 0.3$



Bilinear isotropic hardening material
$\sigma_y = 243$ $E_T = 2E2$

Preprocessing

1.1. Define estimated reference pressure at yield.
We define a parameter “Pres” to be assigned to the pressure applied on the edge.

• ANSYS Utility Menu > Parameters > Scalar Parameters
• Scalar Parameters
Selection [**Pres=-100**] > Accept
• Parameters > Save Parameters > OK
• Scalar Parameters > Close
• ANSYS Toolbar > SAVE_DB

1. 2. Begin the model creation.

Create a solid model of a square with a quarter of a circular hole. Use any of your favorite method. The easiest way is perhaps through Boolean operations.

1.3. Define the element type (PLANE182) and material properties.

Plane182 is a 4 node quad element that deals with both small and large strains, with a variety of material options including elasto-plasticity. In ANSYS 18x elements, material options are input separately. In this problem, elasticity is defined by Young’s modulus and Poisson’s ratio, and plasticity is defined by yield stress and the elastic-plastic tangent modulus.

• ANSYS Main Menu
• Preprocessor > Element Type > Add/Edit/ Delete >Add
Solid > [**Quad 4node 182**]
• Preprocessor > Material Props > Material Model > Structural > Linear >Elastic > Isotropic
> EX [**7E4**]
> PRXY [**0.3**]
! define the plastic parameters
• Preprocessor > Material Props > Material Model > Structural > Nonlinear > Inelastic > Rate Independent > Isotropic Hardening Plasticity > Mises Plasticity > Bilinear
Yield Stss [**243**] Tang Mod [**2E2**]
>Material>Exit
• ANSYS Toolbar > SAVE_DB

1. 4. Mesh the solid model.

• ANSYS Utility Menu > Plot > Areas
• ANSYS Main Menu
• Preprocessor > -Meshing- **Size Control**
-ManualSize-... > -Global- **Size** > NDIV [**20**] ! define the mesh density by # of divisions
• Preprocessor > -Meshing- **Mesh**
-Areas- **Free** > Pick All
• ANSYS Toolbar > SAVE_DB

Solution

1. Specify load steps and apply force and boundary conditions.

Perform two load steps analysis in the solution processor, the first step is a null solution. The purpose of this null solution is just to let the graphic display starts from the zero. And second step used 1.5 * Pres as the applied pressure loading. If the second load step is not large enough to let the structure go to the plastic range, an even larger load application can be used here. The second load steps is divided into 10 substeps, which means that in each substep, an increment of 1/10 of the total load is applied.

• ANSYS Utility Menu
• PlotCtrls > Symbols > [/PBC] [ALL Applied B.C.'s] > [/PSF] [Pressures] Show pres and
  Convects as [Arrows] > OK
• ANSYS Main Menu
  • Solution > Analysis Type> Sol'n Control > Analysis Options [**Small Displacement Static**]
  > Time at the end of loadstep [1E-7] > Number of substeps [**1**] >
  > Write Items to Result File > All Solution Items
  > Frequency [Write every N eubstep] where N=[**1**]> OK
  • Solution > -Loads- Apply
  -Structural- Displacements > -Symmetry B.C.- On Lines
  > [Left vertical line] > Apply
  > [Both Bottom lines] > OK
  • Solution > -Define Loads- Apply
  -Structural- Pressure > On Lines
  > [The right vertical line where the forces applied] > OK > VALI [0] > OK

  • Solution > Load Step Options> Write LS File > [**1**] > OK

  ! Load Step 2. The essential boundary conditions carry over to the next step unless explicitly modified here.

  • Solution > -Define Loads- Apply
  -Structural- Pressure > On Lines
  [The right vertical line where the forces applied] > OK > VALI [**1.5*Pres**] > OK
  > Sol'n Control > Time at the end of load step [**1.5**] > Number of Substeps [**10**]
  > Write Items to Result File … **>** Frequency [Write Every N Substep] where N =[**2**]> OK
  • Solution > Write LS File > **[2**] > OK

  • ANSYS Main Menu > Solution
  -Solve- From LS Files > LSMIN [1] LSMAX [2] > OK

  It might take a few minutes to complete the solution.

Postprocessing

1. Review the results using the time history postprocessor (POST26).

We want to plot the time history of the x-displacement at a chosen point. In order to plot the load-displacement plot, we need to specify two sets of variables. The first set is time (ANSYS default **NVAR 1.** In this problem, the "time" is not the physical time but the load factor). The second is the displacement history UX at a point, say the left-lower corner of the plate. In the following commands, we assign **NVAR 2** for this displacement set.

• ANSYS main Menu
• TimeHist Postproc
> Settings > Graph... > [XVAR] [**Single variable**] Single Variable no. [**2**] > OK
> Define Variables... > Add > Nodal DOF > [**Pick the left lower node**] > OK
> NVAR [**2**]
Name User-Specified label [**UX**]
Item, Comp [DOF solution] [Translation UX] > OK > Close
> List Variables... > NVAR1 [**2**] > OK ! to check if this variable is defined.
• ANSYS Utility Menu
• PlotCtrls > Style > Graph > Modify Axes > [/AXLAB] X-axis label [**Displacement**]
[/AXLAB] Y-axis label [**Load Factor**] > OK
• TimeHist Postproc >
Graph Variables... > NVAR1 [**1**] > OK

You can get the displacement-load factor curve here, which is part of your report.

2. Review the results using the general postprocessor (POST1).

View equivalent plastic strain from the load step 2

• ANSYS Main Menu > General Postproc
> -Read Results- **Last Set** > Plot Results
-Contour Plot- **Nodal Solution**
-Contour Nodal Solution Data > [**Strain-plastic**] > [**Eqv plastic EPEQ**] > OK

Note: the equivalent plastic strain is a measure of the intensity of the plastic strain.

LAB Report

Include the following plots for the result part of the lab report:
(1). The contour of the equivalent plastic strain plotted on the deformed mesh;
(2). The contour von Mises stress. By inspection, check that the values in the plastic zone are close to the yield stress σ_y. The von Mises stress (output SEQV) is defined by

$$\sigma_e = \{\frac{1}{2}[(\sigma 1 - \sigma 2)^2 + (\sigma 2 - \sigma 3)^2 + (\sigma 3 - \sigma 1)^2]\}^{\frac{1}{2}}$$

(3). The displacement-force curve.