Exercise 1

3-Point Bending Using the GUI and the Bottom-up-Method

Contents

Learn how to
Given2
Questions
Taking advantage of symmetries2
A. Preprocessor (Setting up the Model)
A.1 Build the Geometry by Means of the Bottom-up Method3
A.1.1 Create Keypoints
A.1.2 Create Lines via Connecting the Keypoints
A.1.3 Create Area via Connecting the Lines
A.1.4 Visualization of the created elements
A.2 Material Properties4
A.2.1 Define Material Properties4
A.3 Meshing
A.3.1 Assign the Finite Element Type5
A.3.2 Define Real Constants7
A.3.3 Meshing
A.4 Applying Loads and Boundary Conditions9
A.4.1 Boundary Conditions
A.4.2 Applying the Loads
B. Solving
B.1 Basic Settings: Analysis Type10
B.2 Start solving11
C. Postprocessor
C.1 Contour Plot of Deformed Shape11
C.2 Contour Plot: X-Component of Total Strain14
C.3 Contour Plot X-Component of Stress14
C.4 Contour Plot: Von-Mises Equivalent Stress15
Answering the Questions:

Learn how to ...

- ... use the ANSYS GUI (graphical user interface)
- ... build a model using the "bottom up method"
- ... 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

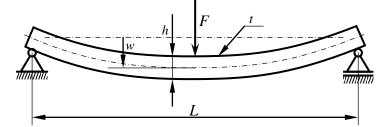


Fig. 1: Beam under 3-point bending.

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
Ε	= 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!

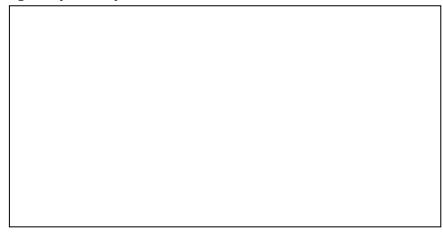


Fig. 2: Space for drawing of simplified beam model taking advantage of symmetry.

A. Preprocessor (Setting up the Model)

A.1 Build the Geometry by Means of the Bottom-up Method

The "bottom-up method" generates the geometry of the system going from key points (0D) to lines (1D) and areas (2D) up to volumes (3D).

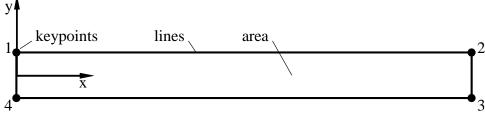


Fig. 3: 2D geometry of the beam

A.1.1 Create Keypoints

We need to create four keypoints:

 $\textit{Main Menu} \rightarrow \textit{Preprocessor} \rightarrow \textit{Modeling} \rightarrow \textit{Create} \rightarrow \textit{Keypoints} \rightarrow \textit{In Active CS:}$

Create Keypoints in Active Coordinate System	×
[K] Create Keypoints in Active Coordinate System	
NPT Keypoint number	1
X,Y,Z Location in active CS	0 30 0
OK Apply	Cancel Help

Fig. 4: Dialog box for the creation of keypoints

Enter the corresponding coordinates of each of the four necessary keypoints and confirm each by pressing **[Apply]**. After you have entered all four keypoints, press **[OK]** to close the dialog box.

A.1.2 Create Lines via Connecting the Keypoints

Main Menu \rightarrow *Preprocessor* \rightarrow *Modeling* \rightarrow *Create* \rightarrow *Lines* \rightarrow *Straight Line* Connect the keypoints by clicking with the mouse pointer \uparrow on the first and then on the second keypoint of each line. After creating the four lines confirm with **[OK]**.

A.1.3 Create Area via Connecting the Lines

Main Menu \rightarrow *Preprocessor* \rightarrow *Modeling* \rightarrow *Create* \rightarrow *Areas* \rightarrow *Arbitrary* \rightarrow *By Lines* Pick the four lines by the mouse pointer \uparrow and then confirm with **[OK]**.

A.1.4 Visualization of the created elements

To plot the numeric IDs associated with each created item, select ...

