

**EXERCISE 4: PLASTICITY AND ANISOTROPY****CONTENTS**

Goals .....	2
<b>A. Cold Forming</b> .....	2
<b>B. Anisotropic Material</b> .....	5
<b>C. Bonus: Anisotropy &amp; 2D Modeling</b> .....	9

## GOALS

Today we 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.

## A. 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 the **Design Modeler**.

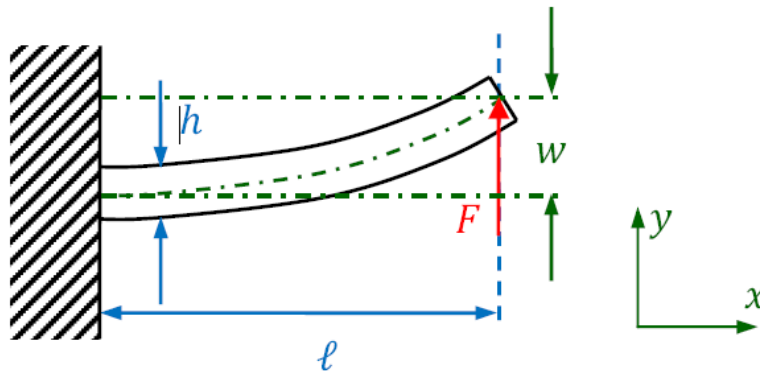


Figure 1: Cantilever beam with applied load at the right end

$l$	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
$\nu$	0,33	Poisson's ratio
$E_t$	7310 MPa	Tangent modulus
$\sigma_{\text{yield}}$	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.

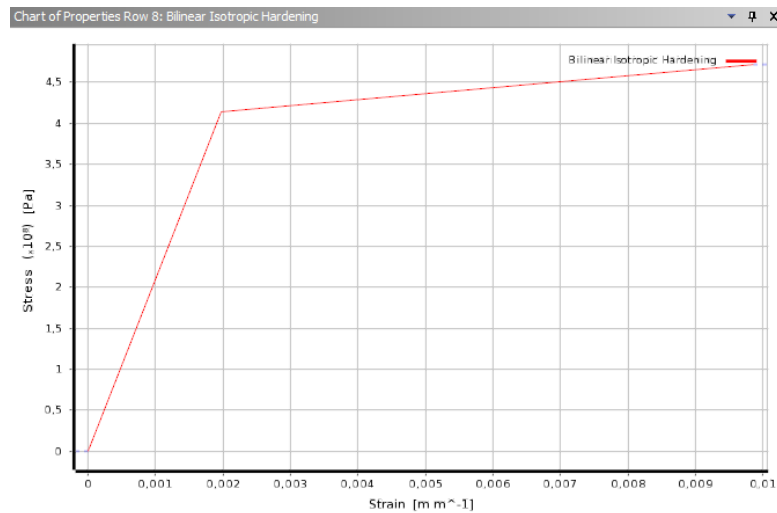


FIGURE 2: STRESS-STRAIN CURVE FOR OUR ELASTO-PLASTIC MATERIAL

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.

