Exercise 3: Advanced Material Models (Plasticity and Anisotropy)

Goals

Today you will learn how to use advanced material models beyond linear isotropic elasticity. We will first have a look at plasticity with strain hardening. Then we will switch to wood as an example for anisotropic material behavior.

1. Cold Forming

For this first 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 **DesignModeler**.
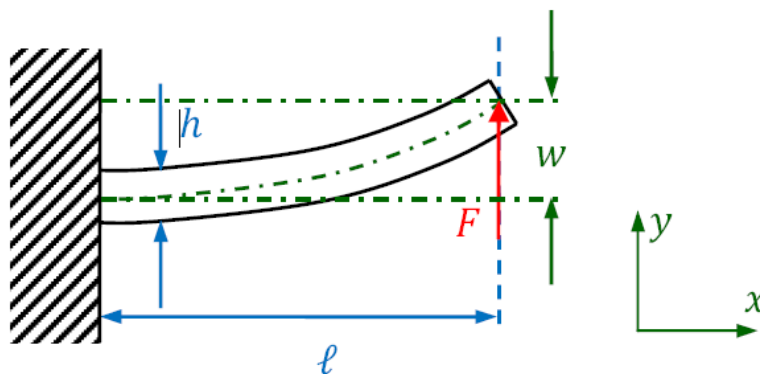

Figure 1: Cantilever beam with load *F* applied at the right end

Given:

Table 1: Geometry and material data

l	1000 mm	Beam length
h	60 mm	Beam height
t	40 mm	Beam thickness
F_1	22 000 N	Force during load step 1
F_2	−25 000 N	Force during load step 2
F_3	0 N	Force during load step 3
E	73 100 MPa	Young's modulus of aluminium
ν	0.33	Poisson's ratio
E_τ	7310 MPa	Tangent modulus
σ_{yield}	414 MPa	Yield strength

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.
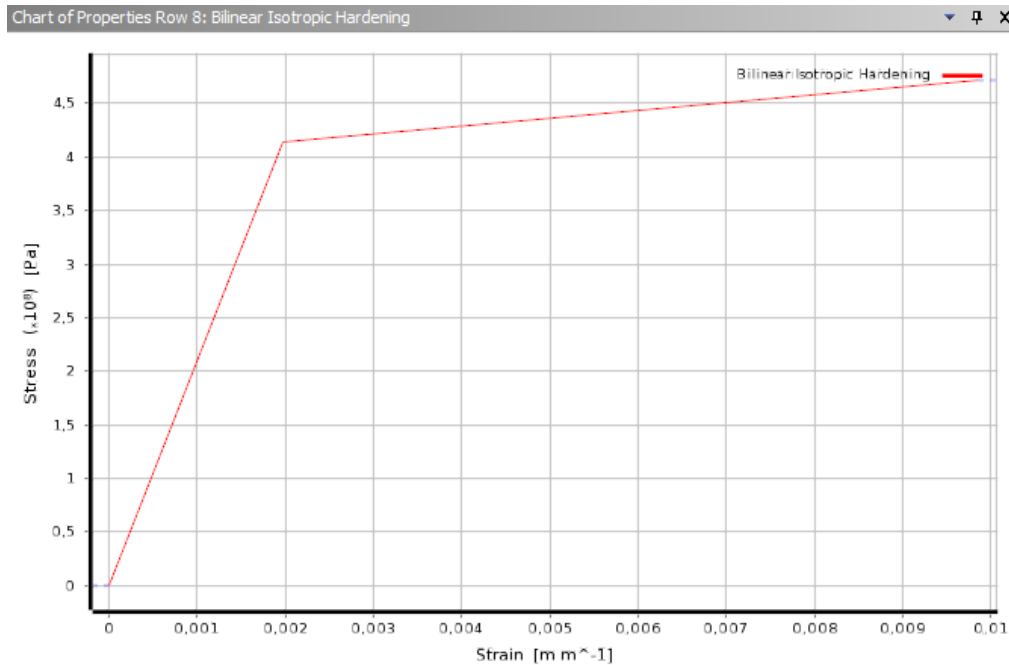

