Exercise 1

3-Point Bending Using the Static Structural Module of Ansys Workbench 12

Contents

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 Fig. 1

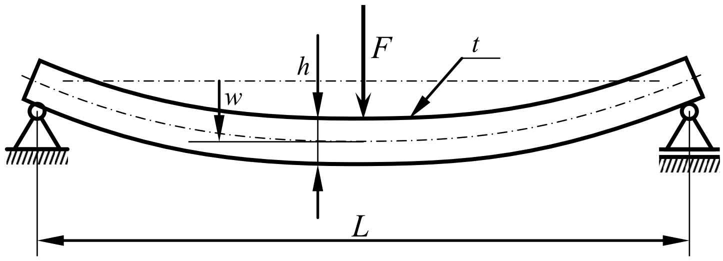


Fig. 1: Beam under 3-point bending.

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!

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

A. Getting started

A.1 Login to your local Linux PC

Enter your kiz account name and password to login to the local Linux PC.

A.2 Open a Terminal

Start a terminal (e.g. "Konsole") from the desktop main menu:

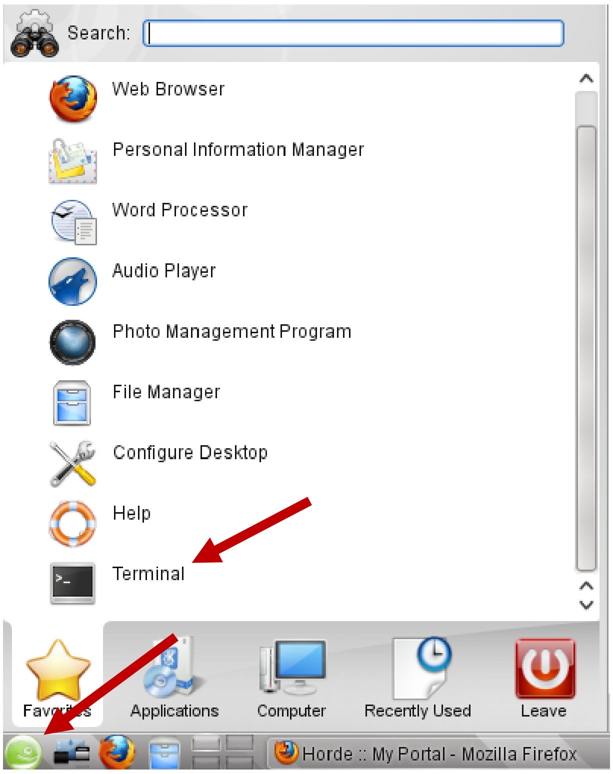


Figure 1: Open a terminal and a file manager from the "K menu".

A.3 Start a remote session

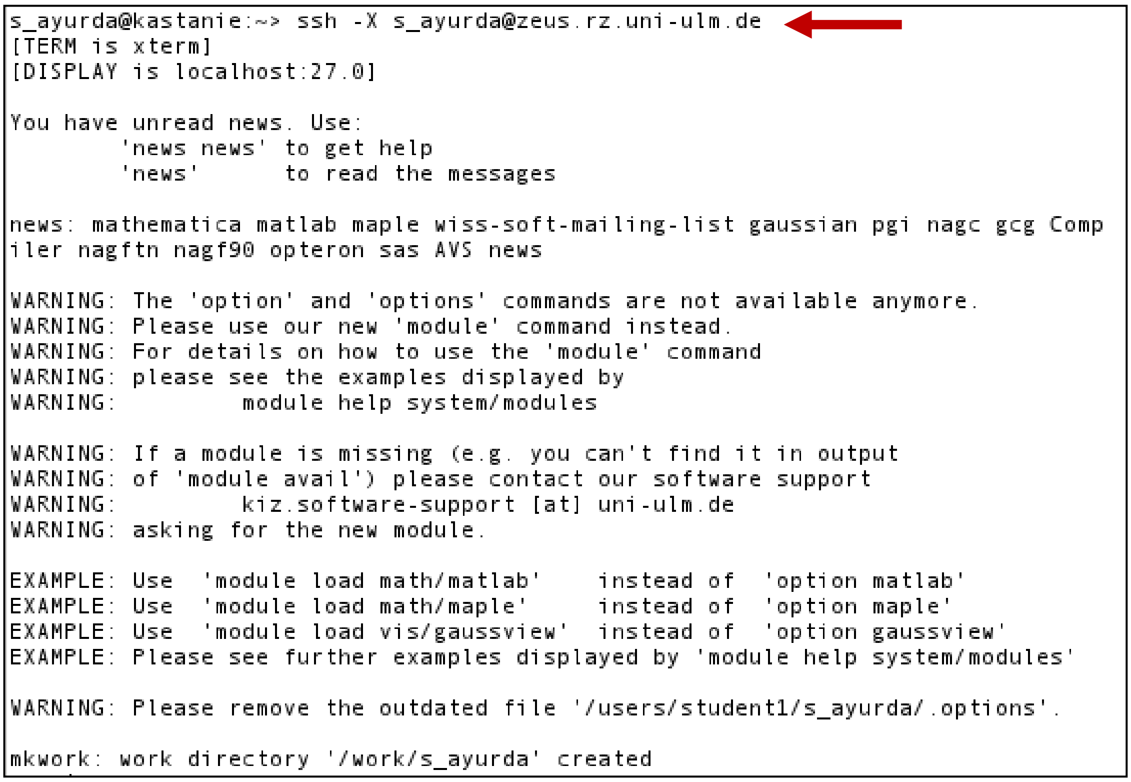
Start a remote session (SSH = secure shell) by entering the command

ssh <yourname>@zeus.rz.uni-ulm.de

or

ssh <yourname>@andromeda.rz.uni-ulm.de

in the terminal. Please use your account name as "**<yourname>**" and enter your pass-word when being asked. The password should be the same as the one you used to login locally to your Linux machine.



```
s_ayurda@kastanie:~> ssh -X s_ayurda@zeus.rz.uni-ulm.de
[TERM is xterm]
[DISPLAY is localhost:27.0]

You have unread news. Use:
        'news news' to get help
        'news'      to read the messages

news: mathematica matlab maple wiss-soft-mailing-list gaussian pgi nagc gcg Comp
iler nagftn nagf90 opteron sas AVS news

WARNING: The 'option' and 'options' commands are not available anymore.
WARNING: Please use our new 'module' command instead.
WARNING: For details on how to use the 'module' command
WARNING: please see the examples displayed by
WARNING:         module help system/modules

WARNING: If a module is missing (e.g. you can't find it in output
WARNING: of 'module avail') please contact our software support
WARNING:         kiz.software-support [at] uni-ulm.de
WARNING: asking for the new module.

EXAMPLE: Use  'module load math/matlab'    instead of  'option matlab'
EXAMPLE: Use  'module load math/maple'     instead of  'option maple'
EXAMPLE: Use  'module load vis/gaussview'  instead of  'option gaussview'
EXAMPLE: Please see further examples displayed by 'module help system/modules'

WARNING: Please remove the outdated file '/users/student1/s_ayurda/.options'.

mkwork: work directory '/work/s_ayurda' created
```

Figure 2: Remote login via ssh command.

For testing purposes, try to run a new **xterm** session on the remote host by issuing the command

xterm&

(*Note:* The ampersand detaches the new xterm process from the SSH console session so you can enter further commands without needing to close the xterm window.)
This is also a first check whether the display settings are appropriate to let the host computer (zeus/andromeda) export whole X11 windows (and not only text out-put) to the local screen.

A.4 Start the FE software ANSYS

Before being able to use a software package on the computer centers' Unix machines you have to choose an appropriate "**module**" command to set necessary environment va-riables. A list of all available software is displayed if you simply enter the command "**modules**" without any further parameters.

