Exercise 1

3-Point Bending Using the Static Structural Module of Ansys Workbench 14.0

Contents

Learn how to … 1
Given 2
Questions 2
Taking advantage of symmetries 2
A. Getting started 3
A.1 Choose the correct ANSYS license 3
A.2 Start the FE software ANSYS 3
B. Preprocessor (Setting up the Model) 4
B.1 Build the Geometry Using DesignModeler Module 4
B.2 Material Properties 6
i. Define Material Properties 6
B.3 Meshing 8
B.4 Applying Loads and Boundary Conditions 9
i. Boundary Conditions 9
ii. Applying the Loads 10
A. Solving 11
a. Contour Plot of Deformed Shape 11
b. Contour Plot: X-Component of Total Strain 12
c. Contour Plot X-Component of Stress 13
d. Contour Plot: Von-Mises Equivalent Stress 15
Answering the Questions: 15

Learn how to …

… use the ANSYS Workbench DesignModeler

… build a model

… take advantage of symmetries

… perform a full analysis consisting of pre-processing, solution and post-processing

Given

Beam under 3-point bending with a centric applied force F as shown in Figure 1

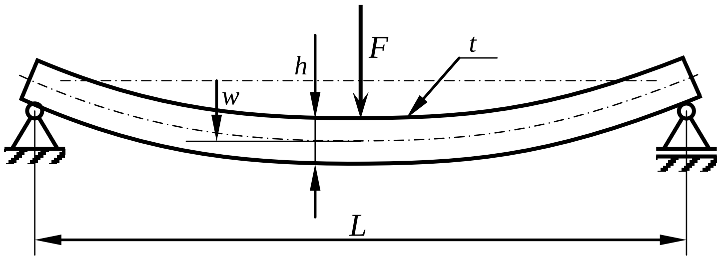


Figure 1: Beam under 3-point bending.

Relevant geometrical and material data for our problem are given in Table 1:

F	= 500,000 N	Applied force
L	= 2,000 mm	Length of the beam
h	= 60 mm	Height of the beam cross section
t	= 20 mm	Thickness of the beam cross section
E	= 210,000 N/mm^2	Young's modulus
ν	= 0.3	Poisson's ration
σ_{yield}	= 235 N/mm^2	Allowable stress: yield stress of steel

Table 1: Geometry and material data.

Questions

Due to this classic 2-dimensional mechanical problem we can state two questions:

1. Will the beam break and were would it start breaking?
2. If not, what would be the maximum deflection *w*?

Taking advantage of symmetries

Can we take advantage of symmetries? Please, draw a simplified beam model which takes advantage of symmetry!

Figure 2: Space for drawing of simplified beam model taking advantage of symmetry.

A. Getting started

A.1 Choose the correct ANSYS license

Before starting ANSYS Workbench, run "User License Preferences 14.0" and ensure that "ANSYS Academic Teaching Advanced" is the preferred license (Figure 3).

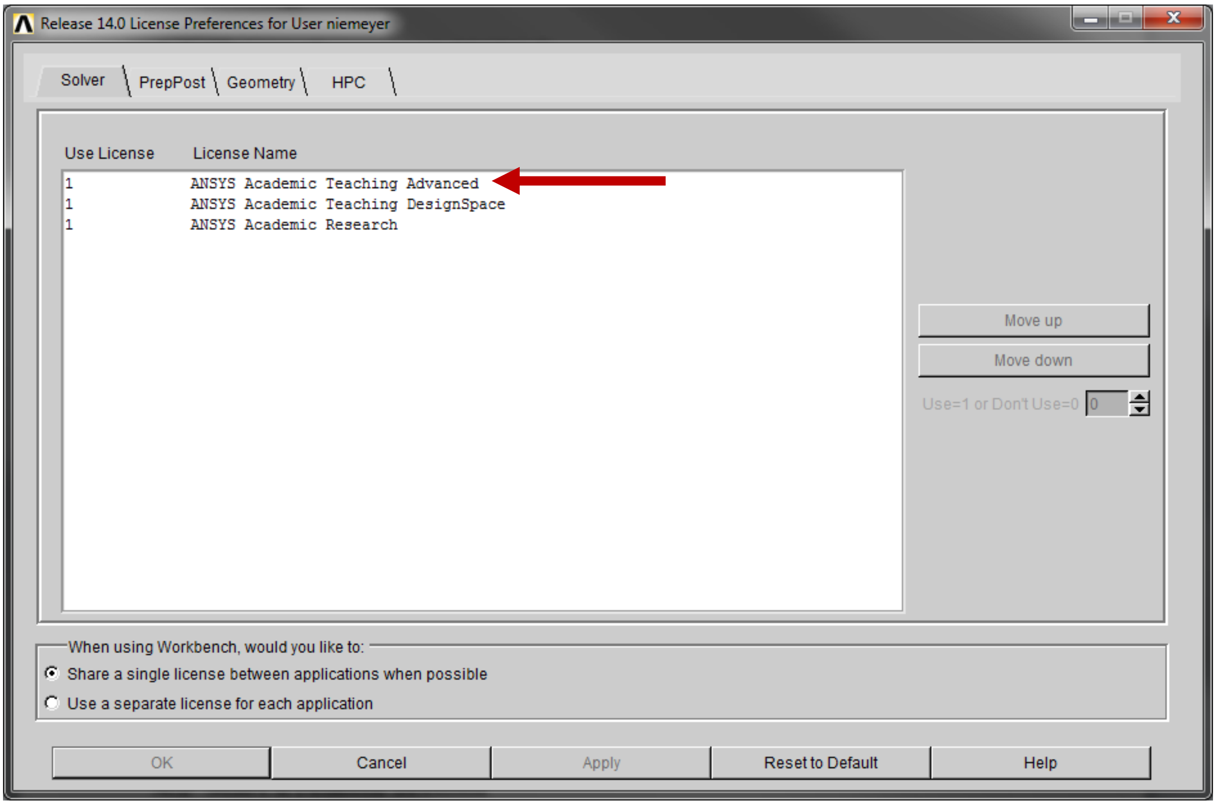


Figure 3: Select "ANSYS Academic Teaching Advanced" as the preferred license by moving it to the top of the list.

A.2 Start the FE software ANSYS

Start ANSYS Workbench by running "Workbench 14.0" (Figure 4).