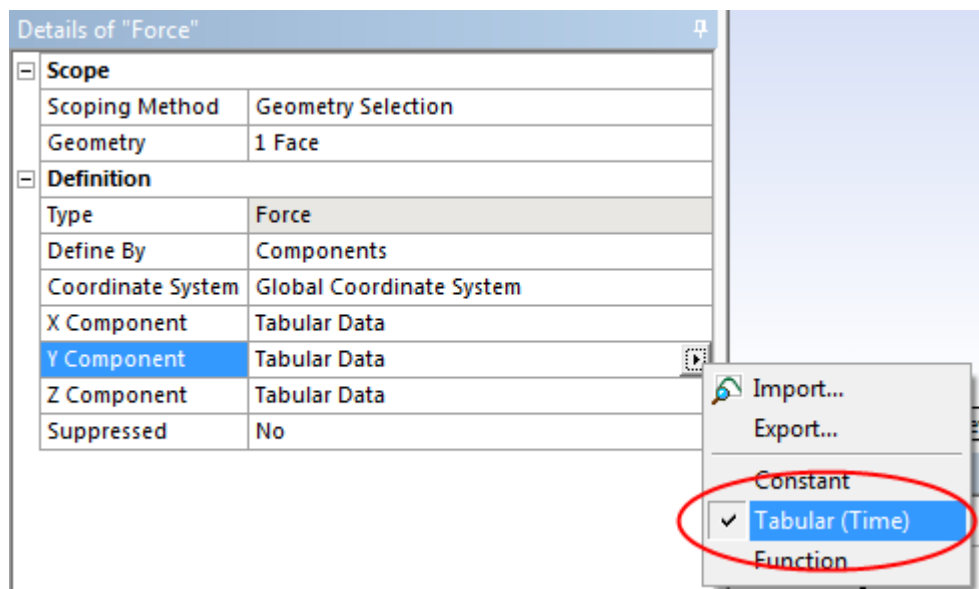


FIGURE 3: USING TABULAR DATA FOR THE FORCE COMPONENTS

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.

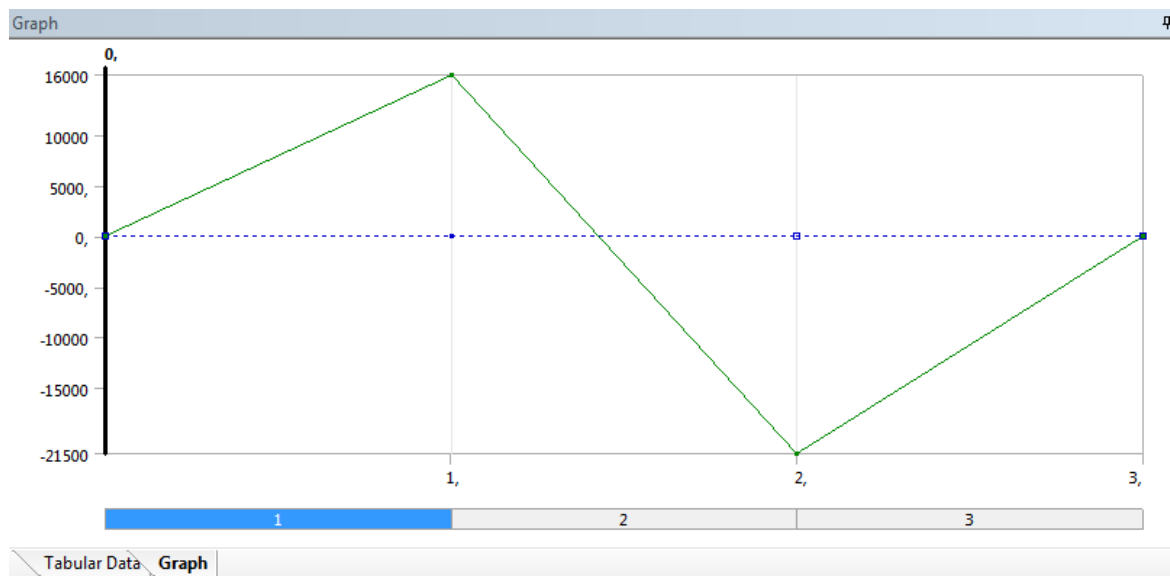


FIGURE 4: LOAD HISTORY WITH THREE DISTINCT STEPS

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

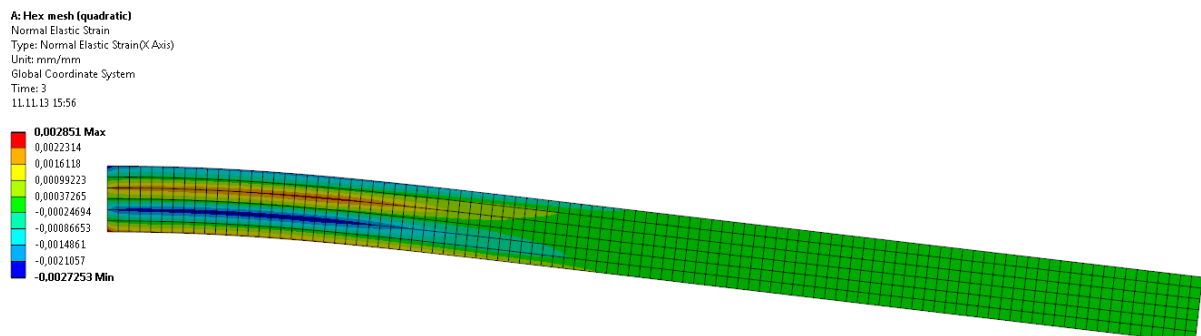


FIGURE 5: NORMAL ELASTIC STRAIN IN X DIRECTION AFTER THE LAST LOAD STEP

## 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?

## B. Anisotropic Material

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 anisotropy to some extent. 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 6).