To start the ANSYS Workbench 12.0 (www.ansys.com) first enter

module load cae/ansys

and then

runwb2&

either into the xterm or directly into your SSH terminal.

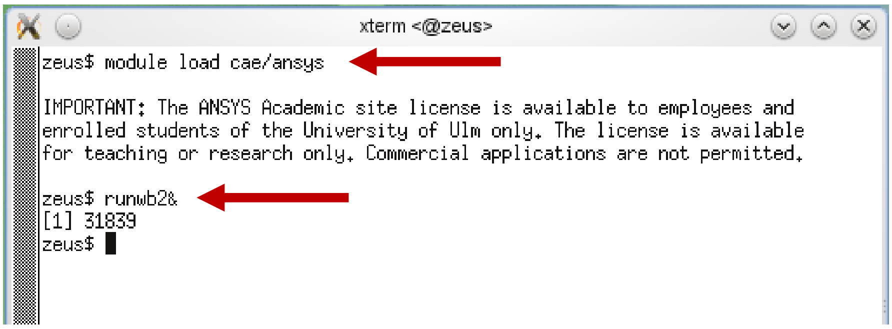


Figure 3: Remote login via xterm& command

Important: Please make sure to choose the right license type
After you have launched Workbench go **Tools → License Preferences** and change license type by using **Move up and Move down** buttons, finally **Apply** the settings.

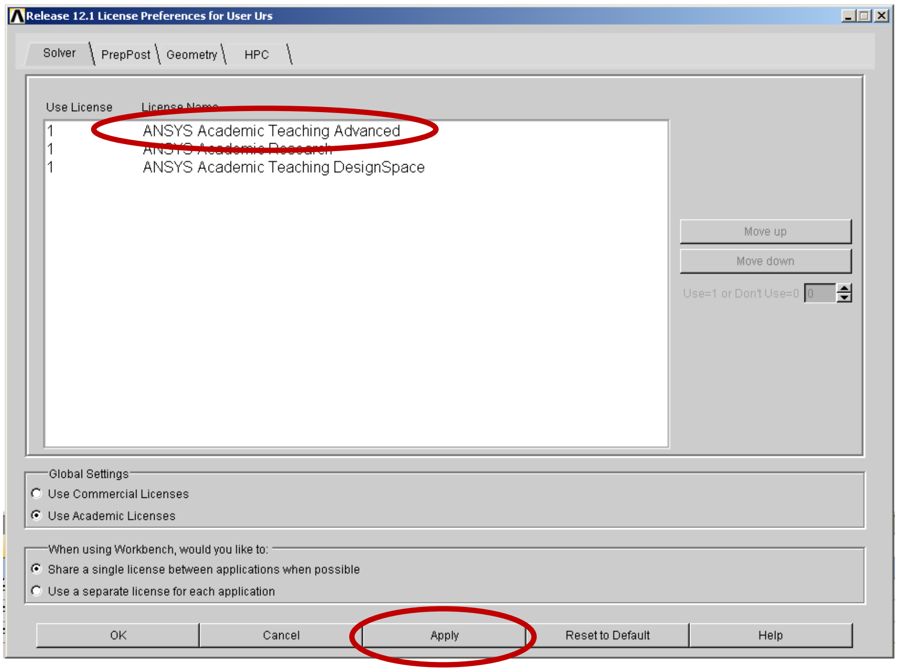


Figure 4: Choosing the right license type

"ANSYS Academic Teaching Introductory" or **"ANSYS Academic Teaching Advanced"**
Do **not** choose "ANSYS Academic Research"!