Figure 2: Stress-strain curve for our elasto-plastic material

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 (Figure 3, Table 1). Done correctly, the load history should look like Figure 4.
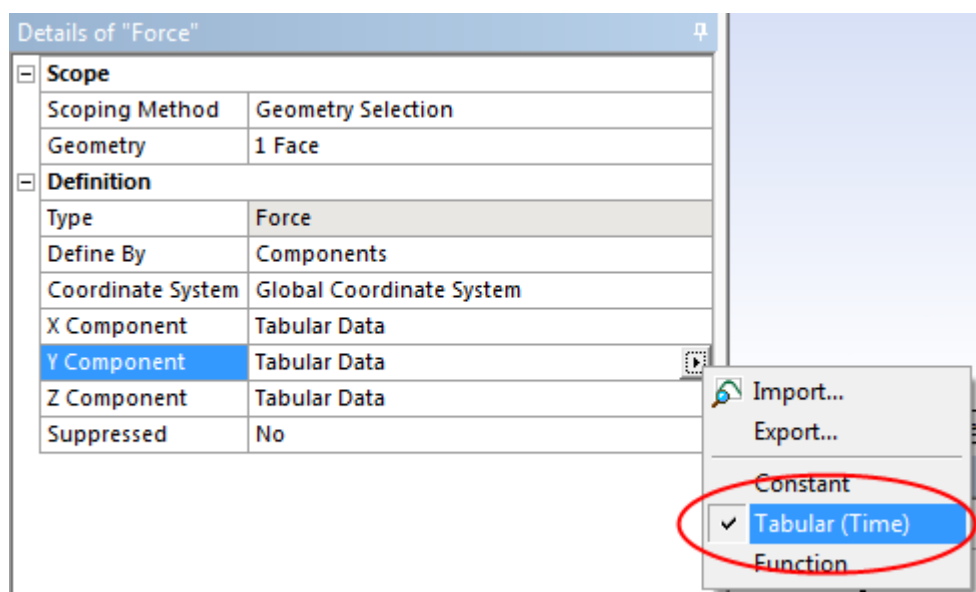

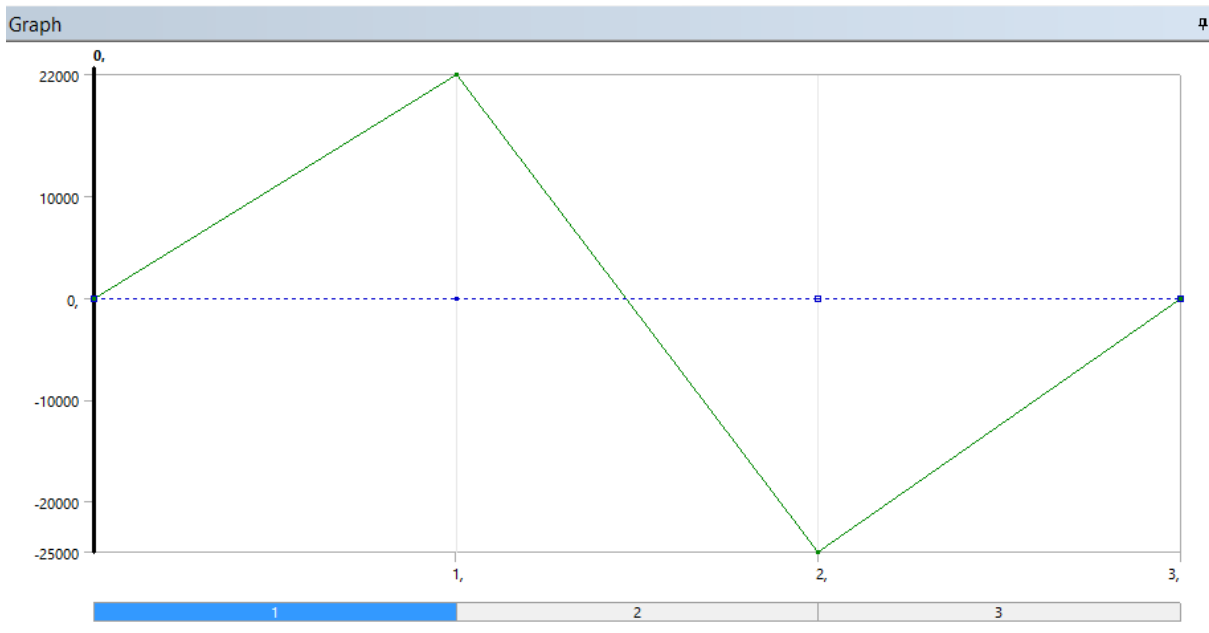Figure 3: Using tabular data for the force components