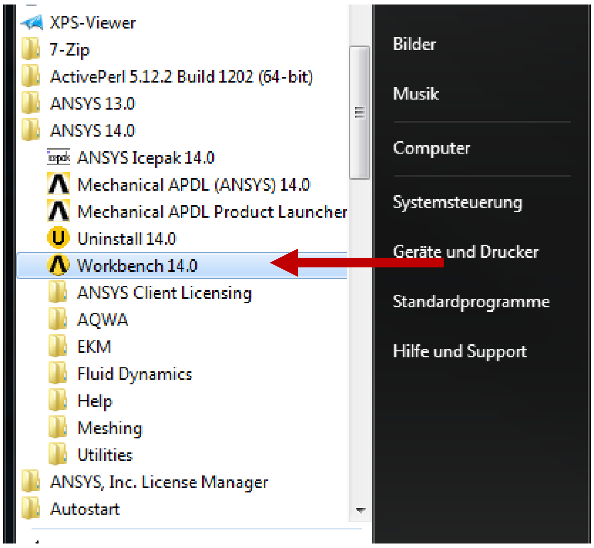


Figure 4: Starting ANSYS Workbench 14.0

B. Preprocessor (Setting up the Model)

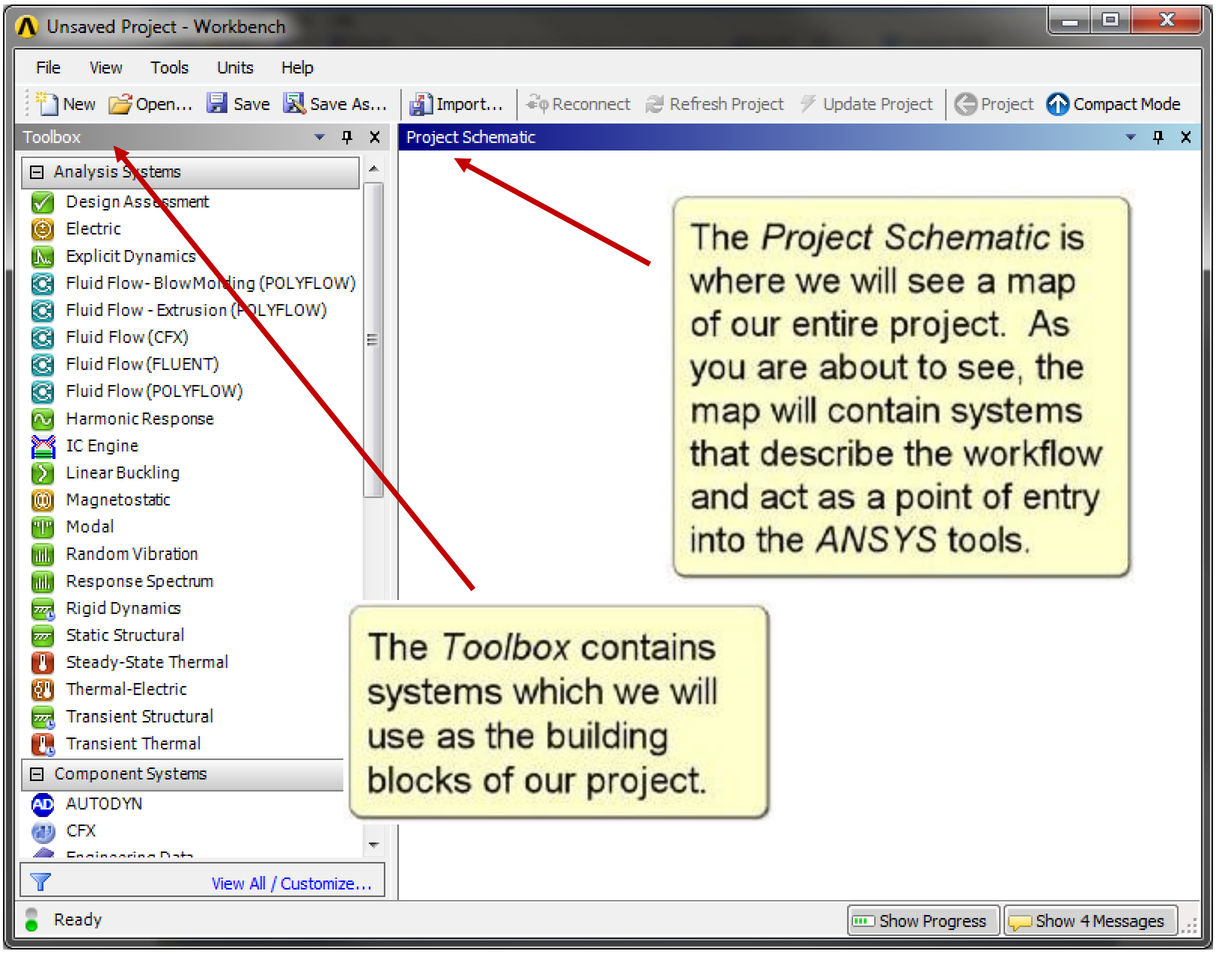


Figure 5: Interface of ANSYS Workbench 14.0

B.1 Build the Geometry Using DesignModeler Module

To begin; drag the **Static Structural Module** from the **Analysis Systems** toolbox and drop into the **Project Schematic** (Figure 6) and double click on the **Geometry** sub-module to open the **DesignModeler**. When asked, choose the desired unit system.

Choose **Create** → **Primitives** → **Box** from the main menu to create a beam by entering the desired dimensions in the **Details** box (Figure 7). Hit the **Generate** button to actually create the geometry.

When specifying the dimension of the beam, make sure that the origin of the coordinate system is located on the center of the beam's cross section. The beam axis must be oriented along the global x-axis.

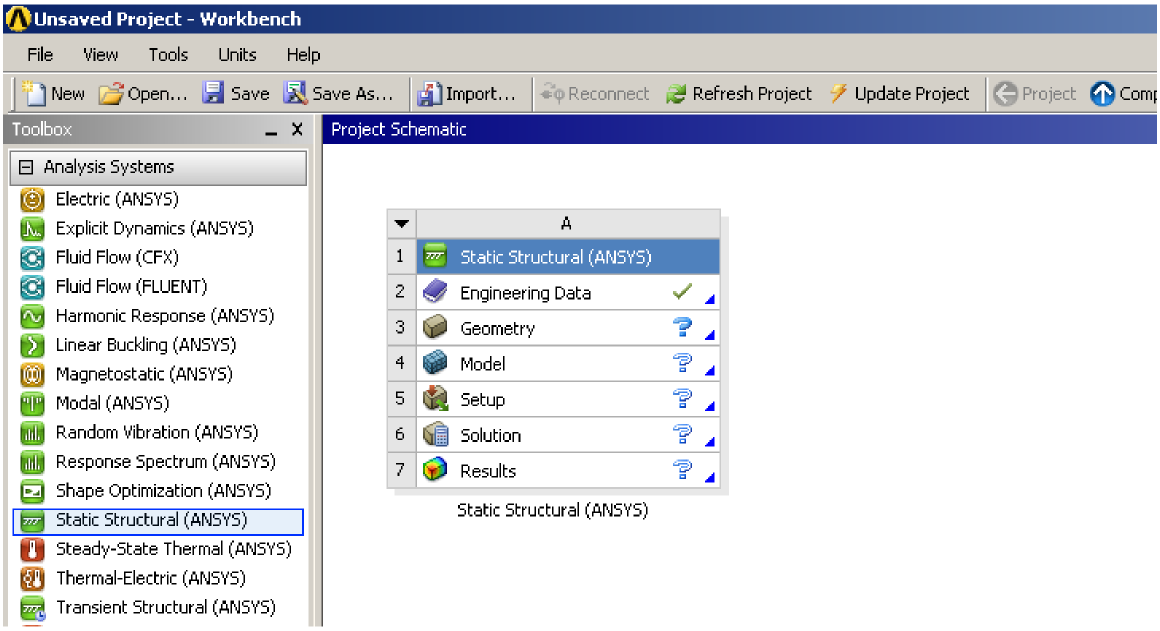


Figure 6: Drag-and-drop the Static Structural module to create a new analysis.