B. Preprocessor (Setting up the Model)

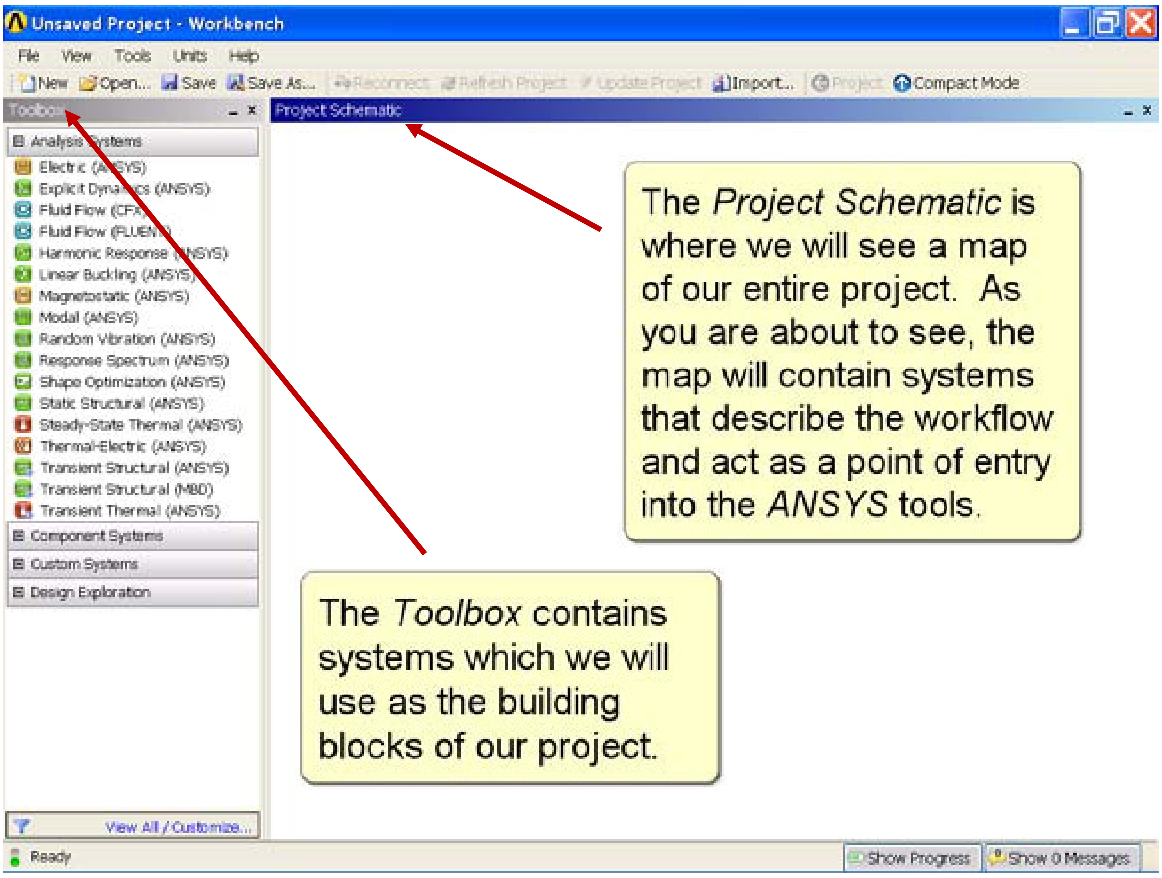


Figure 5: Interface of Ansys Workbench 12

B.1 Build the Geometry Using DesignModeler Module

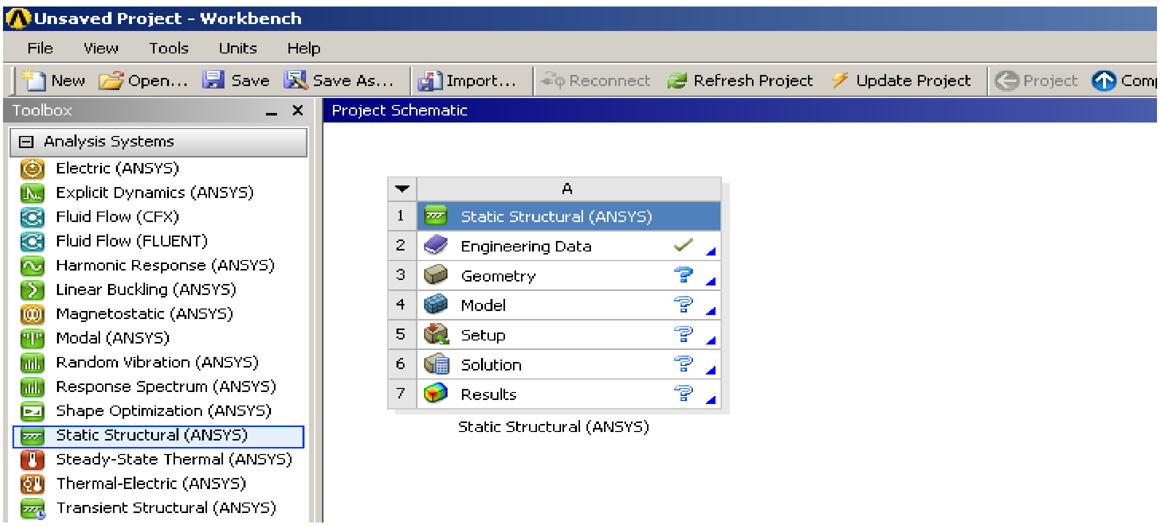


Figure 6: Getting starting with Static Structural module

To begin; drag the **Static Structural Module** and drop into the **Project Schematic** and double click to **Geometry** to open **DesignModeler**.

Choose the desired units.

To create the solid beam; **Create →Basic Elements → Cube** and give the desired dimensions at the **Details** box.

To solve the problem correctly; please be sure that the origin of the coordinate system located on the plane and at the center of the cross section of the beam. The beam axis must be oriented along the global x-axis. Ansys Workbench saves this model automatically; you can close this module and go on.

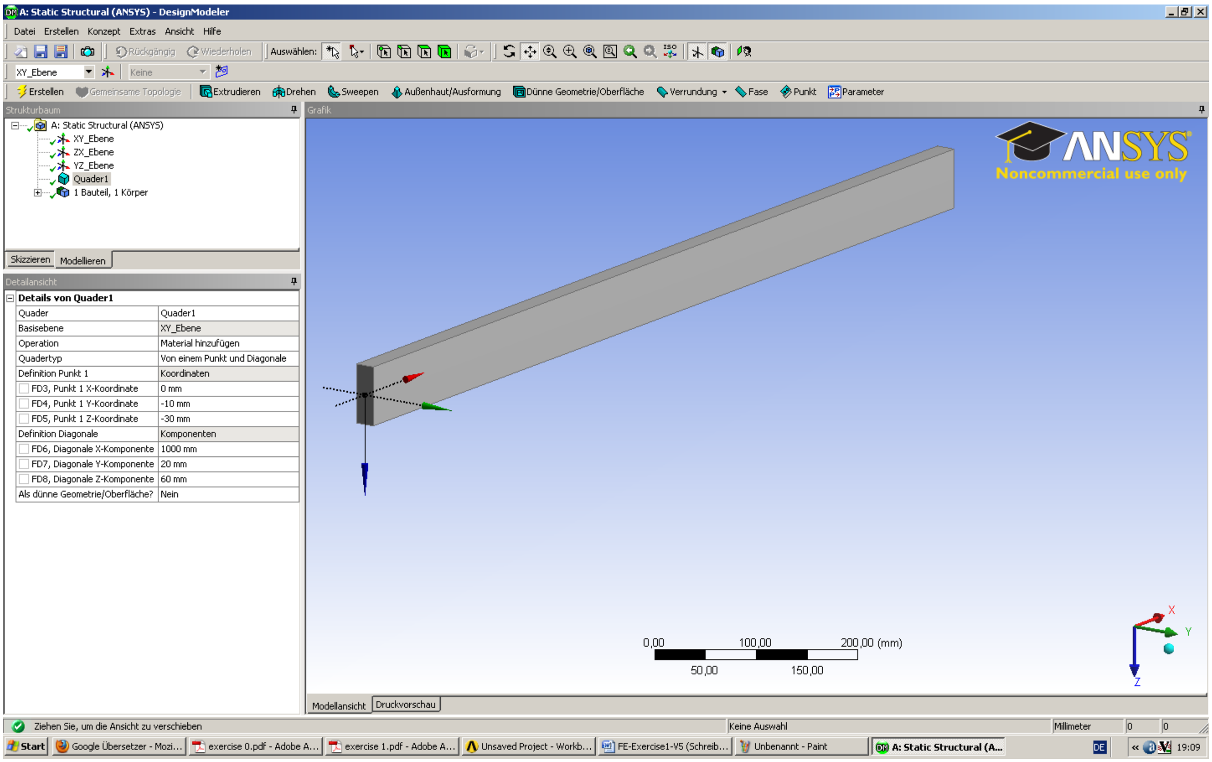


Figure 7: The beam on the DesignModeler

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.

Static Structural → Engineering Data → Edit... → Outline of Schematic A:2 → Add new material

