Exercise 1

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

Contents

Learn how to	1
Given	2
Questions	2
Taking advantage of symmetries	2
B. Preprocessor (Setting up the Model)	6
B.1 Build the Geometry Using DesignModeler Module	6
B.2 Material Properties	7
i. Define Material Properties	7
B.3 Meshing	10
B.4 Applying Loads and Boundary Conditions	11
i. Boundary Conditions	11
ii. Applying the Loads	12
C. Solving	13
a. Contour Plot of Deformed Shape	13
b. Contour Plot: X-Component of Total Strain	14
c. Contour Plot X-Component of Stress	15
d. Contour Plot: Von-Mises Equivalent Stress	16
Answering the Questions:	16

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
$\sigma_{ m yield}$	= 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
                    to read the messages
        'news'
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'
             'module load math/maple'
                                                       'option maple'
EXAMPLE: Use
                                           instead of
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.

🔀 💿 xterm <@zeus>	\odot	\odot	\otimes
zeus\$ module load cae/ansys			
IMPORTANT: The ANSYS Academic site license is available to employees enrolled students of the University of Ulm only. The license is avail for teaching or research only. Commercial applications are not permit	and able ted.		
zeus\$ runwb2&			
zeus\$			

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.

Use License	License Name			
1 <	ANSYS Aca	ademic Teaching Adv	anced	
1	ANSYS Aca	ademic Heesearch ademic Teaching Des	ignSpace	
				Move up
				Move down
				Use=1 or Don't Use=0 0
				·
Global Settings	s			
Jse Commerc	ial Licenses			
Use Academic	Licenses			
When using W	/orkbench, would y	ou like to:		
Phore o cingle	license between a	applications when possible		

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** \rightarrow **Basic Elements** \rightarrow **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 linearelastic, isotropic material model.

Static Structural \rightarrow Engineering Data \rightarrow Edit... \rightarrow Outline of Schematic A:2 \rightarrow 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* \rightarrow *Linear Elastic* \rightarrow *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

🔡 New 🚰 Open 🛃 Save 🔣 Sav	e As 🧯	🚺 Import	💐 🖗 Reconnect 🛛 😂	Refre:	sh Proj	iect	🏏 Update Projec 📢	😋 Return to Project 🕥 Cor	npact Mode	Y
Toolbox _ X	Outline F	=ilter								_ ×
Physical Properties	-		A	В	C			D		
🖻 Linear Elastic	1	C	ata Source	1	Loca	tion		Description		
Isotropic Elasticity	2	🥏 Engineering Data			A2		Contents filtered for	Static Structural (ANSYS).		
Y Orthouropic Electicity	3	🏥 Genera	l Materials				General use material	l samples for use in various an	alyses.	
Experimental Stress Strain Data	4	🎒 Genera	l Non-linear Materials	;			General use material	I samples for use in non-linear	analyses.	
Hyperelastic	5	🎒 Explicit	Materials				Material samples for	use in an explicit anaylsis.		
Plasticity	6	🎒 Hypere	lastic Materials				Material stress-strain	n data samples for curve fittin	g,	
🕀 Life	7	🎒 Magnet	tic B-H Curves				B-H Curve samples s	pecific for use in a magnetic a	nalysis.	
	8	👷 Favorit	es		_		Quick access list and	default items		
	*	Click here to	o add a new library							
							1			
	Outline	of Schematic	02: Engineering Data		-	-				×
			A		в	с		D]	
	1	Contents o	f Engineering Data	È		5	D	escription		
	2	Materia		-		511	5	osciption (
	3	\$	Structural Steel			æ	Fatigue Data at zero 1998 ASME BPV Code	mean stress comes from e, Section 8, Div 2, Table		
		0					5-110.1		-	
	*		here to add a pew	norial					-	
		Circ		acentar					J	
	Properti	es of Outline	Row 4: MY NEW MAT	TERIAL						_ ×
				-				В		с
	1	Propert	у					Value		Unit
	2	🖃 🎽 Is	otropic Elasticity							
	3	Deriv	e from					Young's Modulus and Poisso	on's Ratio 🛛 👻	
	4	Youn	ıg's Modulus					2,1E+05		MPa 🔻
	5	Poiss	on's Ratio					0,3		
	6	Bulk	Modulus					1,75E+11		Pa
	7	Shea	r Modulus					8,0769E+10		Pa

Figure 10: Define the material properties.

B.3 Meshing

NUnsaved Project - Workbench						
File View Tools Units Hel)					
💾 New 💕 Open 🛃 Save 🔣	Save As	👔 Import	<i>≩</i> φ Reconnect	建 Refresh Project	🕖 Update Project	e
Toolbox _ X	Project Sch	nematic				
Analysis Systems						
🙆 Electric (ANSYS)						
🔝 Explicit Dynamics (ANSYS)	-		A			
😋 Fluid Flow (CFX)	1	🚾 Static Stru	ictural (ANSYS)			
😋 Fluid Flow (FLUENT)	2	🥏 Engineerir	ng Data	× .		
Marmonic Response (ANSYS)	3	Geometa -		<u> </u>		
Linear Buckling (ANSYS)	4	Madal		4		
🔘 Magnetostatic (ANSYS)	1	Model				
🕎 Modal (ANSYS)	-	Secup		24		
Random Vibration (ANSYS)	6	💼 Solution		7 🖌		
📶 Response Spectrum (ANSYS)	7	😥 Results		9		
Shape Optimization (ANSYS)		Chabia Chu	ab wal (ANGVC)			
🤓 Static Structural (ANSYS)		Static Stru	ctural (ANSYS)			
🛐 Steady-State Thermal (ANSYS)						
Thermal-Electric (ANSYS)						
👼 Transient Structural (ANSYS)						
🔃 Transient Thermal (ANSYS)						
Component Systems ■ Component ■ Component Systems ■ □						