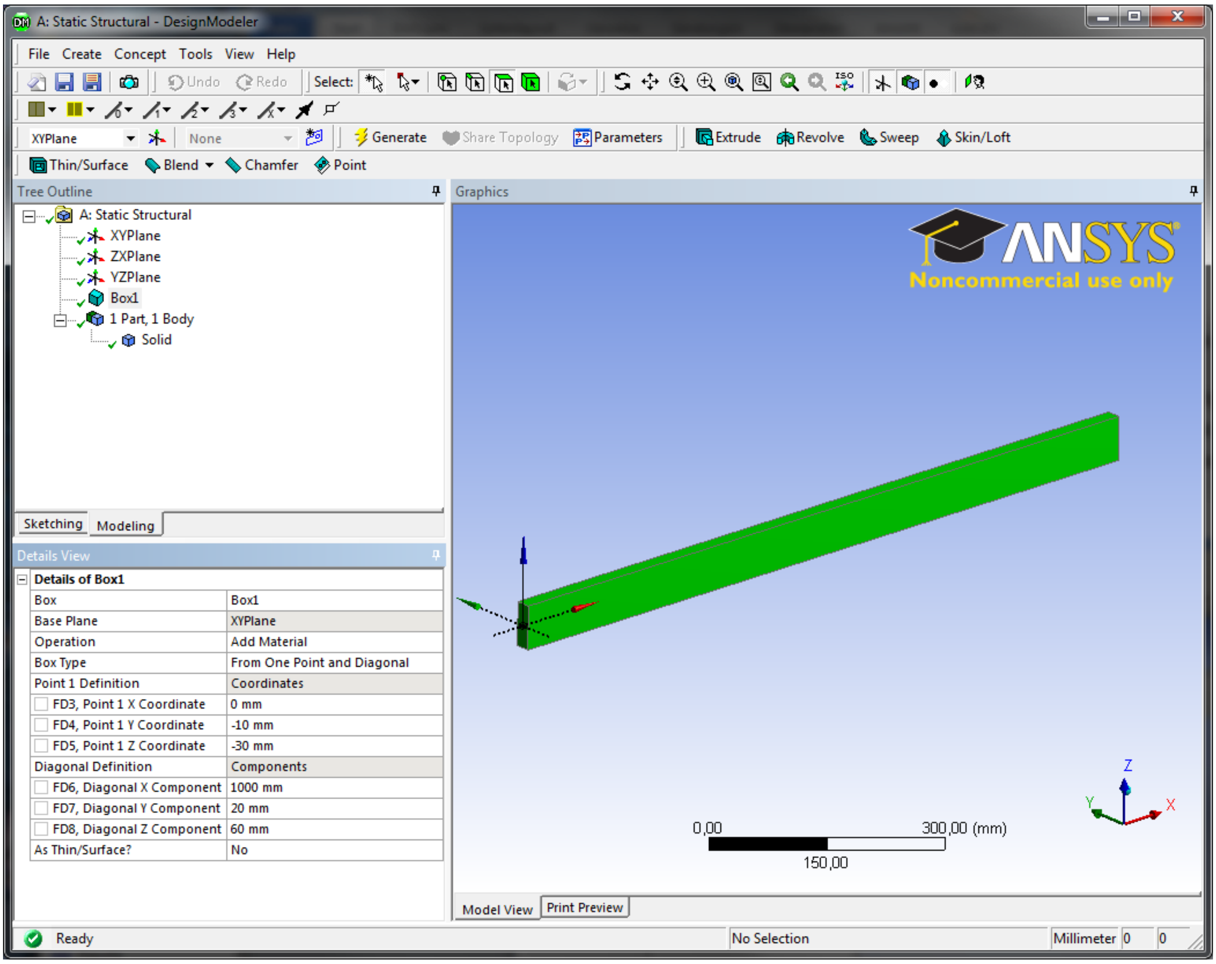


Figure 7: The beam on the DesignModeler

Note that ANSYS Workbench saves the model automatically; you can simply close **DesignModeler** now and continue with the next step.

B.2 Material Properties

i. Define Material Properties

Materials define the mechanical behavior of the FE model. We will use a simple linear-elastic, isotropic material model. In the project schematic, right click on **Engineering Data** to open a context menu and choose **Edit...** (Figure 8).

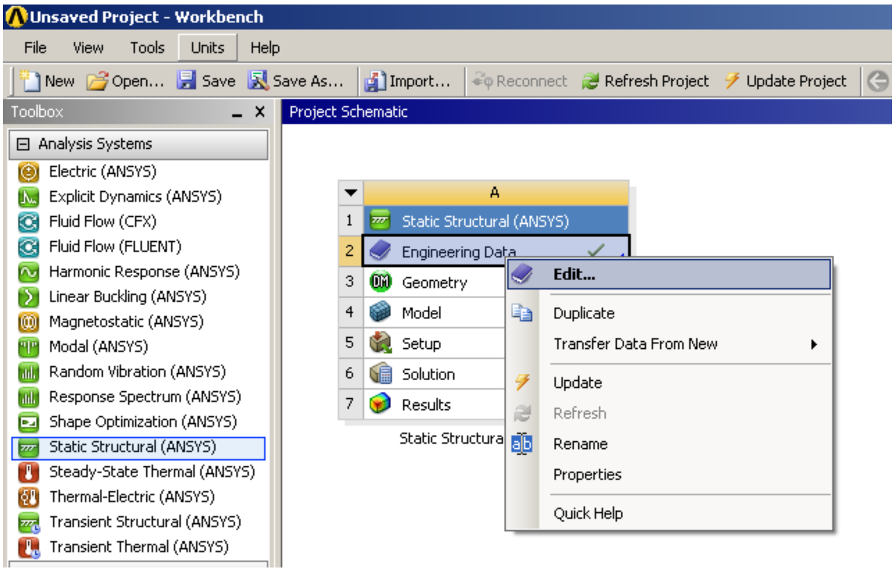


Figure 8: Edit engineering data

Enter a name for your new material in row 4 (Figure 9).

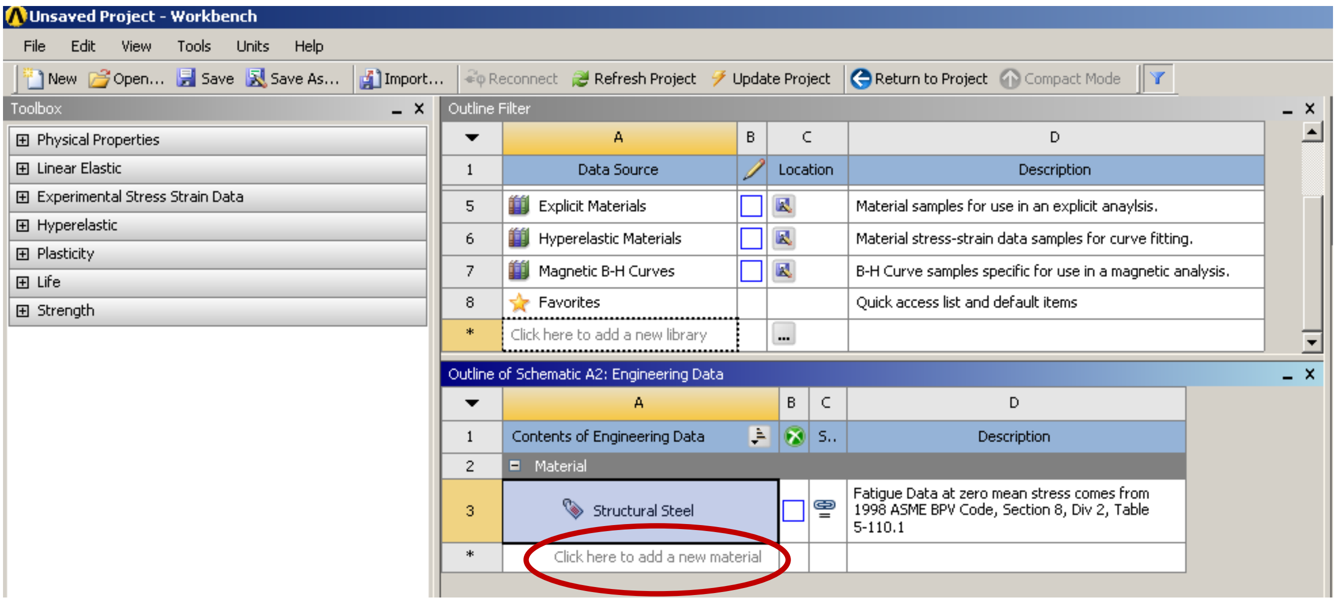


Figure 9: Naming the new material

Choose the simplest available material model by dragging the item **Isotropic Elasticity** from the **Toolbox** and dropping it onto the row with your newly defined material (Figure 10). **Isotropic Elasticity** requires certain material parameters: **Young's Modulus** and **Poisson's Ratio.**

After you entering the appropriate parameters do not forget to push **Return to Project** button and to update the project