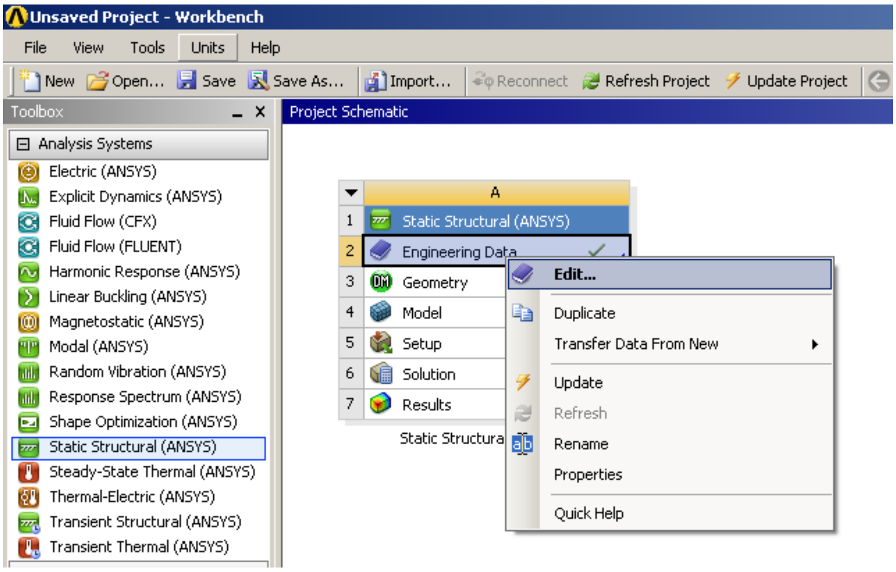


Figure 8: Assignment of the engineering data

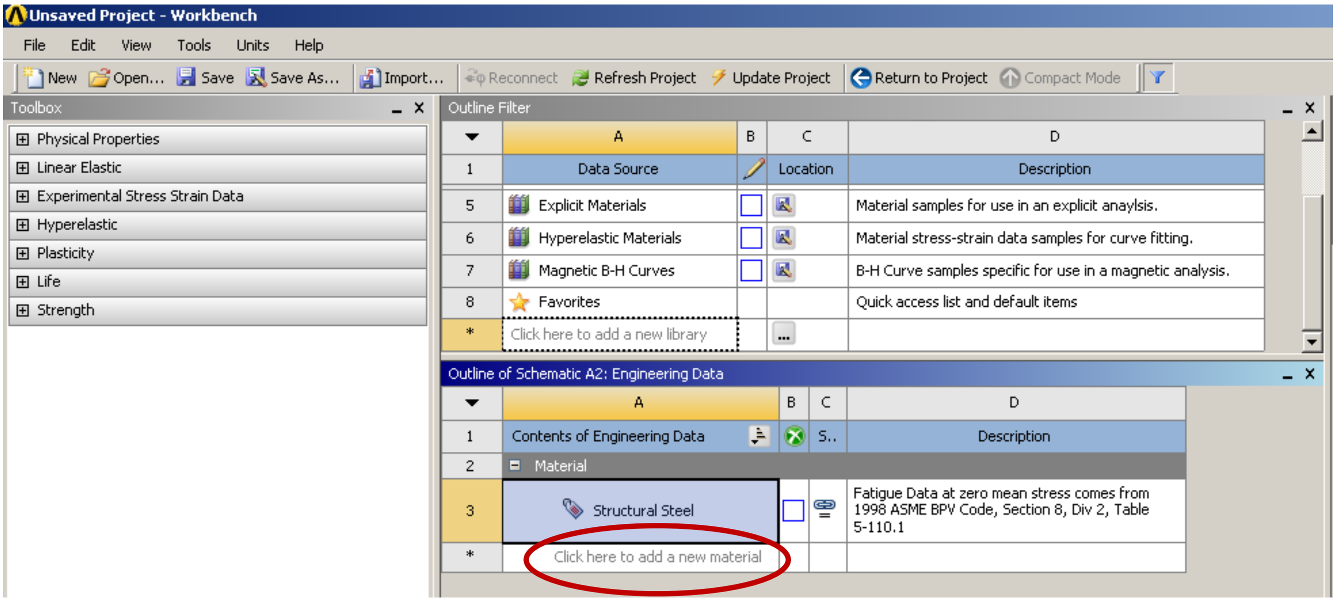


Figure 9: Naming the new material

Choose the simplest available material model by dragging the property from the **Toolbox** and dropping on to **Properties of Outline Row 4: MY NEW MATERIAL**

Toolbox → Linear Elastic → Isotropic Elasticity

Now **Properties of Outline Row 4: MY NEW MATERIAL** requires certain material parameters: **Young´s Modulus** and **Poison Ratio.**

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

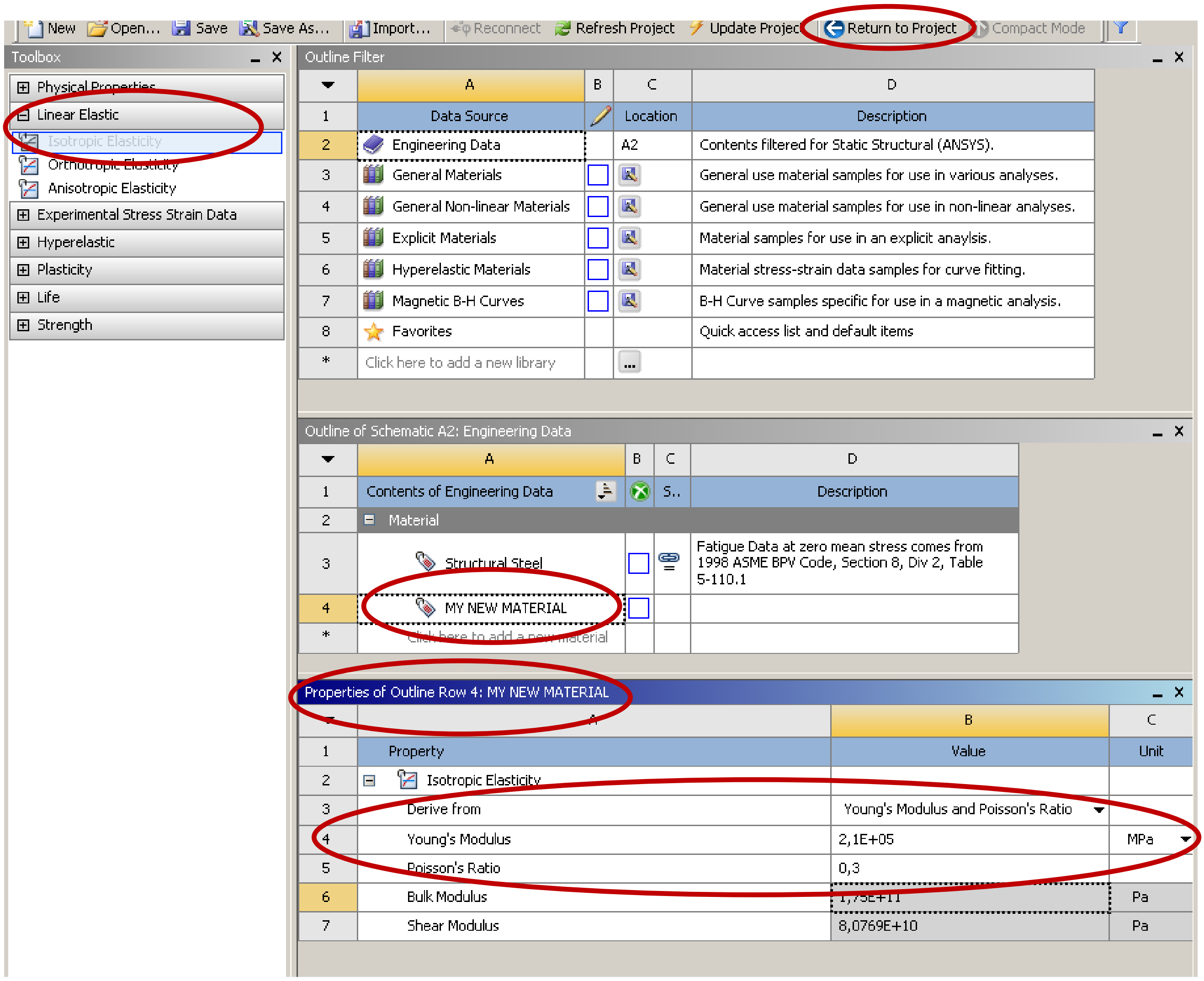


Figure 10: Define the material properties.