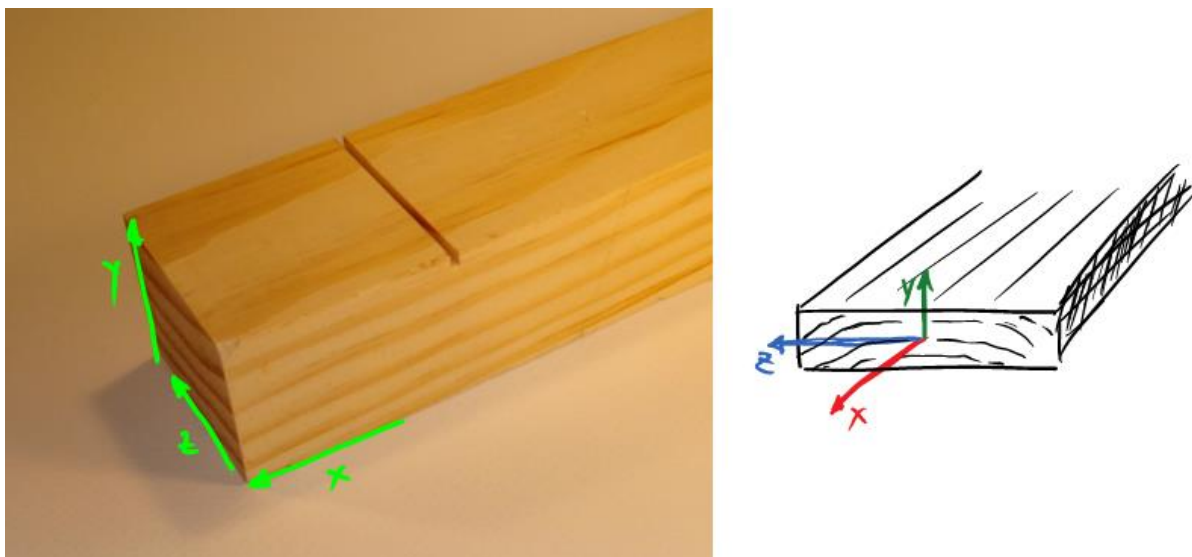


FIGURE 6: MATERIAL COORDINATE SYSTEM FOR WOOD

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](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 6.

Next, start **Design Modeler**. 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 7.

