Exercise 5: Plasticity

A. Cold Forming

For this task, we will use a cantilever beam geometry, very similar to the previous models you built (cf. Figure 1). Use the provided data in Table 1 to create the beam using the **Design Modeler**.

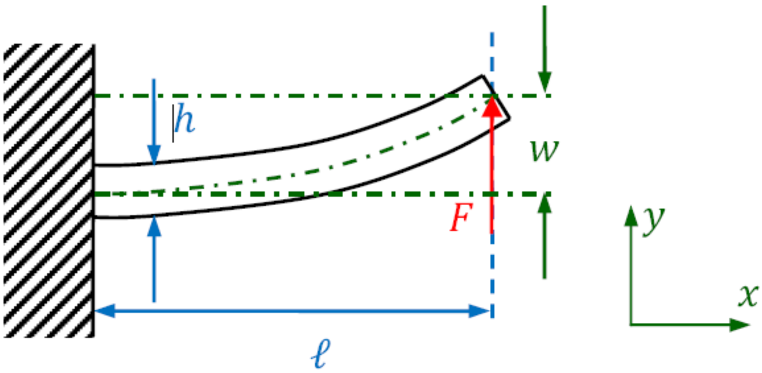

Figure 1: Cantilever beam with applied load at the right end

l	1000 mm	Beam length
h	60 mm	Beam height
t	40 mm	Beam thickness
F_1	16 000 N	Force for load step 1
F_2	−21 500 N	Force for load step 2
F_3	0 N	Force for load step 3
E	73 100 MPa	Young's modulus of aluminum
ν	0,33	Poisson's ratio
E_t	7310 MPa	Tangent modulus
σ_{yield}	414 MPa	Yield strength

Table 1: Geometric and material data

Now add a new material to your **Engineering Data**. Use the material properties provided by Table 1 to define both the material's **Isotropic Elasticity** as well as **Bilinear Isotropic Hardening** (drag and drop from the **Toolbox**). Afterwards, selecting **Bilinear Isotropic Hardening** in the properties windows should bring up a stress-strain curve resembling Figure 2.

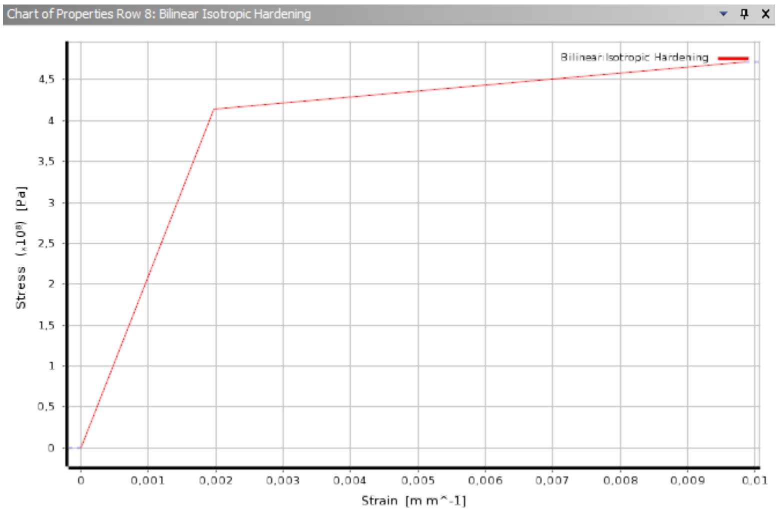

FIGURE 2: STRESS-STRAIN CURVE FOR OUR ELASTO-PLASTIC MATERIAL

In the **Mechanical** module create a hexahedral mesh with an element size (edge length) of 10 mm. To be able to define the three load steps of Table 1, set **Analysis Settings → Step Controls → Number of Steps** to 3. You can then switch the **Y Component** of the force load to **Tabular** to define the three different force magnitudes (Table 1). Done correctly, the load history should look like Figure 4.