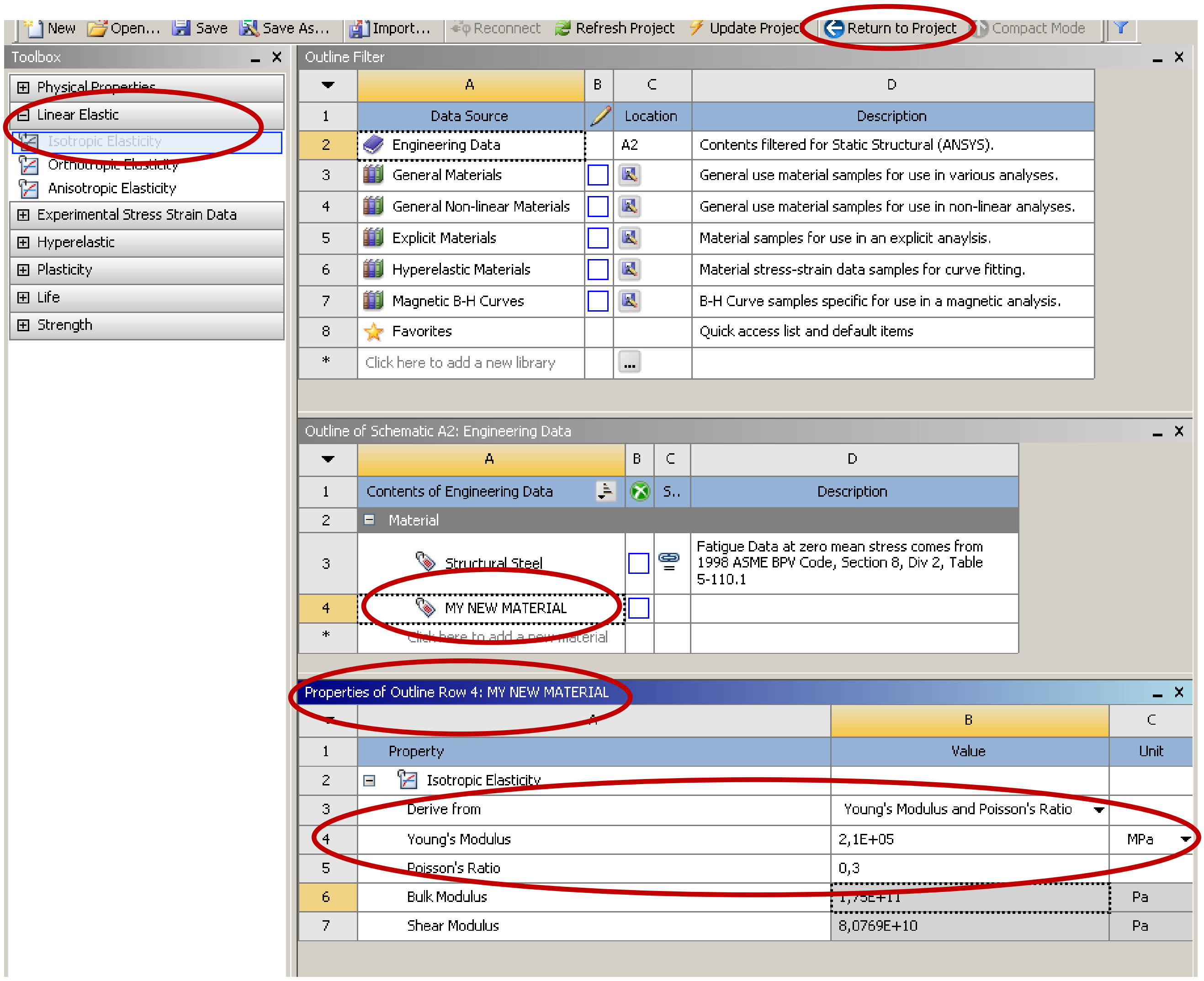


Figure 10: Define the material properties.

B.3 Meshing

To mesh the solid body select double click the **Model** sub-module in the Project schematic to open the **Mechanical** module (Figure 11).

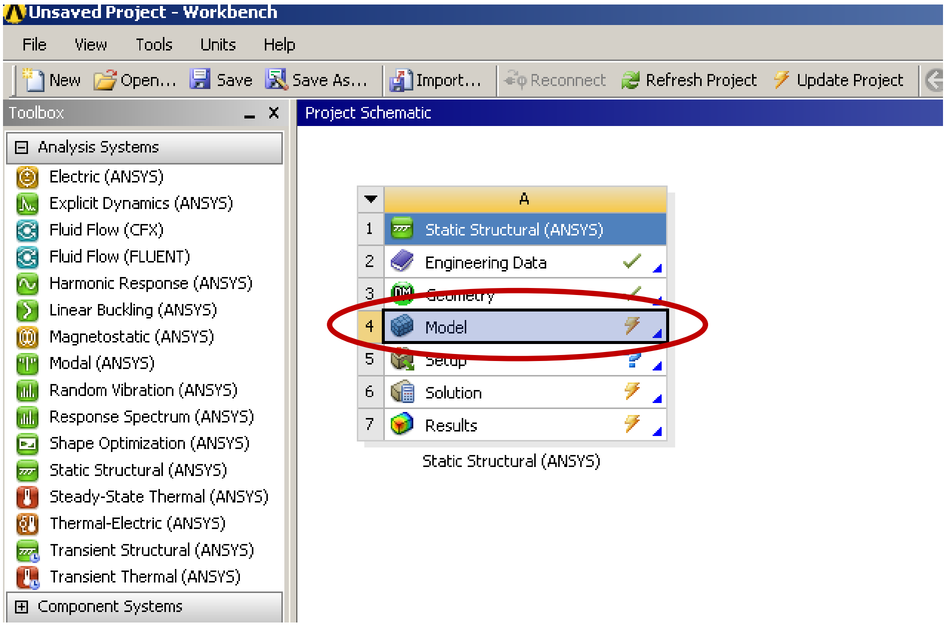


Figure 11: Starting the meshing and analysis module

Right click on **Mesh** in the **Structure Tree** and select a meshing method (Figure 12).

Mesh (right click) →Insert →Mapped Mesh

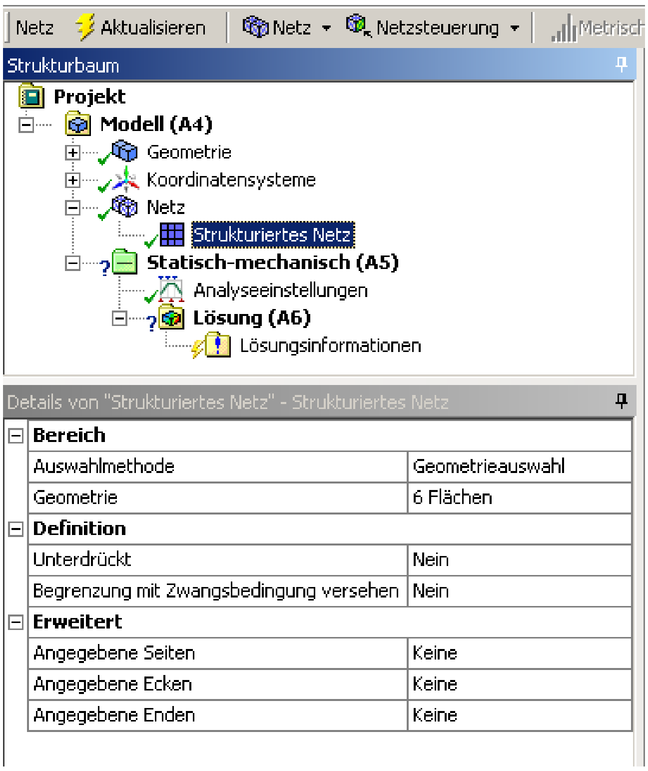


Figure 12: Setup meshing details

At this point focus on the **Details** box and select the geometry (all 6 faces) and do not forget to update the project; then click the **Generate** button.