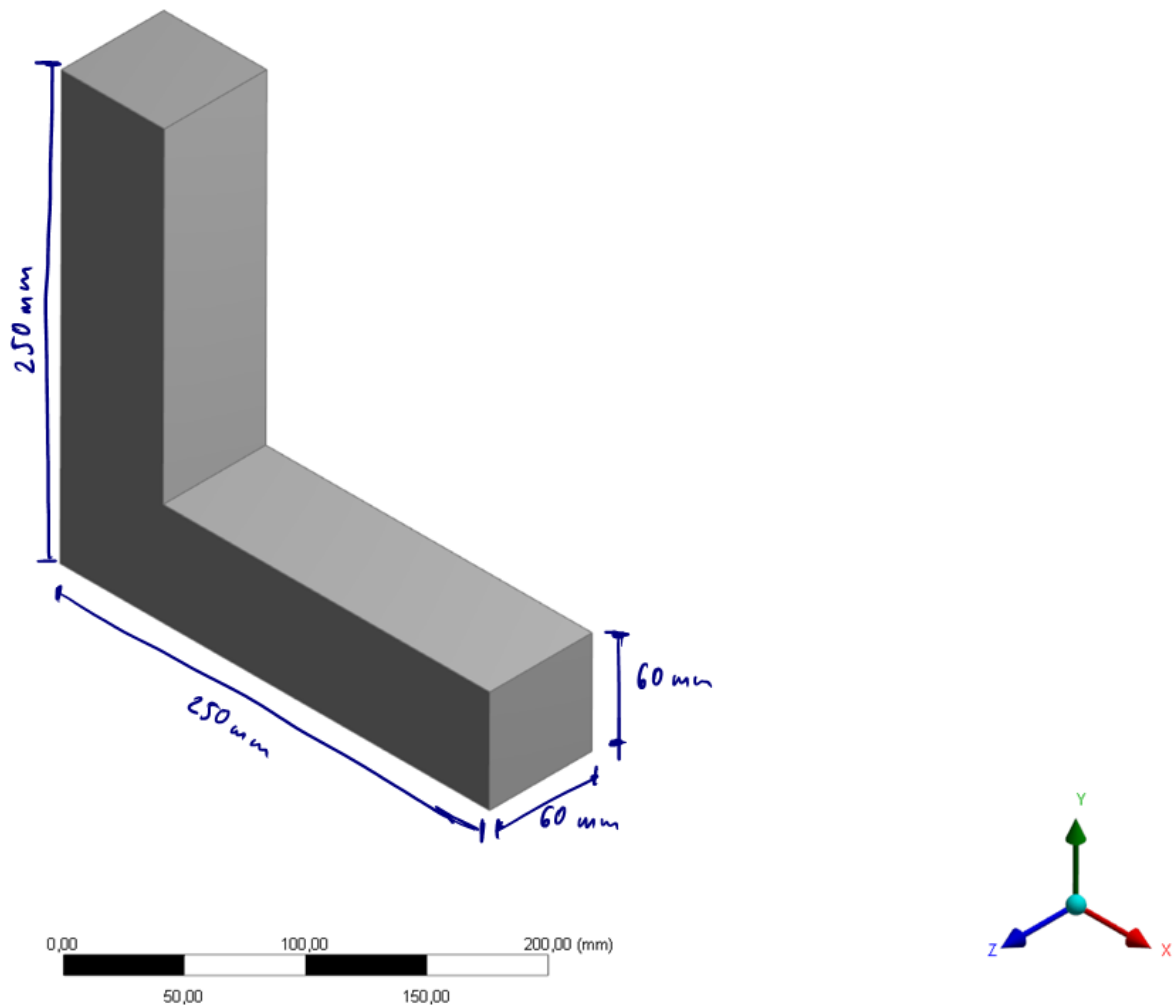


FIGURE 7: DIMENSIONS OF THE L-SHAPED GEOMETRY

Ensure that the result is a *single body*, not two bodies/two 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 8). 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).