Figure 4: Load history with three distinct steps

Choose **Force Convergence** under **Solution (A6) → Solution Information → Solution Output**. Also, enable support for large deformations by setting **Analysis Settings → Solver Controls → Large Deflection** to **On**.

After solving, plot both the normal elastic strain in *x* direction (cf. Figure 5) as well as the normal plastic strain in *x* direction (click on **Worksheet** and right-click **EPPLX → Create User Defined Result**, see Figure 6 and Figure 7).
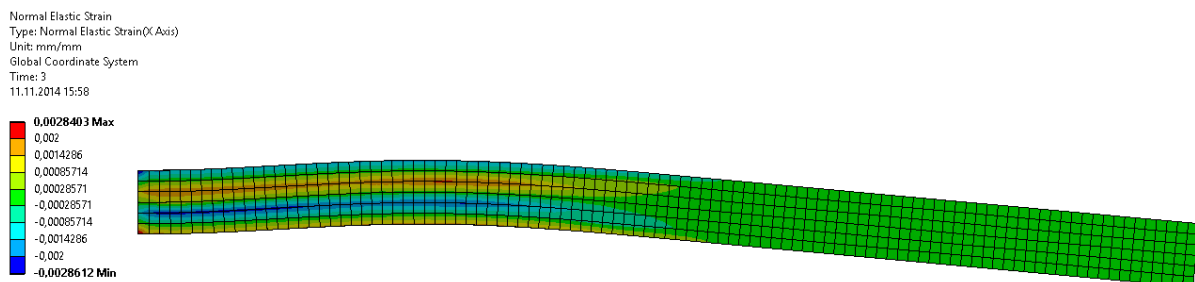

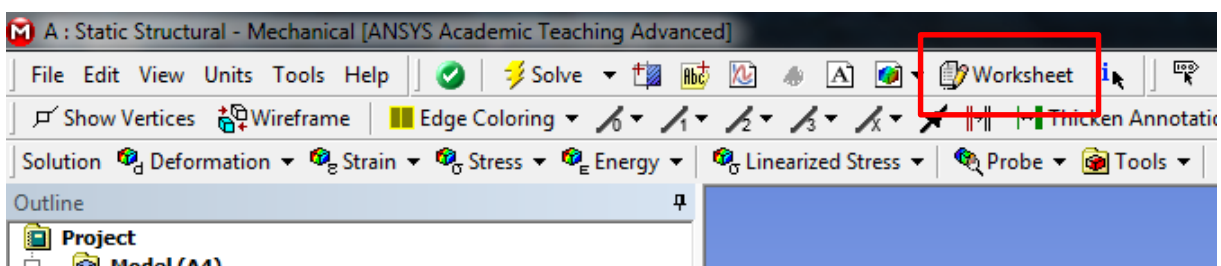Figure 5: Normal elastic strain in *x* direction after the last load step

Figure 6: In order to plot normal plastic strains first click the **Worksheet** button…