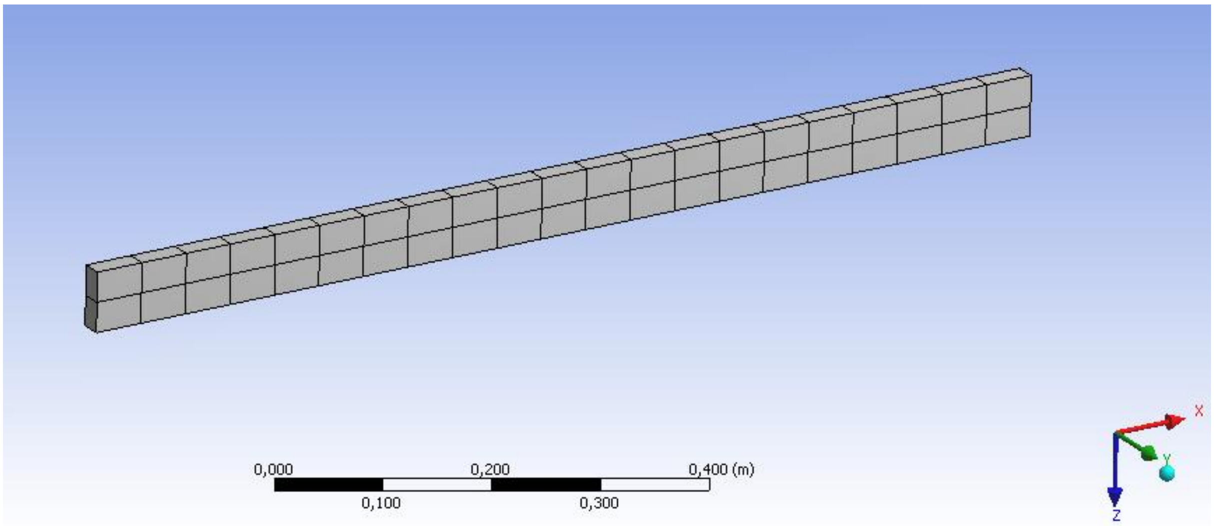


Figure 13: Meshed body

B.4 Applying Loads and Boundary Conditions

i. Boundary Conditions

To setup the necessary support:

Structure Tree → Static-Mechanic (right click) → Insert → Fixed Support

Select the Face you want to assign as a fixed support.

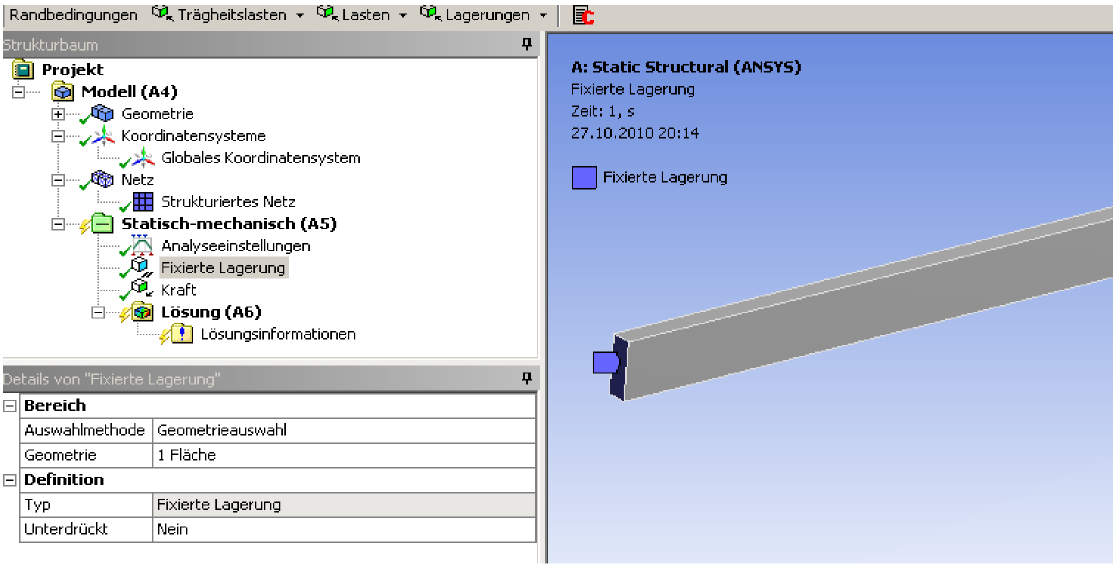


Figure 14: Fixed Face of the body

ii. Applying the Loads

Structure Tree → Static-Mechanic (right click) → Insert → Force

Select the Face you want to assign a force on this surface.

Assign the components of the force vector. Now the problem is ready to be solved.