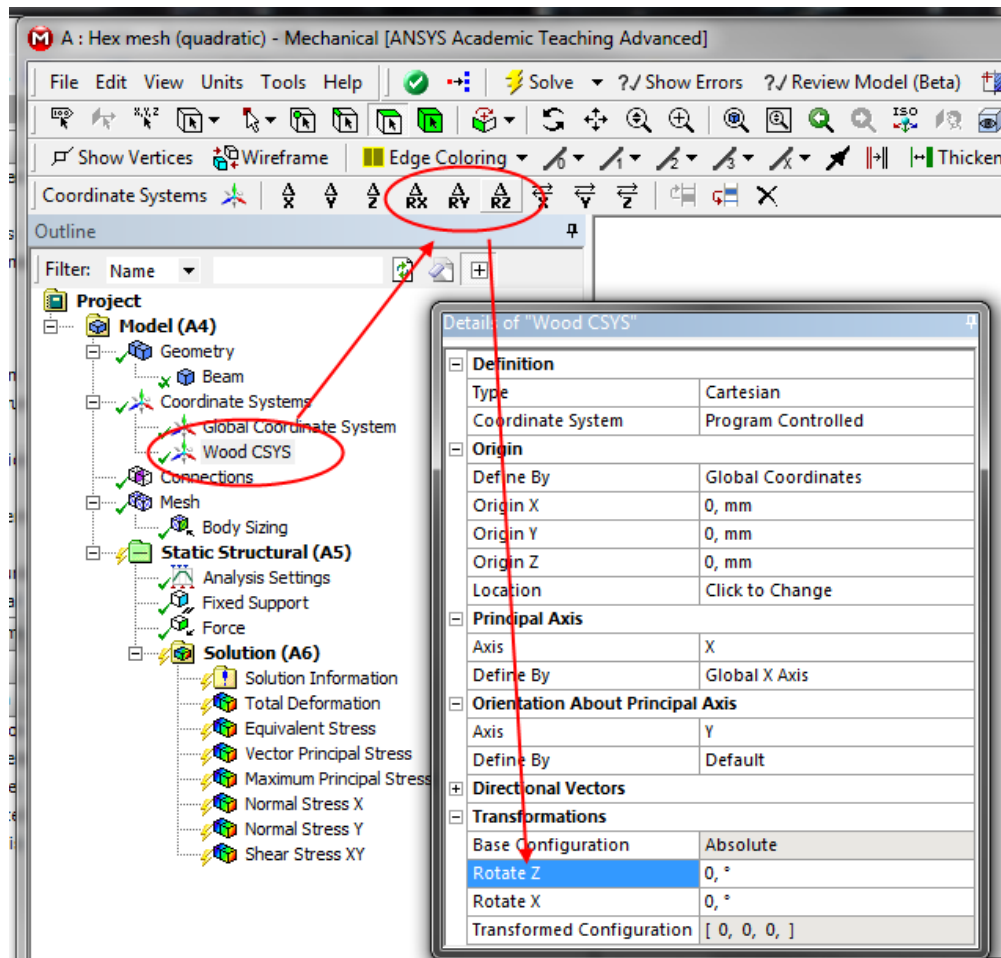


FIGURE 8: ROTATING AND TRANSLATING A CUSTOM COORDINATE SYSTEM

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

After creating an appropriate mesh, add load and boundary conditions as depicted by Figure 9. You may also want to switch on support for large deflections as we did in today's first exercise.

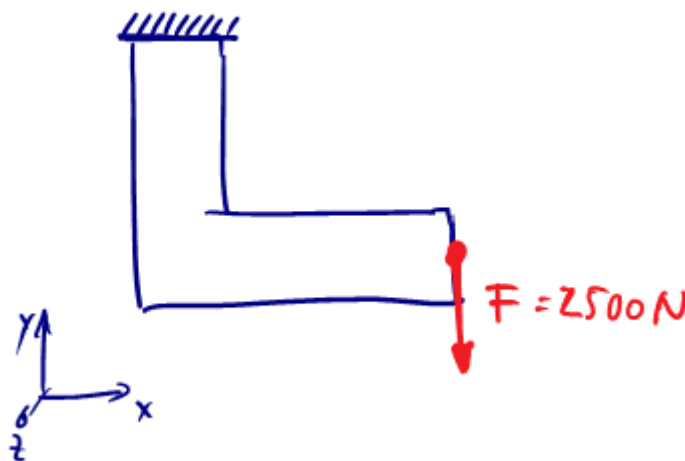


FIGURE 9: LOAD AND BCS APPLIED TO THE WOODEN STRUCTURE

## Questions

- Why didn't we need to define  $v_{yx}$ ,  $v_{zy}$ ,  $v_{zx}$ ?
- 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?

**A: Hex mesh (quadratic)**  
 Vector Principal Stress  
 Type: Vector Principal Stress  
 Unit: MPa  
 Time: 1  
 12.11.13 15:15