Type	Data Type	Data Style	Component	Expression	Output Unit
S	Element Nodal	Scalar	Z	SZ	Stress
S	Element Nodal	Scalar	XY	SXY	Stress
S	Element Nodal	Scalar	YZ	SYZ	Stress
S	Element Nodal	Scalar	XZ	SXZ	Stress
S	Element Nodal	Scalar	1	S1	Stress
S	Element Nodal	Scalar	2	S2	Stress
S	Element Nodal	Scalar	3	S3	Stress
S	Element Nodal	Scalar	INT	SINT	Stress
S	Element Nodal	Scalar	EQV	SEQV	Stress
S	Element Nodal	Tensor	VECTORS	SVECTORS	Stress
S	Element Nodal	Scalar	MAXSHEAR	SMAXSHEAR	Stress
EPEL	Element Nodal	Scalar	X	EPELX	Strain
EPEL	Element Nodal	Scalar	Y	EPELY	Strain
EPEL	Element Nodal	Scalar	Z	EPELZ	Strain
EPEL	Element Nodal	Scalar	XY	EPELXY	Strain
EPEL	Element Nodal	Scalar	YZ	EPELYZ	Strain
EPEL	Element Nodal	Scalar	XZ	EPELXZ	Strain
EPEL	Element Nodal	Scalar	1	EPEL1	Strain
EPEL	Element Nodal	Scalar	2	EPEL2	Strain
EPEL	Element Nodal	Scalar	3	EPEL3	Strain
EPEL	Element Nodal	Scalar	INT	EPELINT	Strain
EPEL	Element Nodal	Tensor Strain	VECTORS	EPELVECTORS	Strain
EPEL	Element Nodal	Scalar	MAXSHEAR	EPELMAXSHEAR	Strain
EPPL	Element Nodal	Scalar	X	EPPLX	Strain
EPPL	Element Nodal	Scalar	Y	EPPLY	Strain
EPPL	Element Nodal	Scalar	Z	EPPLZ	Strain
EPPL	Element Nodal	Scalar	XY	EPPLXY	Strain
EPPL	Element Nodal	Scalar	YZ	EPPLYZ	Strain
EPPL	Element Nodal	Scalar	XZ	EPPLXZ	Strain

Create User Defined Result

Figure 7: … and then add **EPPLX** as a **User Defined Result**

Questions

- Explain the pattern of elastic strains! Why are there still elastic strains remaining when the load is 0 N?
- How would the pattern change if we only applied the first and the last load step?
- Why have the solution times increased considerably compared to the previous beam models? (cf. the force convergence plot under **Solution (A6) → Solution Information**)

2. Wooden Hook

Up until now, we always used isotropic materials, i. e. materials whose properties are the same for every orientation. In reality, though, most materials exhibit more or less anisotropy. Wood, for instance, is a commonly used "organic fiber-reinforced composite," whose material properties parallel to the grain differ significantly from those in directions perpendicular to the fibers. Instead of a fully anisotropic material model (= 21 independent parameters, possible shear coupling) wood is often modeled as either an orthotropic (= 9 independent parameters, three orthogonal planes of symmetry) or transverse isotropic material (= 5 independent parameters, two elasticity moduli, rotational symmetry).

We will use an orthotropic material model to describe the mechanical behavior of a solid body made of a piece of wood. First, we need to agree on a material coordinate system. For wood,