Utility Menu \rightarrow *PlotCtrls* which brings up the following dialog box

Plot Numbering Controls	×
[/PNUM] Plot Numbering Controls	
KP Keypoint numbers	🔽 On
LINE Line numbers	C Off
AREA Area numbers	🔽 On
VOLU Volume numbers	Off
NODE Node numbers	C off
Elem / Attrib numbering	No numbering
TABN Table Names	Off
SVAL Numeric contour values	C off
[/NUM] Numbering shown with	Colors & numbers
[/REPLOT] Replot upon OK/Apply?	Replot
ОК Арріу	Cancel Help

Fig. 5: Dialog box for plot controls, numbering

and switch on the numbering of required items (e.g. Keypoints, Areas). Then choose ... *Utility Menu* \rightarrow *Plot* \rightarrow ...

to visualize the created items.

A.2 Material Properties

A.2.1 Define Material Properties

Materials define the mechanical behavior of the FE model. We will use a simple linearelastic, isotropic material model.

Main Menu \rightarrow Preprocessor \rightarrow Material Props \rightarrow Material Models

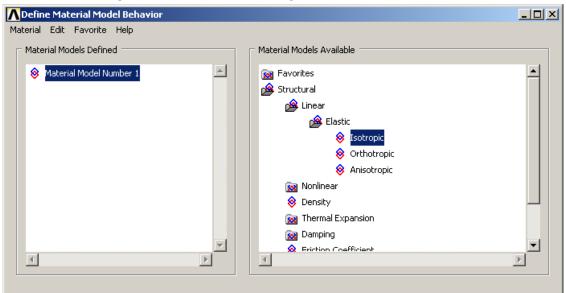


Fig. 6: Define the material properties.

Choose the simplest available material model by selecting (double click) from the tree of available models:

Structural \rightarrow Linear \rightarrow Elastic \rightarrow Isotropic (see Fig. 14)

Now a dialog box appears (Fig. 7), asking you for certain material parameters:

T1 Temperatures EX 210000 PRXY 0.3 Add Temperature Delete Temperature Graph Fig. 7: Define material	Linear Isotropic Properties for Material Number 1	×
PRXY 0.3	T1	
Fig. 7: Define material	EX 210000	
		Fig. 7: Define material

Enter the material properties Young's modulus (EX) and Poisson's ratio (PRXY). Confirm with **[OK]** and close the "Material Models" window (Fig. 6).

A.3 Meshing

A.3.1 Assign the Finite Element Type

Before we can create a mesh for our model, we need to choose which type of finite element we will want to use for our mesh.

▲Element Types			×	
Defined Flowerk Turney				
Defined Element Types: NONE DEFINED				
Add Opt	ions	Delete		
				Fig 0. Empty list
Close		Help		Fig. 8: Empty list defined element type

Prepr

Press the button [Add] and select

Structural Mass \rightarrow Solid \rightarrow Quad 4node 42 and confirm with **[OK]**.

Library of Element Types		×
Library of Element Types	Structural Mass Link Beam Pipe Solid Shell Solid-Shell Constraint	Quad 4node 42 4node 182 8node 183 8node 82 Triangle 6node 2 Axi-har 4node 25 Quad 4node 42
Element type reference number	1 Apply Cancel	Help

Fig. 9: Selection of finite element type (type number = 42).

	ilement Types			×	
	Defined Element Types:				
l	Type 1 PLANE42	2			
	Add	Options	Delete		Fig. 10: List of definedelementtypes
	Close		Help		including our previous choice (Plane42).

We now have defined an element type with ID 1. Plane42 is a simple two-dimensional solid (plane) with four corner nodes and no interior nodes.

Choosing the **[Options...]** button (Fig. 10) brings up the following window.

APLANE42 element type options	×
Options for PLANE42, Element Type Ref. No. 1	
Element coord system defined K1	Parall to global
Extra displacement shapes K2	Include
Element behavior K3	Plane strs w/thk
Extra stress output K5	No extra output
Extra surface output K6	No extra output
OK	Help

Fig. 11: Selection of the finite element behavior.

Choose "*Element behavior K3*" as "*Plane strs w/thk*" (= plane stress with thickness) and confirm with **[OK]**. Then close the previous window by hitting **[Close]**.

This option is appropriate for plane plate-like structures which are only loaded in direction of the plane (Keyword: "plane stress").

A detailed description of the finite element types is available in the help function of ANSYS. Press **[Help]** in the window (Fig. 11) for a detailed description of the used element type.

A.3.2 Define Real Constants

"Real Constants" are simple arrays of floating point numbers. They are used to configure the behavior of element types. Therefore their concrete meaning depends on the element type they are assigned to.

 $\textit{Main Menu} \rightarrow \textit{Preprocessor} \rightarrow \textit{Real Constants} \rightarrow \textit{Add/Edit/Delete}$

First push the **[Add...]** button in ...