B.3 Meshing

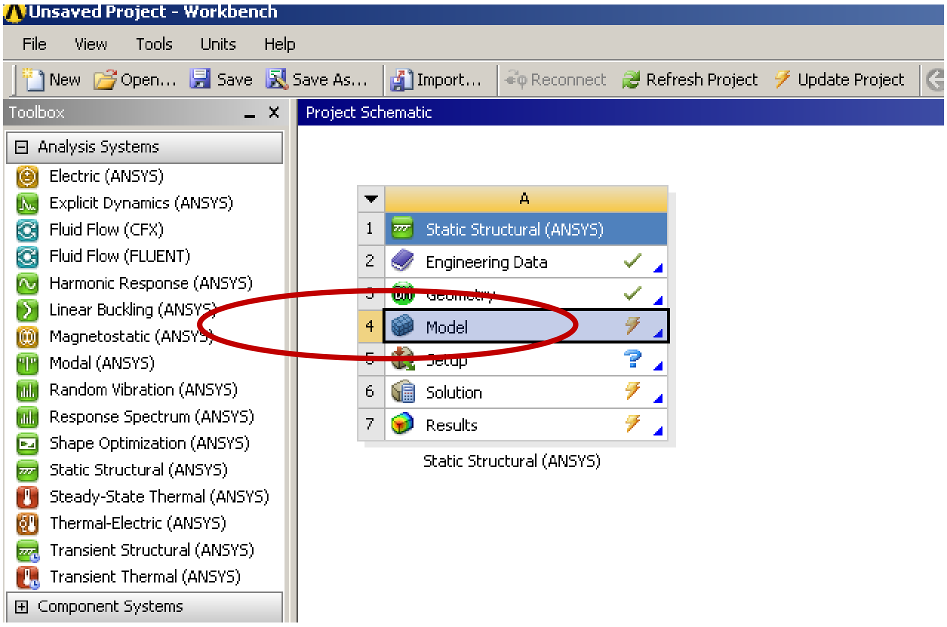


Figure 11: Getting starting to mesh the geometry

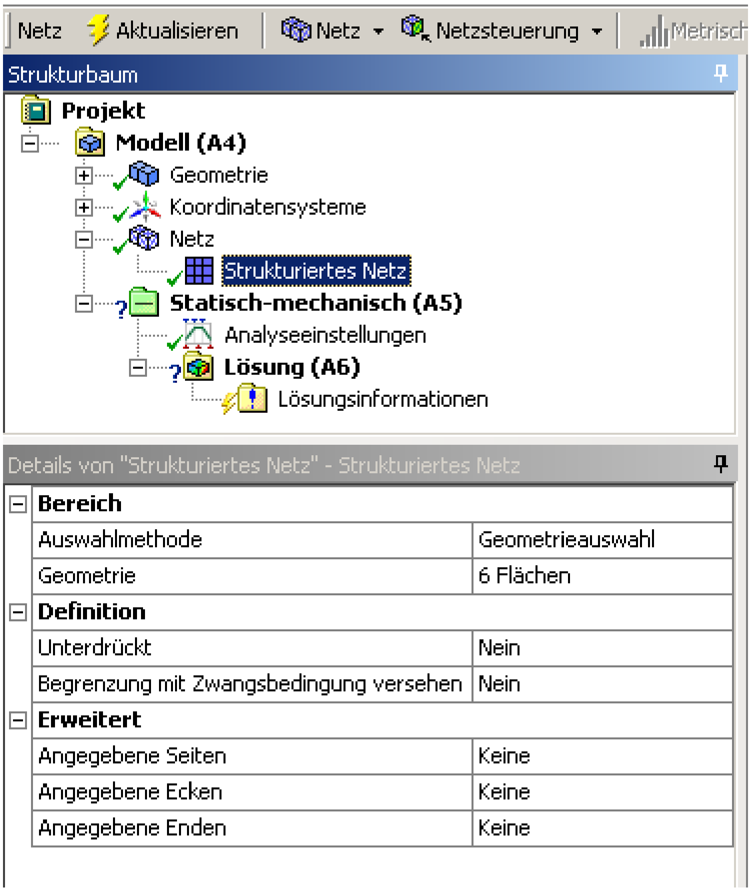


Figure 12: Meshing details

To mesh the solid body select (double click) the Model to open Mechanical Module. Right click on **Mesh** which is on the **Structure Tree** and select a meshing method.

Mesh (right click) →Insert →Mapped Mesh

At this point focus on to **Details**. **Select the Geometry** (all 6 faces) and do not forget to update the project, please click to **Generate** button.

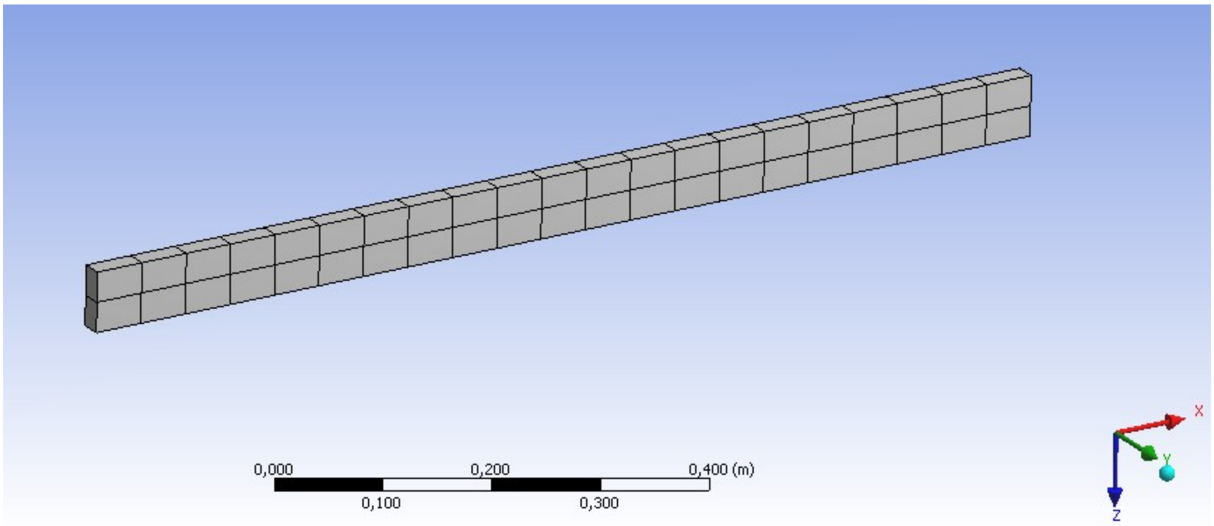


Figure 13: Meshed Body

B.4 Applying Loads and Boundary Conditions

i. Boundary Conditions

To setup the necessary support:

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

At this point focus on to **Details. Select the Face** you want to assign as a fixed support.

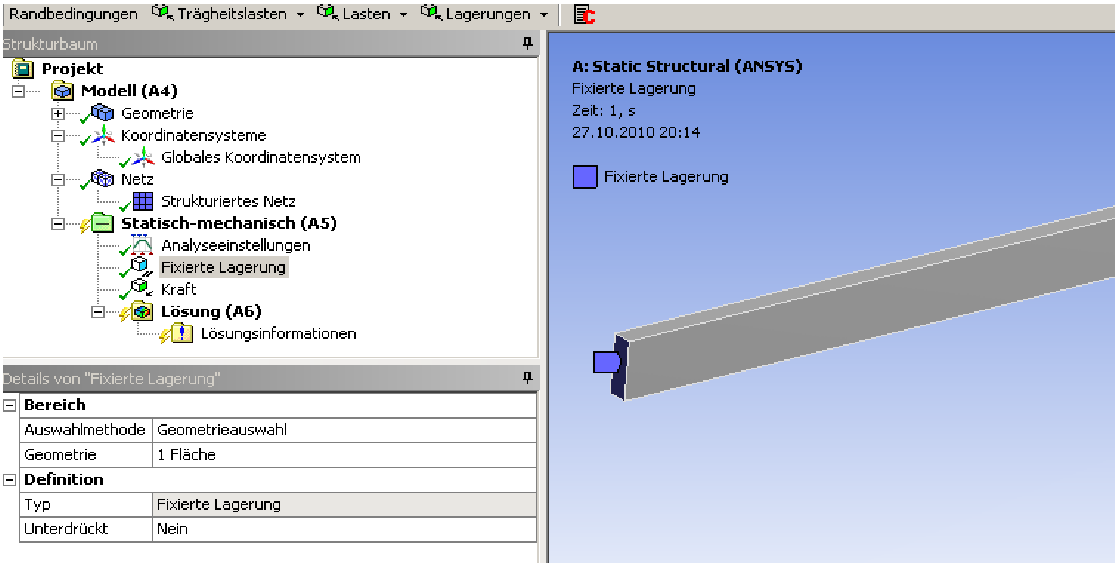


Figure 14: Fixed Face of the body

ii. Applying the Loads

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

At this point focus on to **Details**. **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.