it is common to consider the direction parallel to the grain the *x* (or longitudinal) direction; the direction normal to the growth rings is the *y* (or radial direction); and finally, *z* is the direction tangential to the growth rings (Figure 8).
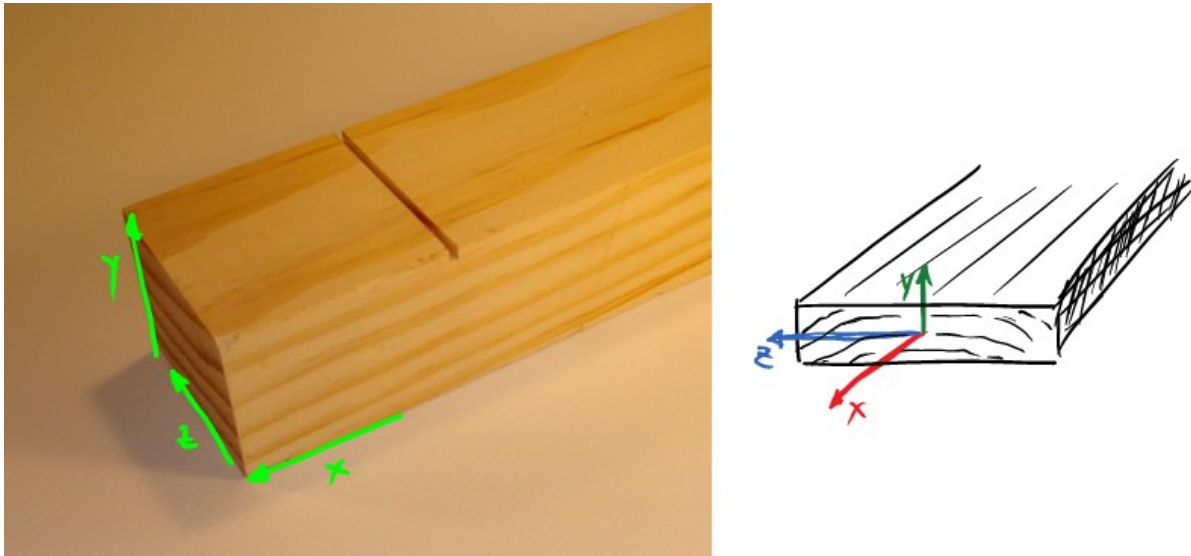

Figure 8: Right-handed material coordinate system for wood

Create a new **Static Structural** analysis system (or an entirely new project). Then add a new orthotropic material model (**Engineering Data** → **Linear Elastic** → **Orthotropic Elasticity**). Extract the 9 required material parameters from the following PDF file about the material properties of different wood types (pages 2 – 8)

   → http://www.woodweb.com/Resources/wood_eng_handbook/Ch04.pdf

You can choose any type of wood you'd like, e. g. pine. Just make sure that your material properties use the coordinate system of Figure 8.

Next, start **DesignModeler**. We will use a slightly more sophisticated geometry this time. Create the geometry using two box primitives to produce an L-shaped form, corresponding to Figure 9.

Ensure that the result is a *single body*, not two bodies/two separate parts! When you are done with the geometry, start the **Mechanical** module.

By default, the coordinate system of each solid body – and hence its material – is the global coordinate system. Because we want to investigate the influence of different material alignment, we need to define a custom coordinate system: Right click on **Coordinate Systems** in the **Outline** and insert a new **Coordinate System** (and give it a distinct name).

To rotate or translate the coordinate system, select it from the outline and click on the corresponding buttons in the toolbar (Figure 10). This will add **Rotate X/Y/Z** or **Translate X/Y/Z** properties to the **Transformations** section of your coordinate system where you can enter the desired values (rotation angles).