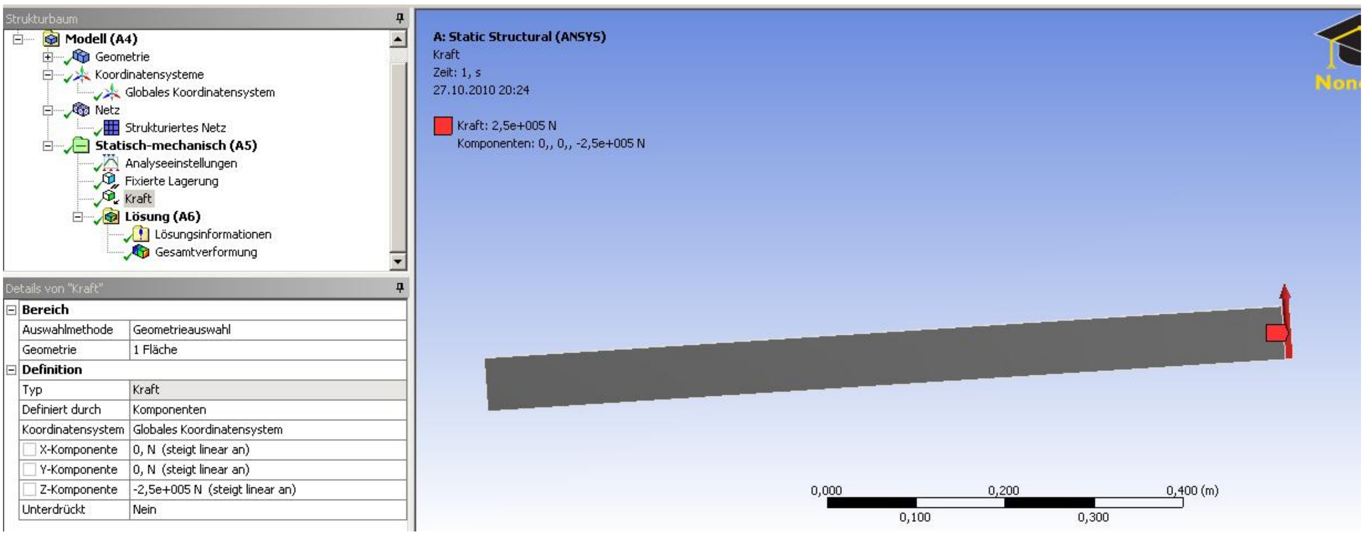


Figure 15: Applied force

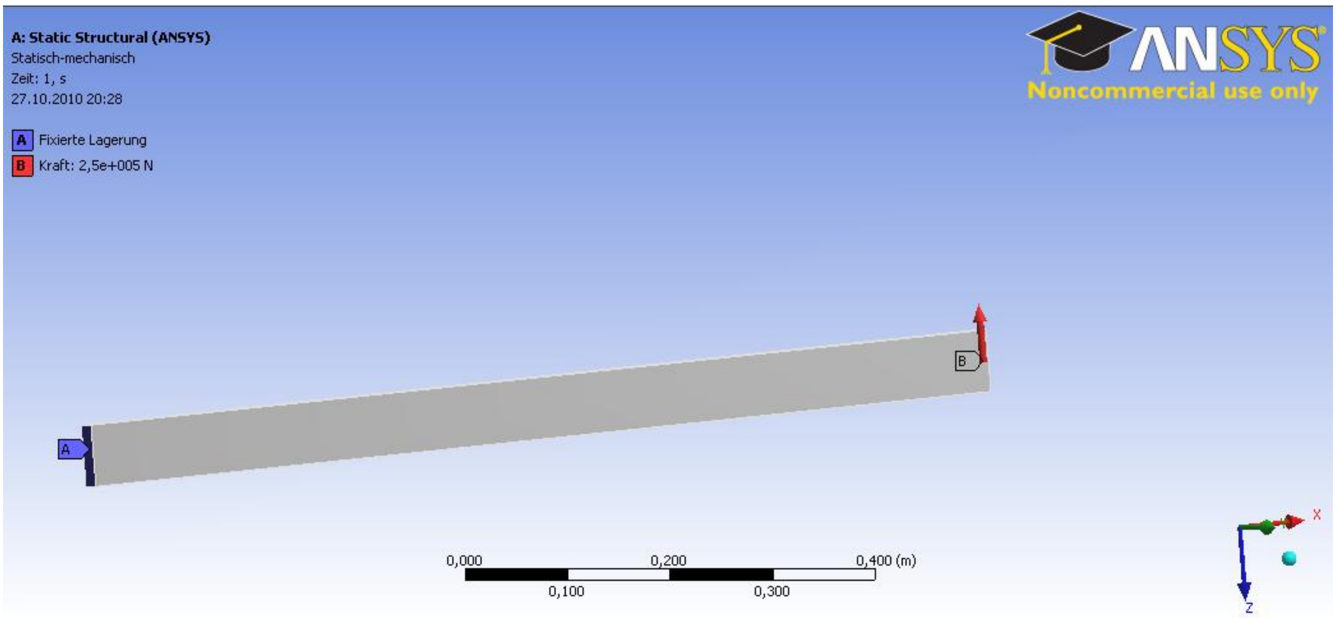


Figure 16: Fixed face of the body and applied force

A. Solving

To solve the problem add some solving tools to the **Solution** node which is part of the **Structure tree.**

Solution (right click)→ Total Deformation
Solution (right click)→ Total Strain
Solution (right click)→ Total Stress
Solution (right click)→ Von-Mises Equivalent Stress

Click **Solve**

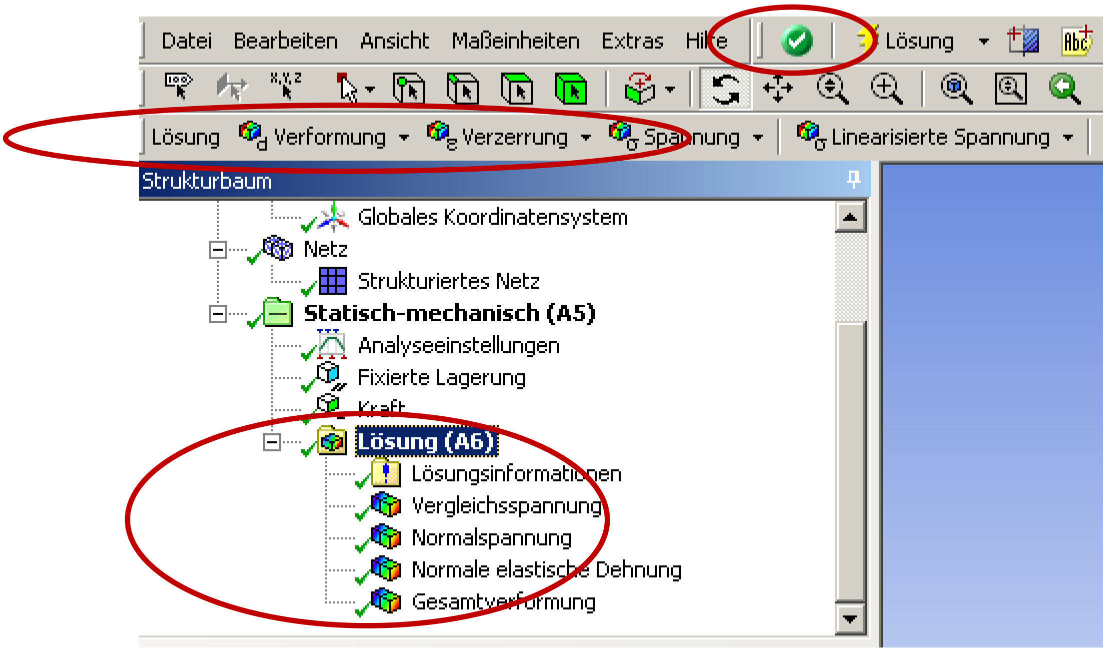


Figure 17: Required solving tools

a. Contour Plot of Deformed Shape

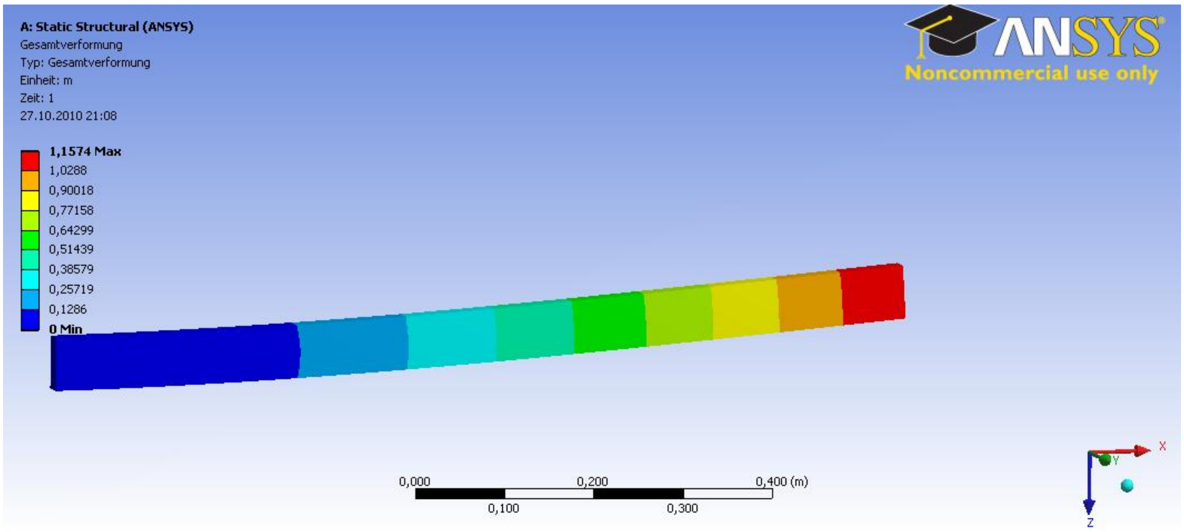


Figure 17: SCALED Displacement solution (z direction) of the beam (with too high force).

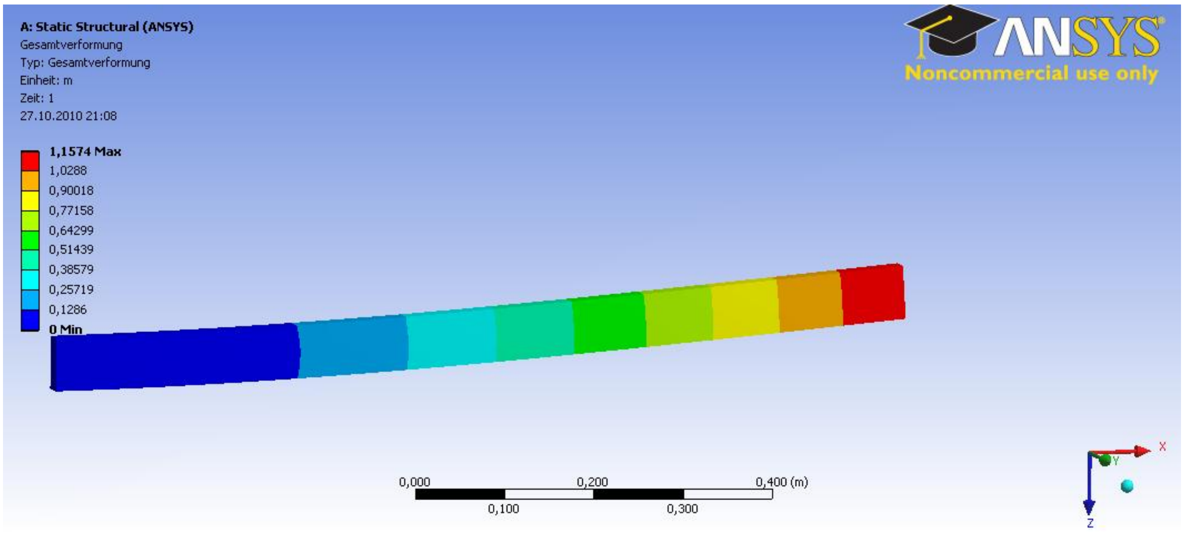


Figure 18: UNSCALED Displacement solution (z direction) of the beam (with too high force).