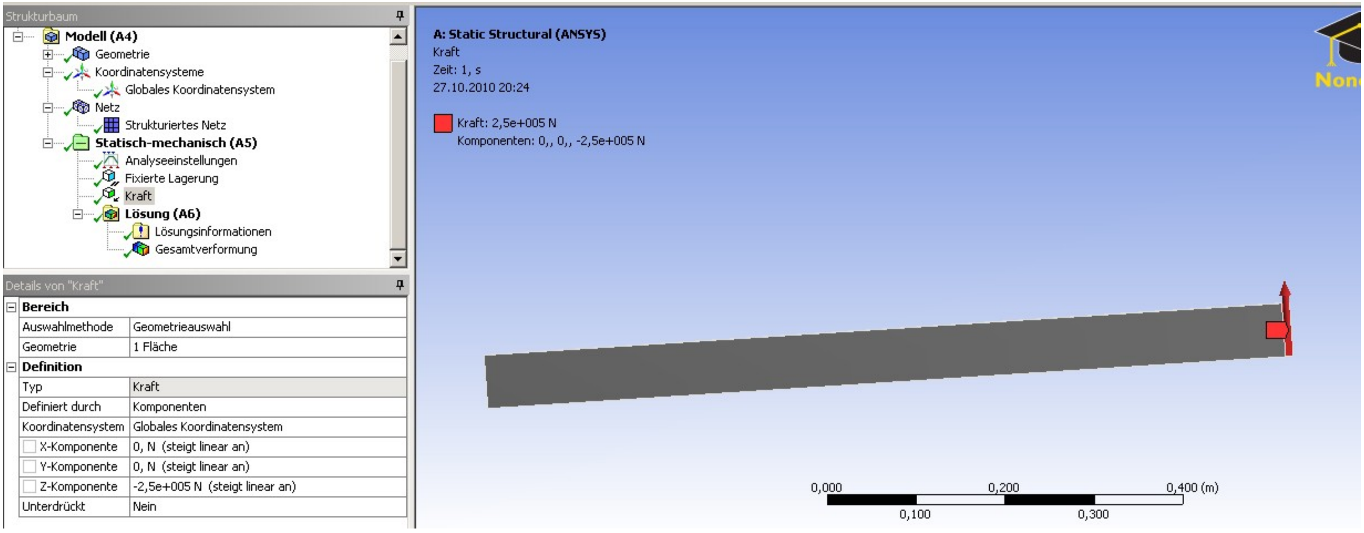


Figure 15: Applied force

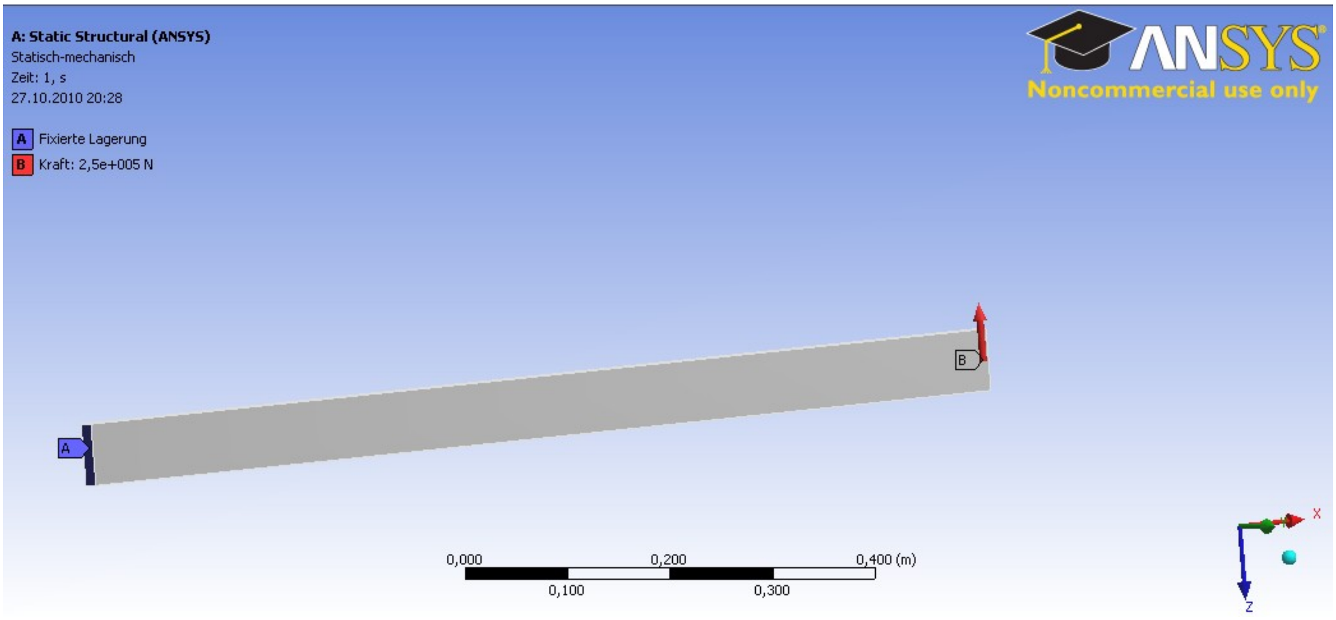


Figure 16: Fixed face of the body and applied force

C. Solving

To solve the problem add some solving tools at the **Solution** which is at 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**