Don't forget to assign both the wood material *and* the new coordinate system to the solid body (**Geometry → Solid → Definition → Coordinate System**)!
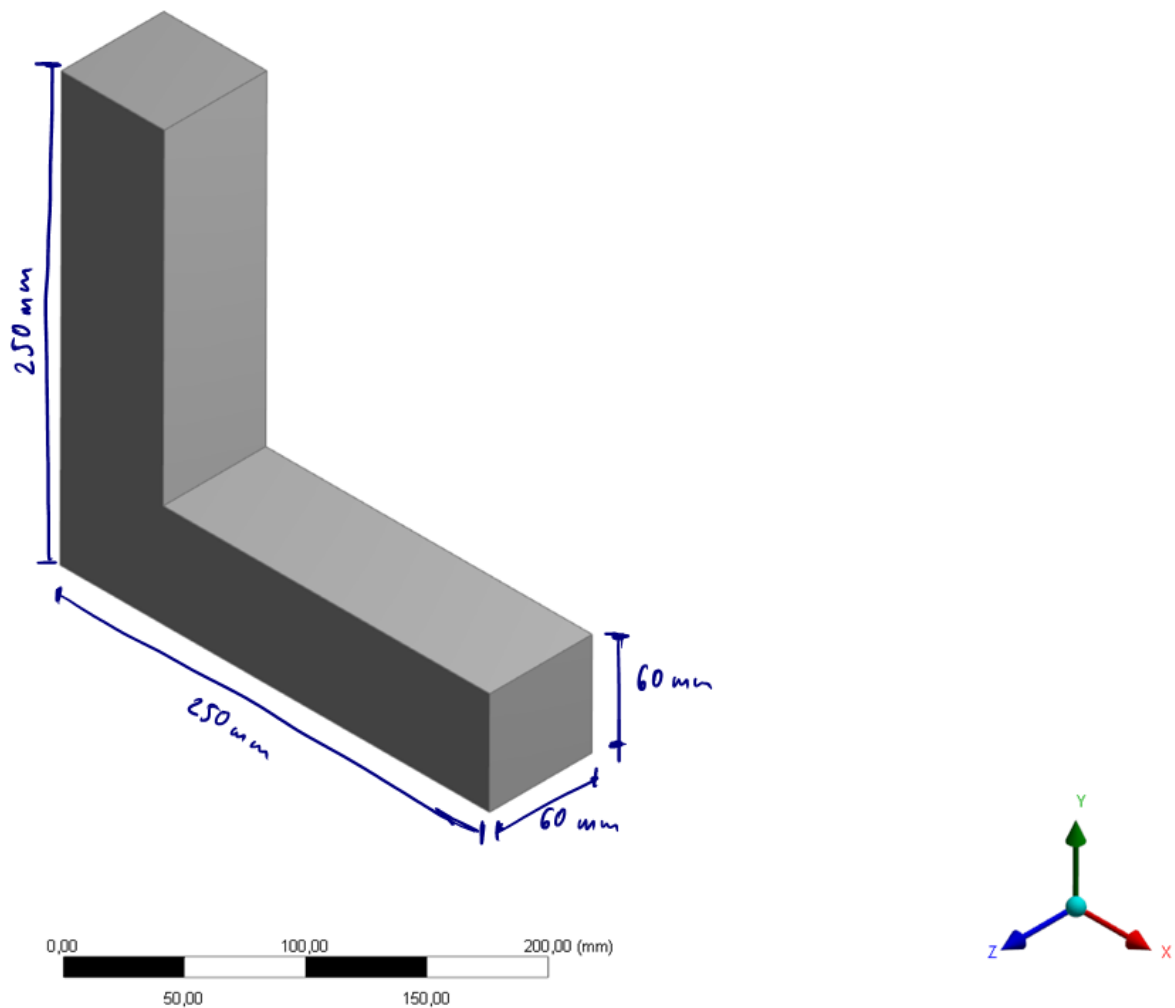