Maximum Principal  
 Middle Principal  
 Minimum Principal

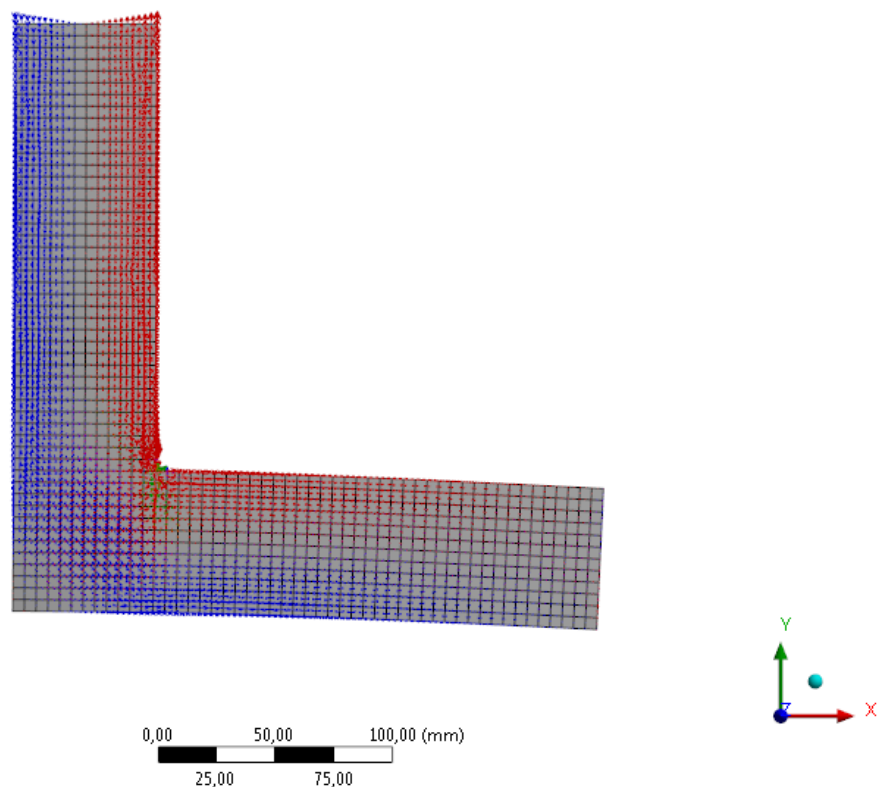


FIGURE 10: PRINCIPAL STRESSES IN THE WOODEN STRUCTURE (VECTOR PLOT)



### C. Bonus: Anisotropy & 2D Modeling

We want to optimize our wooden part a bit further and replace the single solid body by two independent, but “glued together” solid bodies. This way, each body can define its own unique fiber orientation (just as if it was made of two wooden blocks). Instead of a fully three-dimensional model however, we will use an analogous plane stress 2D model this time.

Create a new project and add a **Static Structural** analysis system. Define the wood material as before. Back in the **Project Schematic**, right click **Geometry** and select **Properties**. In the properties window, set the **Analysis Type** to **2D** (Figure 11).