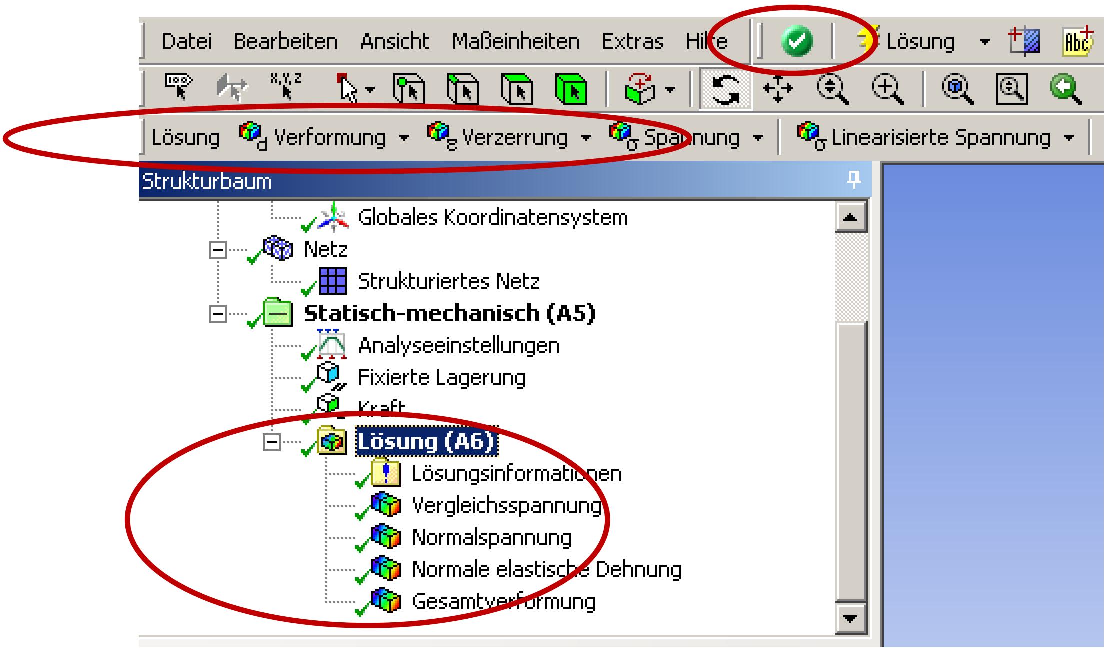


Figure 17: Required solving tools

a. Contour Plot of Deformed Shape

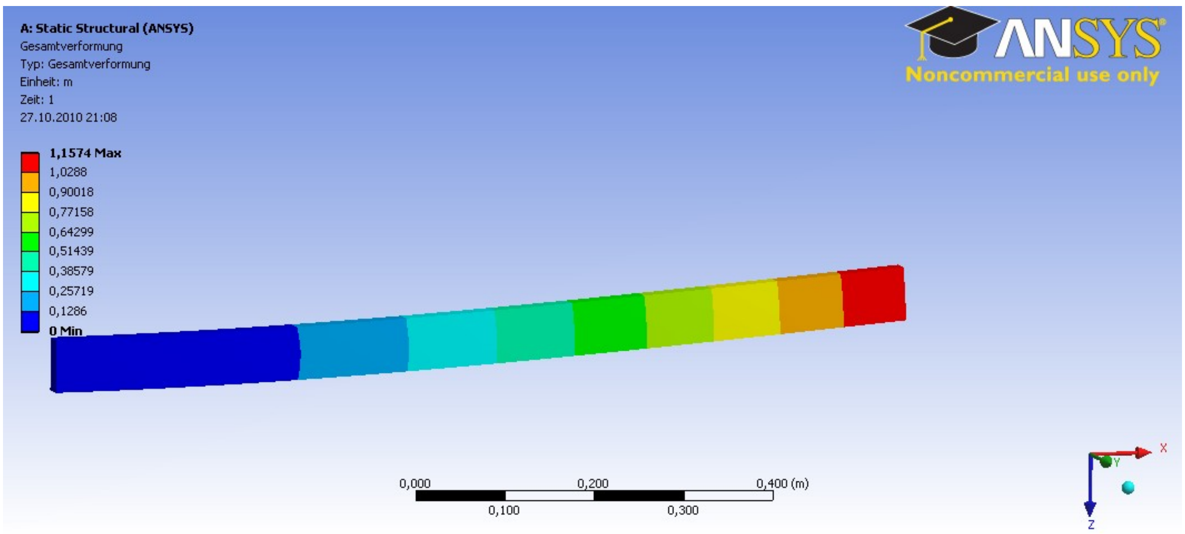


Figure 17: Total deformation

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.**

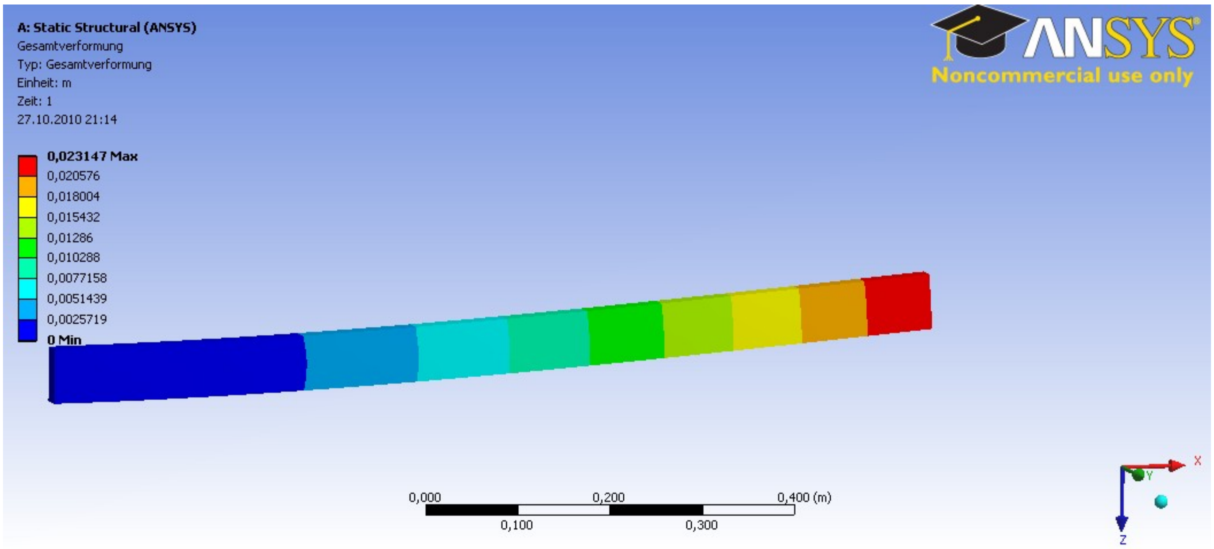
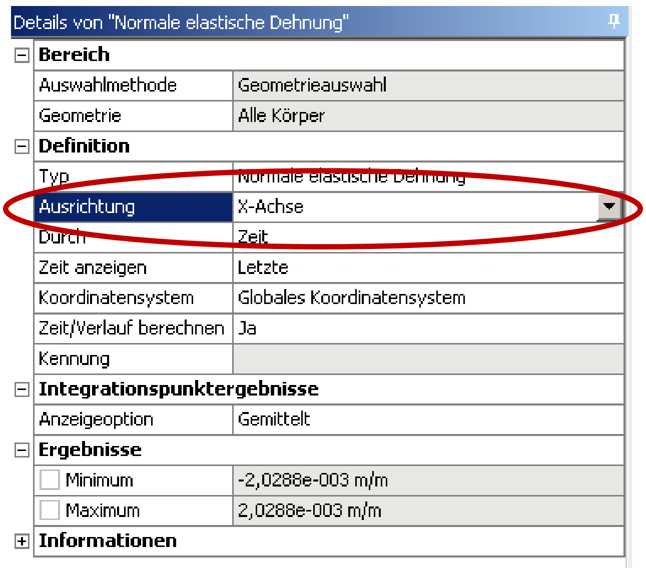


Figure 17: Deformed shape of the beam resulting from corrected force.

b. Contour Plot: X-Component of Total Strain

Structure Tree → Total Strain
and
Details → Direction → X-axis



Details von "Normale elastische Dehnung"

Bereich	
Auswahlmethode	Geometrieauswahl
Geometrie	Alle Körper
Definition	
Typ	Normale elastische Dehnung
Ausrichtung	X-Achse
Durch	Zeit
Zeit anzeigen	Letzte
Koordinatensystem	Globales Koordinatensystem
Zeit/Verlauf berechnen	Ja
Kennung	
Integrationspunktergebnisse	
Anzeigeoption	Gemittelt
Ergebnisse	
Minimum	-2,0288e-003 m/m
Maximum	2,0288e-003 m/m
Informationen	