Correcting the Model:

The value of the applied force was erroneously *too high* and should be corrected from 500,000 to 5,000 Newton.

Structure tree → Force → Details

Change the value of the force 500,000 to 5,000 N and click to **Solve.**

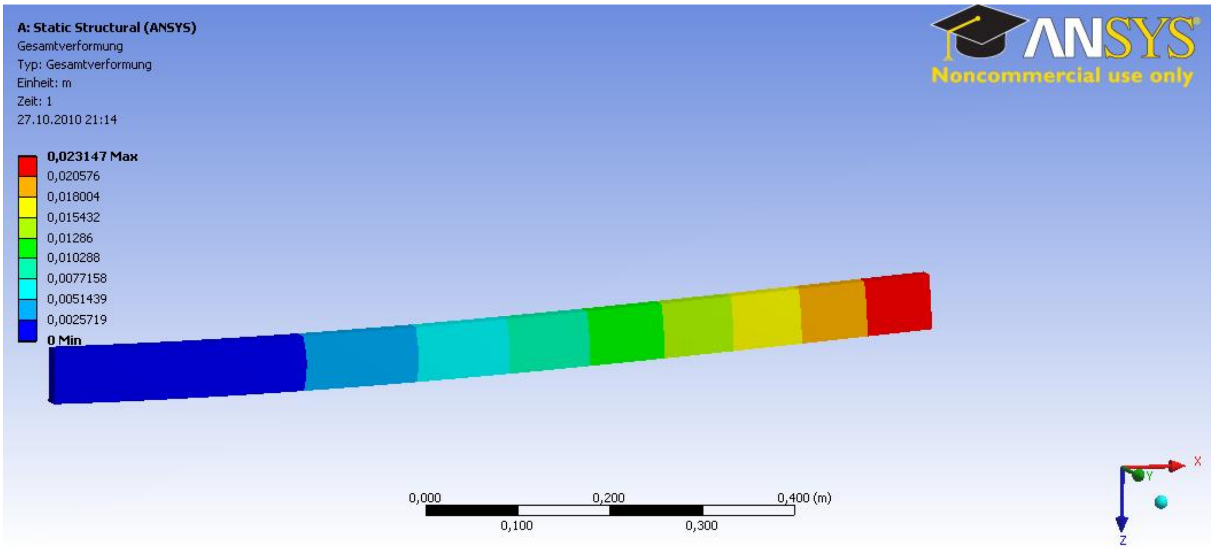


Figure 19: Deformed shape of the beam resulting from corrected force.

b. Contour Plot: X-Component of Total Strain

Structure Tree → Total Strain

and

Details → Direction → X-axis

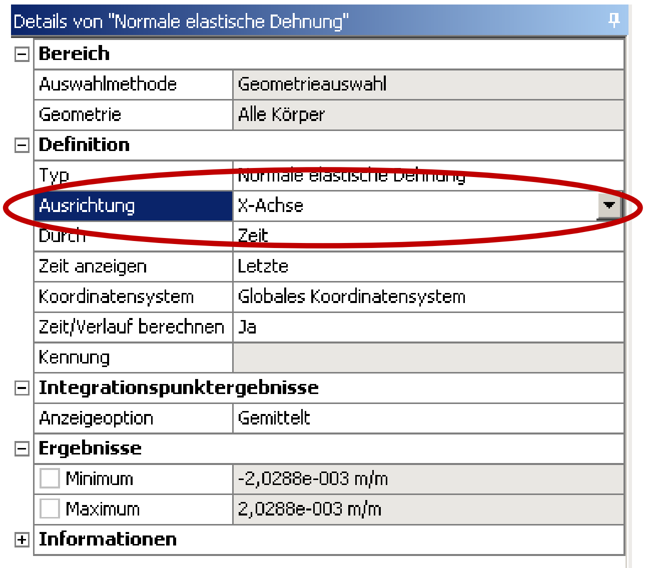


Figure 17: Details of total strain

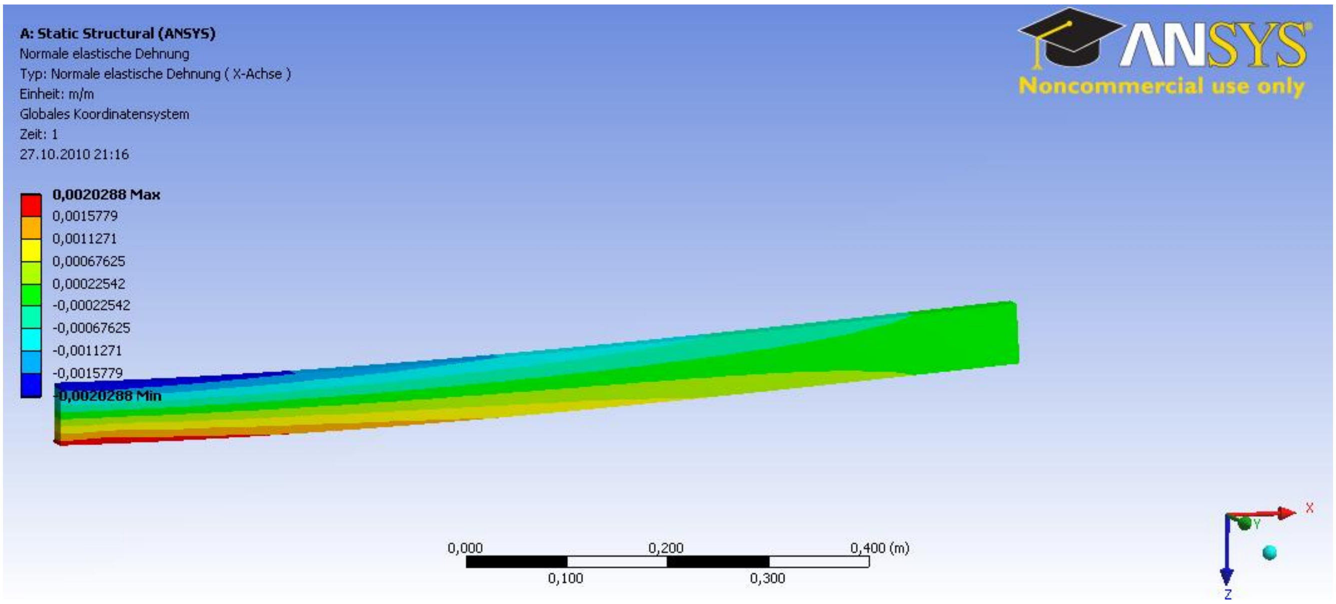


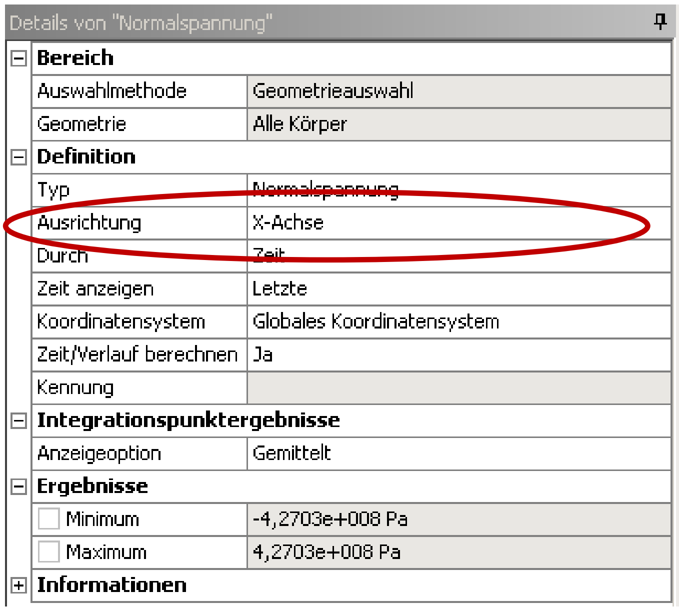
Figure 20: Contour plot of x-component of total strain.