Figure 11: Getting starting to mesh the geometry

N	letz 🛭 誟 Aktualisieren 📔 🌚 Netz 👻 🔍 Net	zsteuerung 👻 🚬 Metrisch					
Sti	Strukturbaum 7						
Projekt Modell (A4) Koordinatensysteme Koordinatensysteme Koordinatensysteme Strukturiertes Netz Strukturiertes Netz Statisch-mechanisch (A5) Kosung (A6) Lösungsinformationen							
	Details von "Strukturiertes Netz" - Strukturiertes Netz 🛛 📍						
De	tails von "Strukturiertes Netz" - Strukturiertes	Netz 🕈					
De	tails von "Strukturiertes Netz" - Strukturiertes Bereich	Netz 📍					
De	tails von "Strukturiertes Netz" - Strukturiertes Bereich Auswahlmethode	Netz 4 Geometrieauswahl					
De	tails von "Strukturiertes Netz" - Strukturiertes Bereich Auswahlmethode Geometrie	Netz Netz P Geometrieauswahl 6 Flächen					
	tails von "Strukturiertes Netz" - Strukturiertes Bereich Auswahlmethode Geometrie Definition	Netz Geometrieauswahl 6 Flächen					
	tails von "Strukturiertes Netz" - Strukturiertes Bereich Auswahlmethode Geometrie Definition Unterdrückt	Retz					
	tails von "Strukturiertes Netz" - Strukturiertes Bereich Auswahlmethode Geometrie Definition Unterdrückt Begrenzung mit Zwangsbedingung versehen	Netz Geometrieauswahl 6 Flächen Nein Nein					
	tails von "Strukturiertes Netz" - Strukturiertes Bereich Auswahlmethode Geometrie Definition Unterdrückt Begrenzung mit Zwangsbedingung versehen Erweitert	Netz Netz P Geometrieauswahl 6 6 Flächen Nein Nein					
	tails von "Strukturiertes Netz" - Strukturiertes Bereich Auswahlmethode Geometrie Definition Unterdrückt Begrenzung mit Zwangsbedingung versehen Erweitert Angegebene Seiten	Netz Geometrieauswahl 6 Flächen Nein Nein Keine					
	tails von "Strukturiertes Netz" - Strukturiertes Bereich Auswahlmethode Geometrie Definition Unterdrückt Begrenzung mit Zwangsbedingung versehen Erweitert Angegebene Seiten Angegebene Ecken	Netz Geometrieauswahl 6 Flächen Nein Nein Keine Keine					
	tails von "Strukturiertes Netz" - Strukturiertes Bereich Auswahlmethode Geometrie Definition Unterdrückt Begrenzung mit Zwangsbedingung versehen Erweitert Angegebene Seiten Angegebene Ecken Angegebene Enden	Netz Geometrieauswahl 6 Flächen Nein Nein Keine Keine Keine					

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) \rightarrow Insert \rightarrow 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 \rightarrow Static-Mechanic (right click) \rightarrow Insert \rightarrow 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 \rightarrow Static-Mechanic (right click) \rightarrow Insert \rightarrow 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 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



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 \rightarrow Force \rightarrow 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 \rightarrow Total	Strain		
and			
Details \rightarrow Direction $\rightarrow X$	K-axis		
De	tails von "Normale elasti:	sche Dehnung" 🛛 🗸 🕂	
	Bereich		
	Auswahlmethode	Geometrieauswahl	
	Geometrie	Alle Körper	
	Definition		
	Тур	Normale elastische behinang	
\sim	Ausrichtung	X-Achse)
	Durch	Zeit	
	Zeit anzeigen	Letzte	
	Koordinatensystem	Globales Koordinatensystem	
	Zeit/Verlauf berechnen	Ja	
	Kennung		
	Integrationspunkter	gebnisse	
	Anzeigeoption	Gemittelt	
	Ergebnisse		
	Minimum	-2,0288e-003 m/m	
	Maximum	2,0288e-003 m/m	
+	Informationen		





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

c. Contour Plot X-Component of Stress

Structure Tree \rightarrow Total Stress and Details \rightarrow Direction \rightarrow X-axis

De	etails von "Normalspannung" 🛛 🗣					
Ξ	Bereich					
	Auswahlmethode	Geometrieauswahl				
	Geometrie	Alle Körper				
Ξ	Definition	·				
	Тур	Normalspannung				
<	Ausrichtung	X-Achse				
	Durch	Zeit				
	Zeit anzeigen	Letzte				
	Koordinatensystem	Globales Koordinatensystem				
	Zeit/Verlauf berechnen	Ja				
	Kennung					
Ξ	Integrationspunkter	gebnisse				
	Anzeigeoption	Gemittelt				
Ξ	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 $\sigma_{pred} = 206$ N/mm² (**Fehler! Verweisquelle konnte nicht gefunden werden.**), whereas the ultimate yield stress $\sigma_{yield} = 235$ N/mm² is higher. That means the failure criterion $\sigma_{pred} > \sigma_{yield}$ 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.