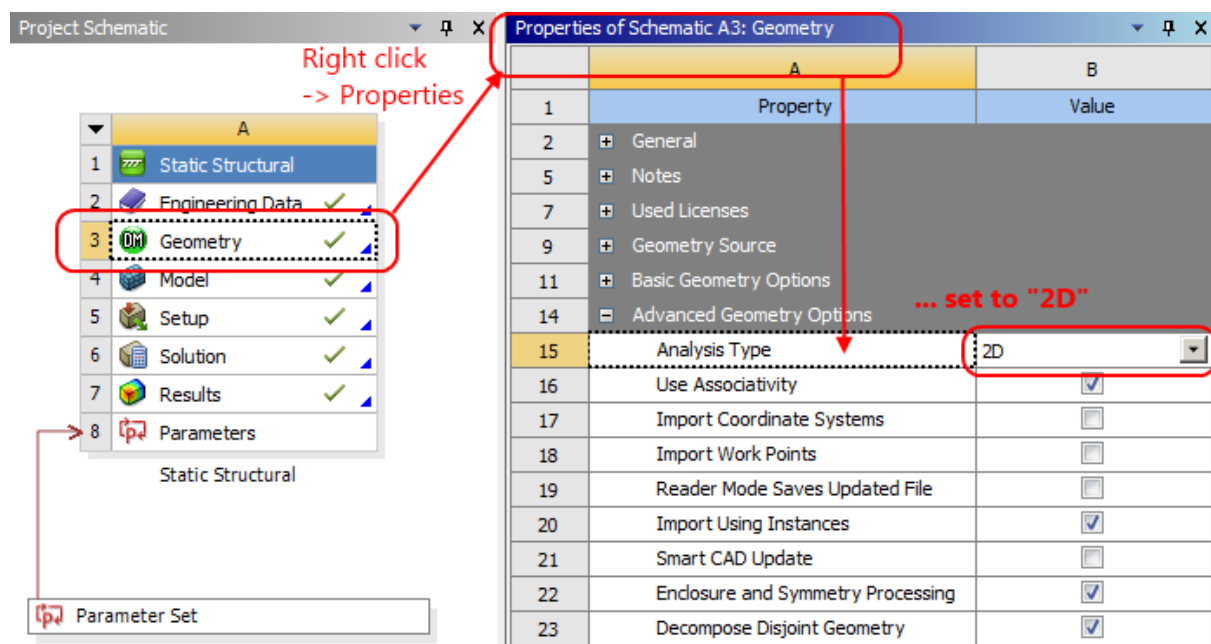


FIGURE 11: CHANGING THE ANALYSIS TYPE TO 2D

Now, launch **DesignModeler** and add a sketch to the **XYPlane** (see D.1 of lab 2 for details). Sketch the edges of the horizontal part of the L-shape (Figure 12). You may again consult the previous lab for details on how to add dimensions (lengths, angles) etc.

When you are done sketching, insert a surface body (**Concept** → **Surfaces from Sketches**). Select **Sketch1** as the Base Object and set **Operation** to **Frozen**. Next, we want to create the vertical part of the L-shape. After the **SurfaceSk1** item, insert a **Body Operation** (**Create** → **Body Operation**). Set **Preserve Bodies?** to **Yes** (we want to make a copy), set its type to **Rotate**, its **Z Component** to 1 and the **Angle** to 90°. After this body operation, insert a **Surface Flip** (**Tools** → **Surface Flip**) and after that another body operation. In this case, set **Preserve Bodies?** to **No**, set the **Y Component** to 1 and the **Angle** to 180°. Clicking the Generate button should create two parts and two bodies. Group both bodies into a single part (Figure 13).