c. Contour Plot X-Component of Stress

Structure Tree → *Total Stress*

and

Details → *Direction* → *X-axis*



Details von "Normalspannung"

Bereich	
Auswahlmethode	Geometrieauswahl
Geometrie	Alle Körper
Definition	
Typ	Normalspannung
Ausrichtung	X-Achse
Durch	Zeit
Zeit anzeigen	Letzte
Koordinatensystem	Globales Koordinatensystem
Zeit/Verlauf berechnen	Ja
Kennung	
Integrationspunktergebnisse	
Anzeigeoption	Gemittelt
Ergebnisse	
Minimum	-4,2703e+008 Pa
Maximum	4,2703e+008 Pa
Informationen	

Figure 21: Details of total stress

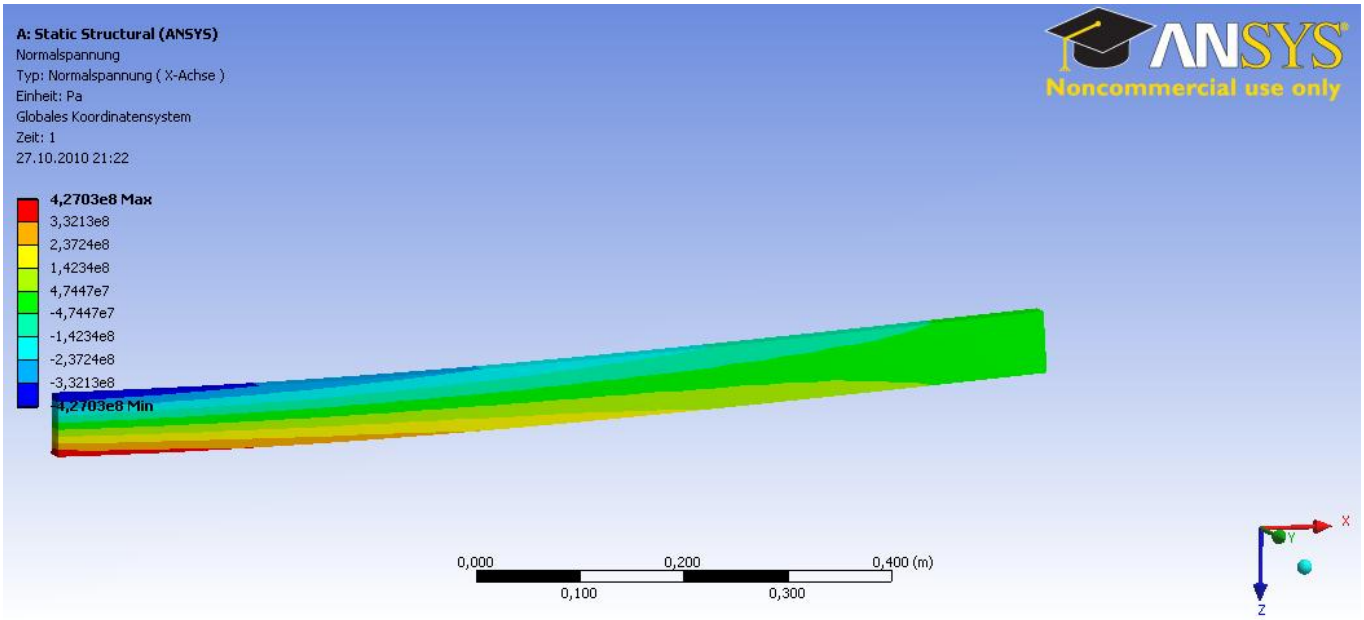


Figure 22: Contour plot of x-component of stress.

d. Contour Plot: Von-Mises Equivalent Stress

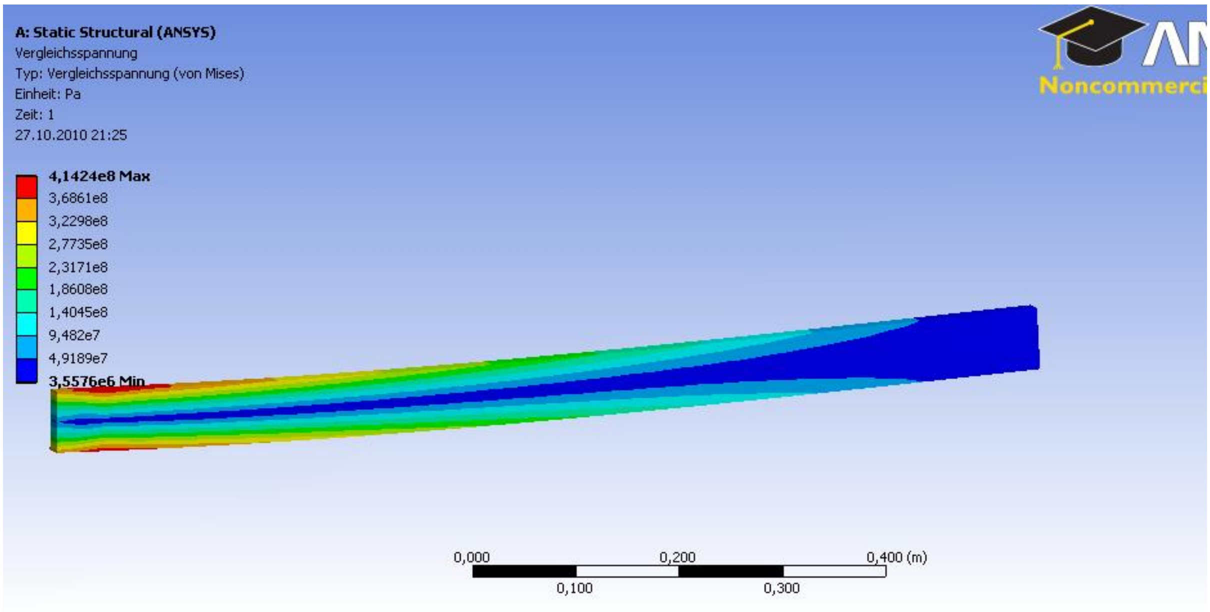


Figure 22: Contour plot of Von-Mises equivalent stress.

Answering the Questions:

1 Will the beam break and were would it start breaking?

With the corrected force (F = 5,000 N) the beam will not break. The maximal predicted von Mises stress reaches values of σ_{pred} = 206 N/mm², whereas the ultimate yield stress σ_{yield} = 235 N/mm² is higher. That means the failure criterion $\sigma_{\mathrm{pred}} > \sigma_{\mathrm{yield}}$ is not fulfilled. However, the difference between the two values is small. In many technical applications the factor of security should be 2.0 or even higher. The factor of security $\sigma_{\mathrm{yield}}/\sigma_{\mathrm{pred}}$ reached in our example is much smaller.

The critical region, were we would expect the beginning of a failure, is located at the left end of the half beam, Symbol MX) at the location of maximum stresses. For the full length beam the critical region would lay in the middle where the force was applied (Figure 1).

2 If not, what would be the maximum deflection *w?*

We predicted a maximum deflection of w = 11 mm appearing at the free end (right side, symbol MX) of the simplified half model. The full length beam under 3-point-bending (Figure 1) will show a maximum deflection of the same amount in the middle.