Figure 9: Dimensions of the L-shaped geometry

After creating an appropriate mesh, add load and boundary conditions as depicted by Figure 11. You may also want to switch on support for large deflections as we did in today's first exercise.
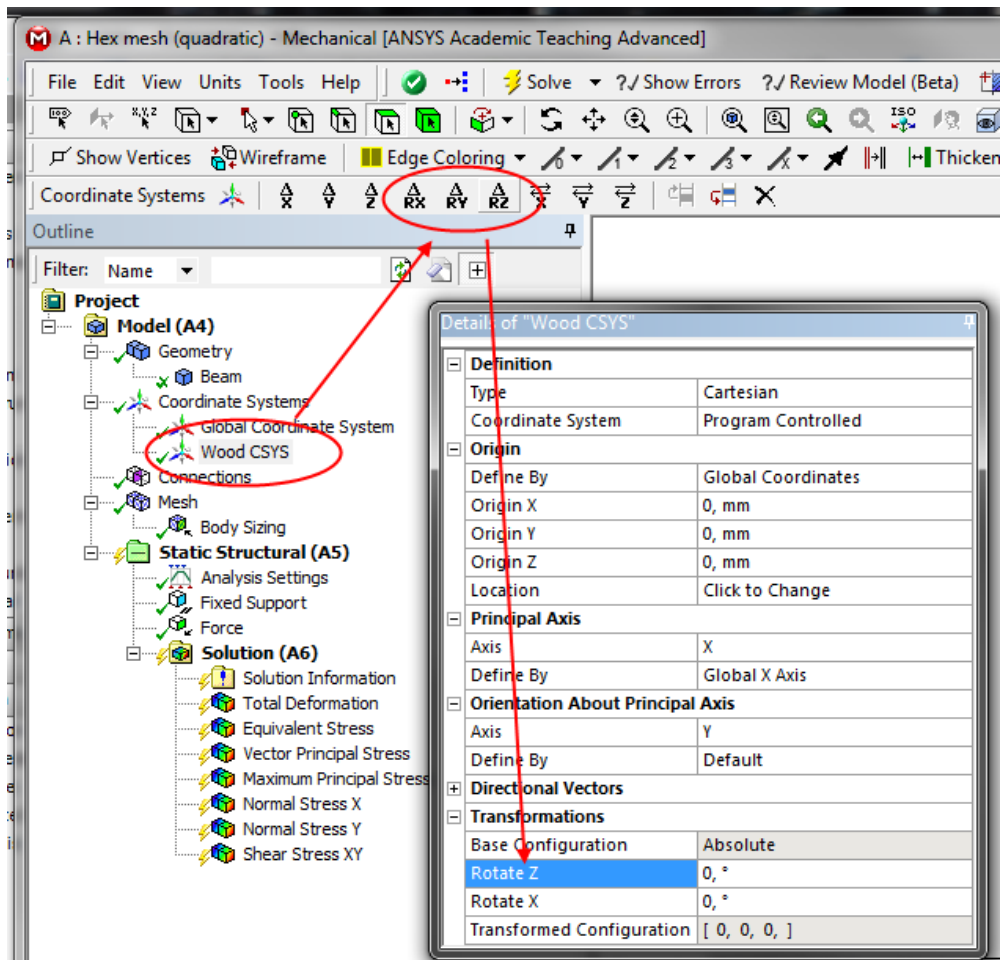
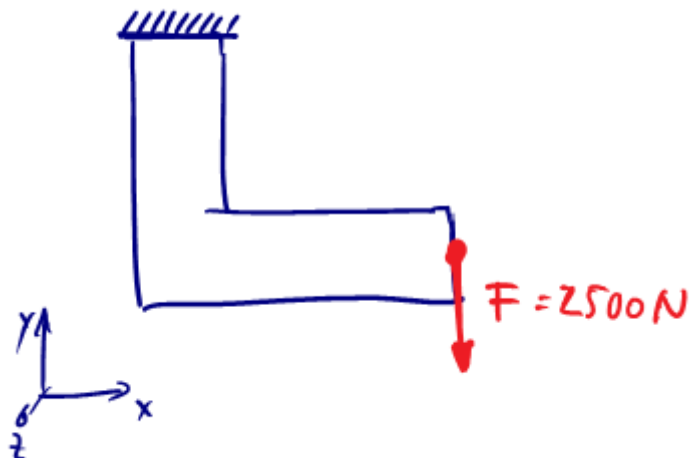Figure 10: Rotating and translating a custom coordinate system