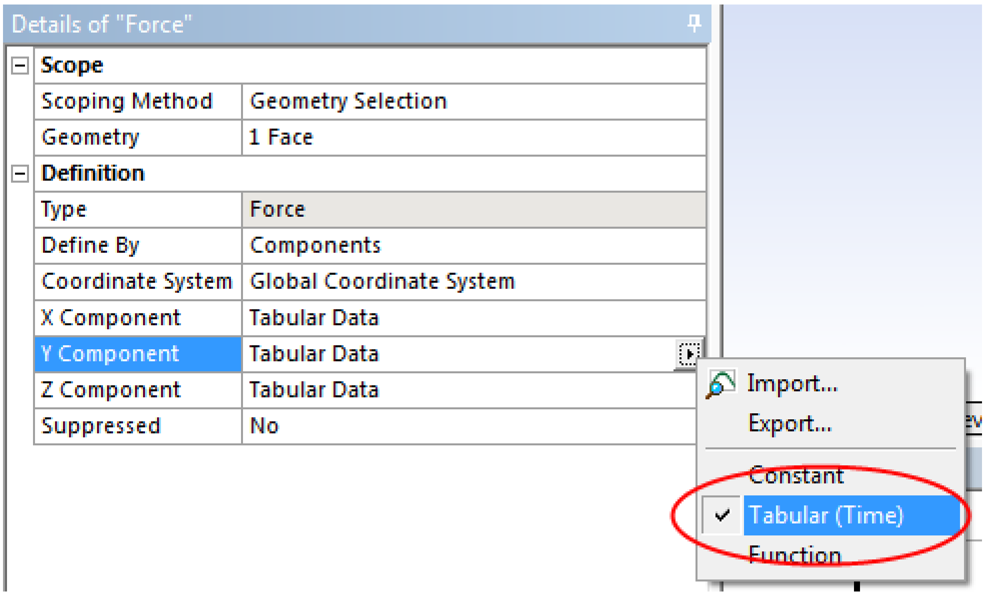

FIGURE 3: USING TABULAR DATA FOR THE FORCE COMPONENTS

Also set **Analysis Settings → Solver Controls → Large Deflection** to **On**. Switch **Solution (A6) → Solution Information → Solution Output** to **Force Convergence**. This setting ensures that you can follow the solution progress while ANSYS tries to solve the system of non-linear equations; simply select **Solution Information** during or after the solution phase to view the convergence graph.

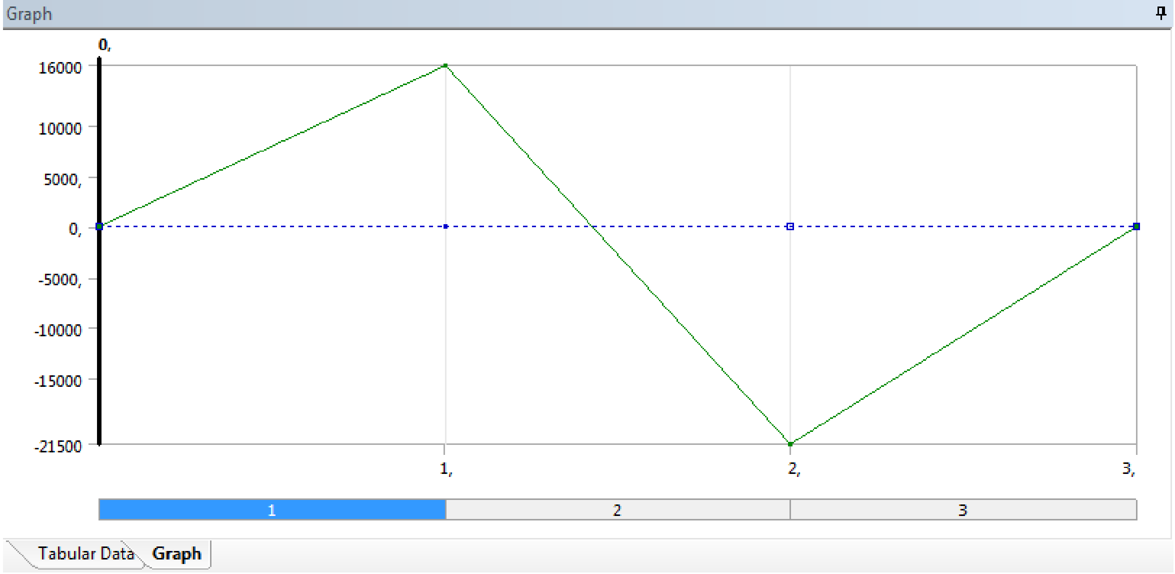

FIGURE 4: LOAD HISTORY WITH THREE DISTINCT STEPS

After solving, plot the normal elastic strain in *x* direction (cf. Figure 5).

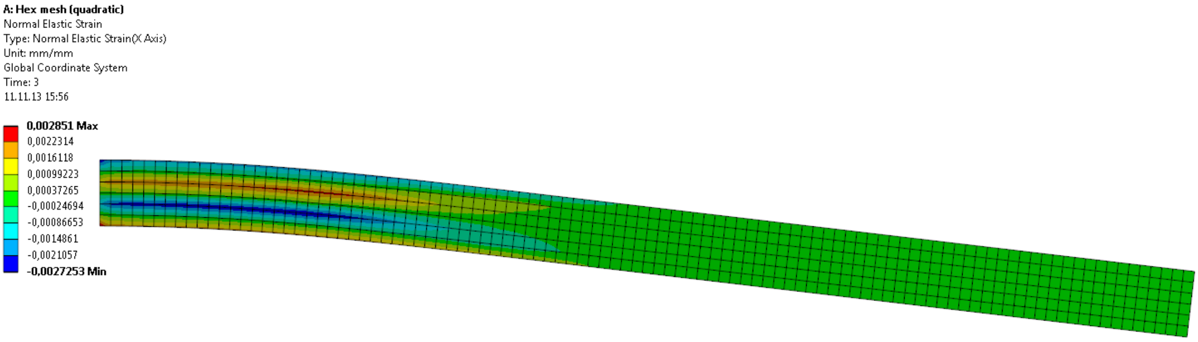

FIGURE 5: NORMAL ELASTIC STRAIN IN *X* DIRECTION AFTER THE LAST LOAD STEP

Questions

- Why have the solution times increased considerably compared to the previous beam models?
- Why are there still elastic strains remaining after setting the force to 0 N? Explain the pattern of elastic strains!
- How would the pattern change if we applied only the first and the last load step?