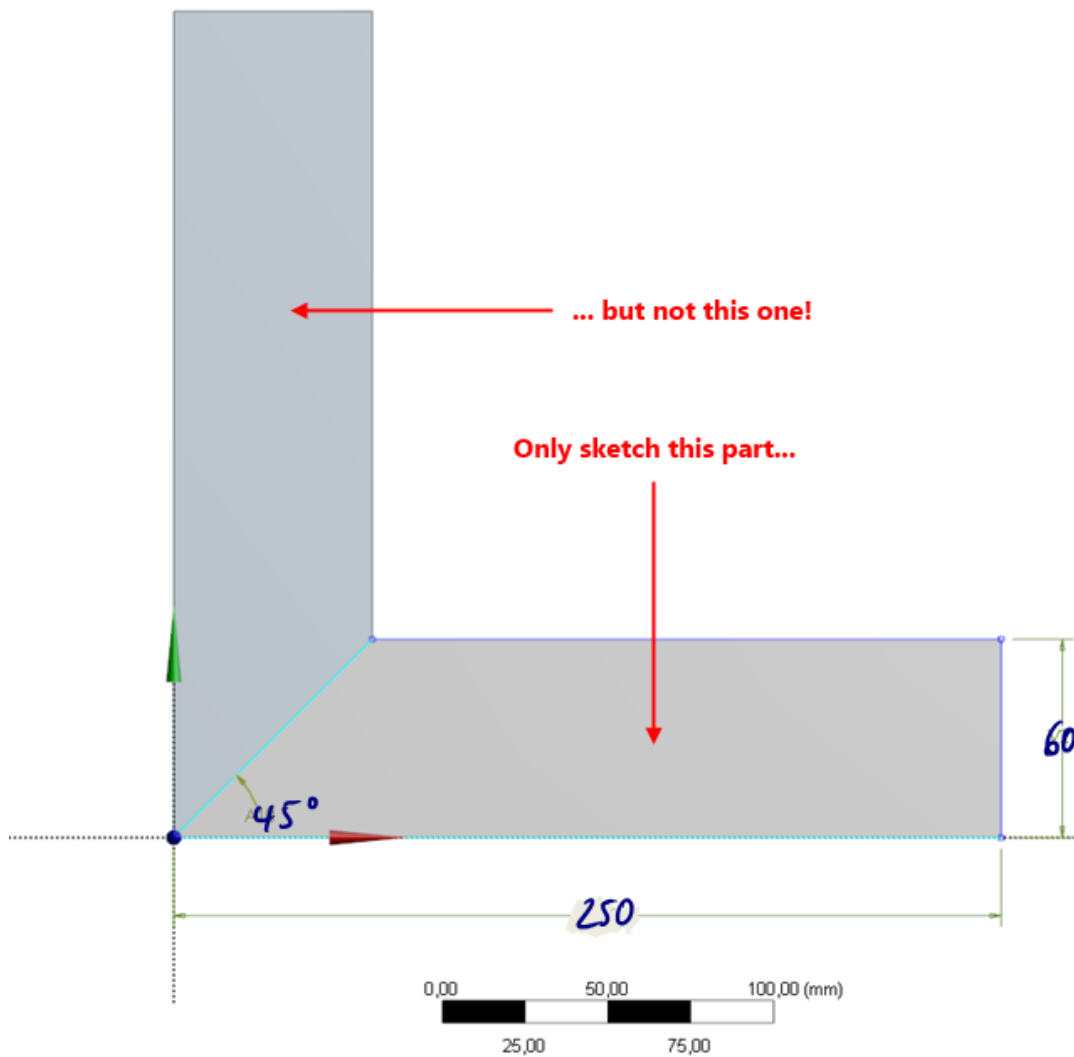


FIGURE 12: SKETCHING THE 2D GEOMETRY

You can close the **DesignModeler** and launch the **Mechanical** module. Assign a unique custom coordinate system to each of the two bodies (horizontal and vertical part of the L-shape). Also make sure that the **Thickness** property of each body is set to 60 mm. Create a mesh and apply load and boundary conditions analogous to those of the 3D model.

### Questions

- 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?

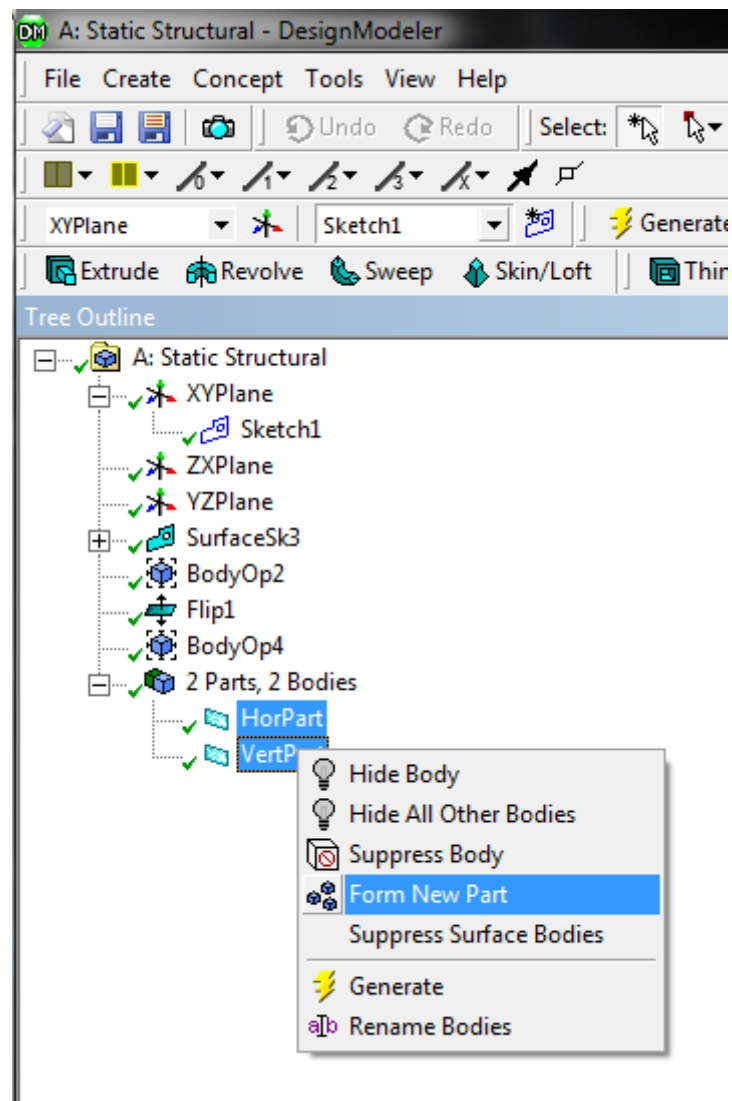


FIGURE 13: GROUPING BOTH SOLID BODIES INTO A SINGLE PART