Figure 11: Load and boundary conditions applied to the wooden structure

Questions

- Why didn't we have to define v_{yx}, v_{zy} and v_{zx}?
- Plot and compare principal strains and stresses (vector plots). What would you expect to see, had we used an isotropic material instead?
- Where do you see compression, where tension?
- Try out different fiber alignments (e. g. 45°, 90°, 135°, –45° …). Which material orientation leads to the lowest amount of deformation? Why?
- Optimize the wooden part a bit further and subdivide the single solid body into two independent, but "glued together" solid bodies (**Design Modeler → Create → New Plane** to define the slice plane orientation; **Create → Slice** to slice the body in two). This way, each body can define its own unique fiber orientation (just as if it was made of two wooden blocks). How do you need to align the wood of the two parts to minimize the deflection? Why?
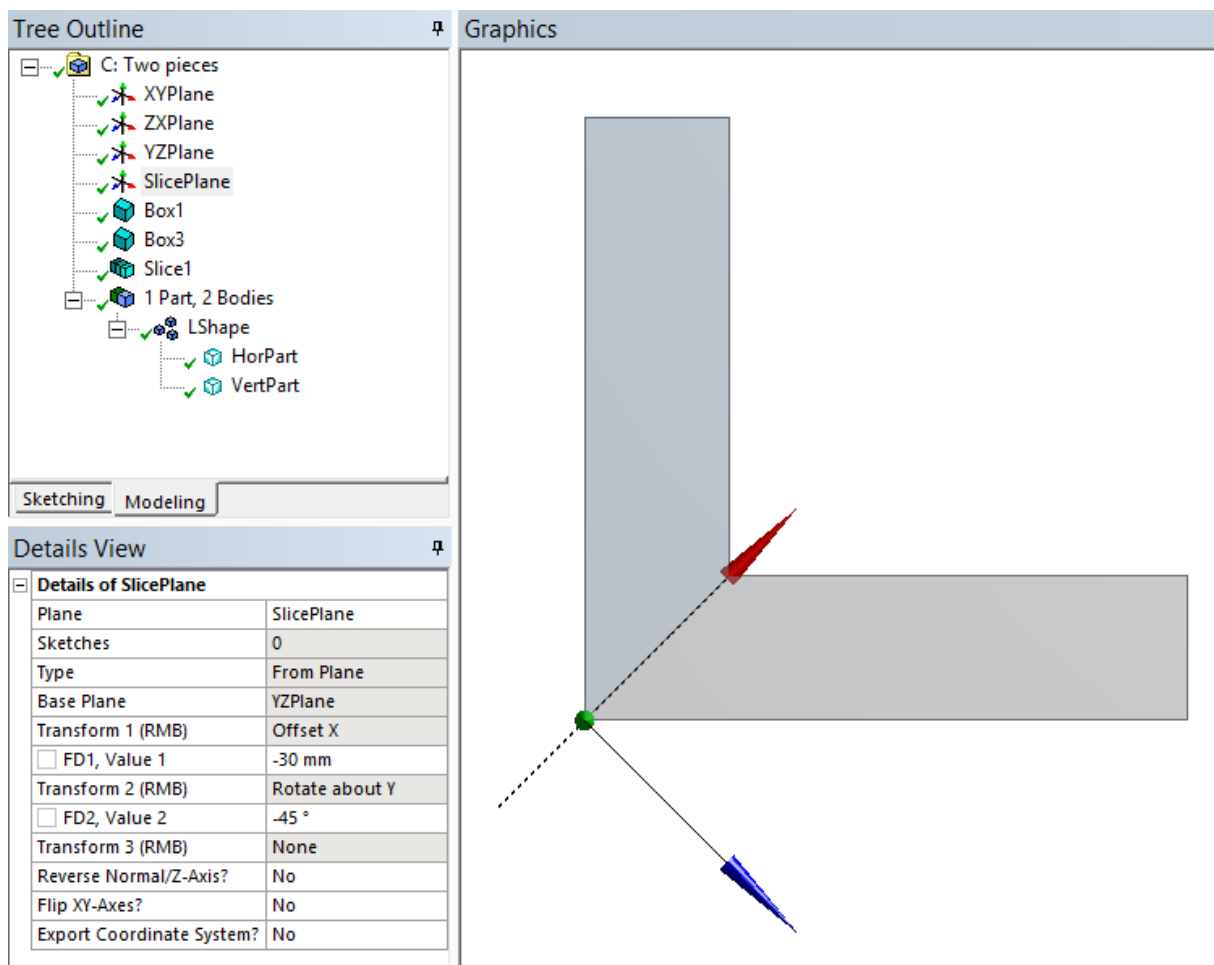- What is the deflection for the opposite configuration, i. e. when swapping the orientations of the two parts?

Figure 12: Subdivided part