Figure 17: Details of total strain

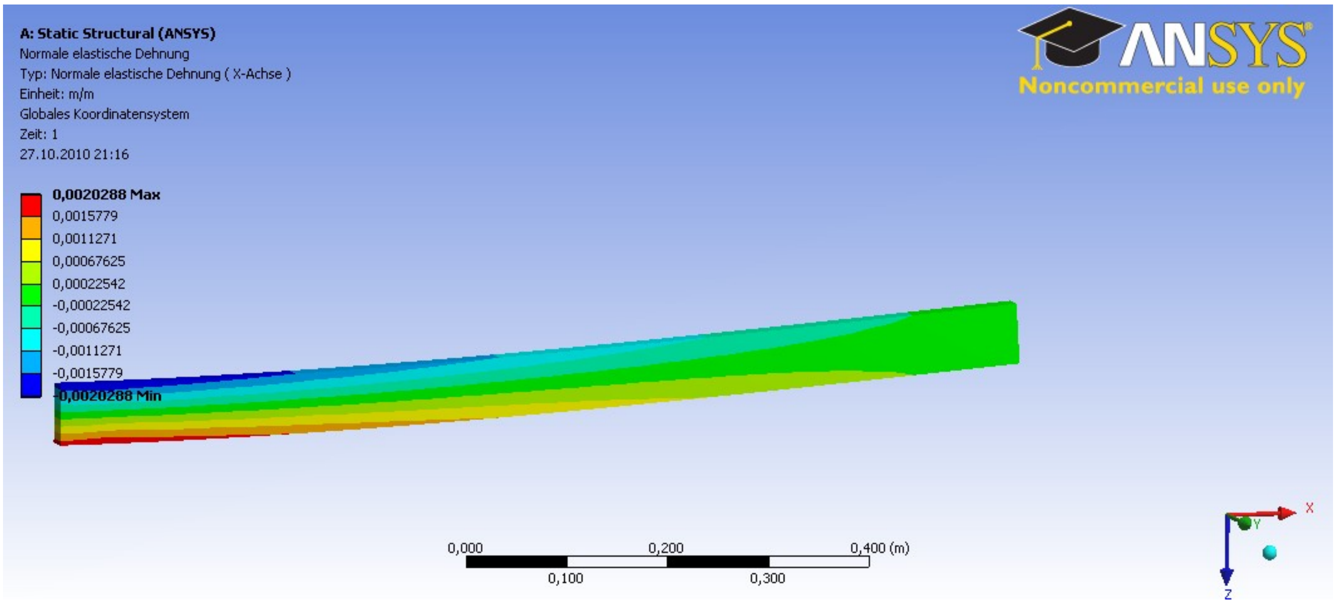


Figure 18: Contour plot of x-component of total strain.

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 18: Details of total stress

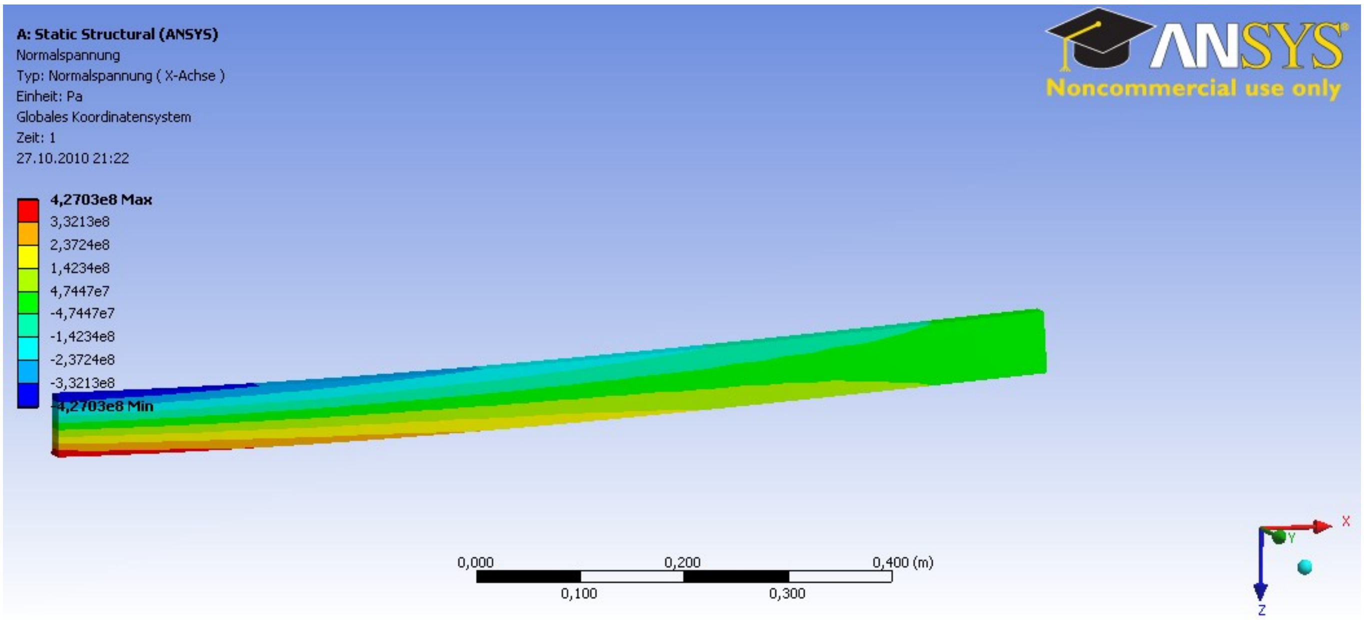


Figure 19: Contour plot of x-component of stress.

d. Contour Plot: Von-Mises Equivalent Stress

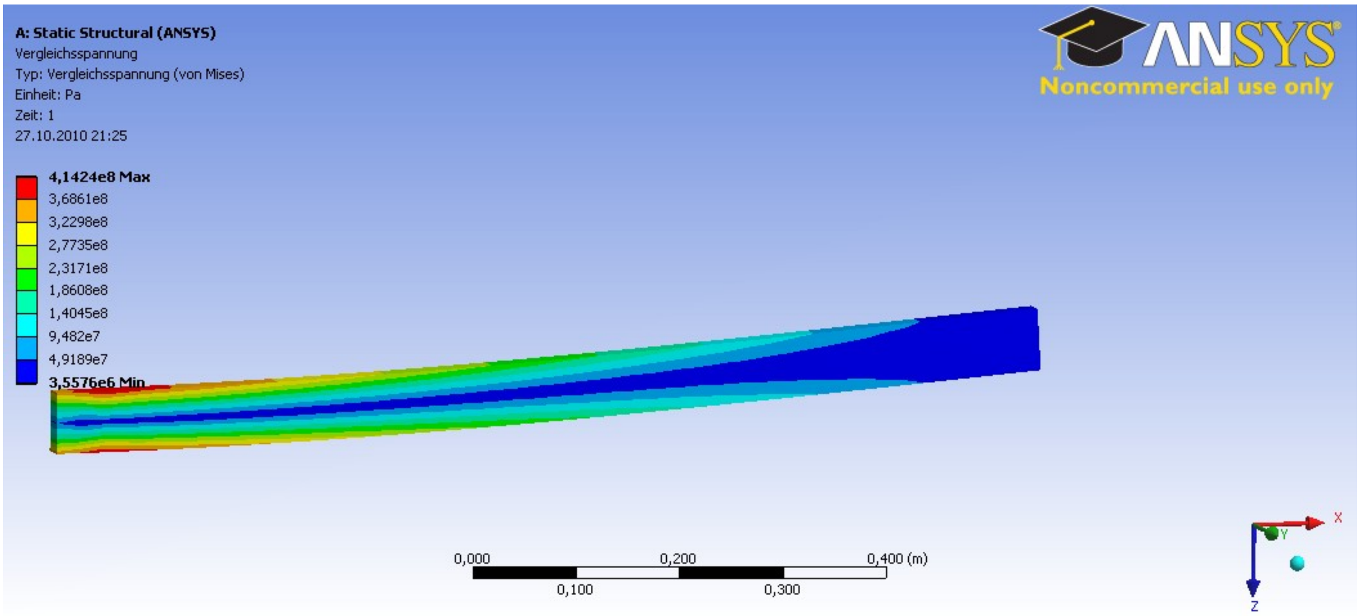


Figure 20: Contour plot of Von-Mises equivalent stress.

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 (**Fehler! Verweisquelle konnte nicht gefunden werden.**), whereas the ultimate yield stress σ_{yield} = 235 N/mm² is higher. That means the failure criterion $\sigma_{\text{pred}} > \sigma_{\text{yield}}$ 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 $\sigma_{yield}/\sigma_{pred}$ 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 (**Fehler! Verweisquelle konnte nicht gefunden werden.**, 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 (Fig. 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 (**Fehler! Verweisquelle konnte nicht gefunden werden.**). The full length beam under 3-point-bending (Fig. 1) will show a maximum deflection of the same amount in the middle.