∧Real Constants	×
Defined Real Constant	Sets
NONE DEFINED	
Add Edit	Delete
Close	Hala
Close	Help

Fig. 12: Selection of "real constant" sets.

then choose Type 1 and confirm with [OK

🔥 Element Type for Real C 💶 🗖 🗙	l
Choose element type:	
Type 1 PLANE42	
OK Cancel	

Fig. 13: Choose the element type.

Real Constant Set Number 1, for PLANE42	×
Element Type Reference No. 1	
Real Constant Set No.	1
Real Constant for Plane Stress with Thickness (KEYOPT(3)=3)	
Thickness THK	20
OK Apply	Cancel Help
OK Apply	Cancel Help

Fig. 14: Define the real constants.

Enter the appropriate value for the thickness of the beam and close the dialog box by confirming with **[OK]** and **[Close]** the following window.

A.3.3 Meshing

We now create a mesh by subdividing the geometry into finite elements of the specified type:

Preprocessor \rightarrow Meshing \rightarrow Mesh \rightarrow Areas \rightarrow Mapped \rightarrow 3 or 4 sided

Pick the area with the mouse pointer \uparrow and confirm with **[OK]**.



A.4 Applying Loads and Boundary Conditions

A.4.1 Boundary Conditions

To setup the necessary support, we need to define nodal displacement constraints:

 $Preprocessor \rightarrow Loads \rightarrow Define \ Loads \rightarrow Apply \rightarrow Structural \rightarrow Displacement \rightarrow On$ Nodes

Pick the line to be fixed with the mouse pointer \uparrow and confirm with **[OK]**.

Apply U,ROT on Nodes	×
[D] Apply Displacements (U,ROT) on Nodes	
Lab2 DOFs to be constrained	All DOF UX UY
Apply as	Constant value
If Constant value then:	
VALUE Displacement value	0.0
OK Apply Cance	el Help

Fig. 16: Applying boundary conditions

Set the upper value to "All DOF" (DOF = degrees of freedom) to apply a fully constrained boundary condition for the selected nodes and confirm with **[OK]**.

A.4.2 Applying the Loads

Preprocessor \rightarrow Loads \rightarrow Define Loads \rightarrow Apply \rightarrow Structural \rightarrow Force/Moment \rightarrow On Nodes

Pick appropriate nodes to apply the load and confirm with **[OK]**.

Apply F/M on Nodes	X
[F] Apply Force/Moment on Nodes	
Lab Direction of force/mom	FY 💌
Apply as	Constant value
If Constant value then:	
VALUE Force/moment value	500000
OK Apply Cancel	Help

Fig. 17: Apply forces on nodes.

Enter the values in the appearing window and confirm with **[OK]**.

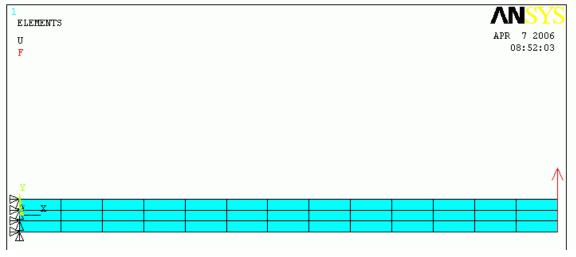


Fig. 18: Beam with elements, load and boundary conditions.

B. Solving

B.1 Basic Settings: Analysis Type

To set the analysis type to "static" choose the following path in the menu: *Main Menu* \rightarrow *Solution* \rightarrow *Analysis Type* \rightarrow *New Analysis*

New Analysis			×
[ANTYPE] Type of analysis			
		Static	
		🔿 Modal	
		C Harmonic	
		C Transient	
		C Spectrum	
		C Eigen Buckling	
		Substructuring/CMS	
ОК	Cancel	Help	

Fig. 19: Choose type of analysis.

Then select the option "Static".

B.2 Start solving

Main Menu \rightarrow *Solution* \rightarrow *Solve* \rightarrow *Current LS* and confirm with **[OK]**.

C. Postprocessor

The post-processor is a tool to visualize the results of the solution phase.

C.1 Contour Plot of Deformed Shape

Go to ...

 $\textit{Main Menu} \rightarrow \textit{General Postproc} \rightarrow \textit{Contour Plot} \rightarrow \textit{Nodal Solu}$

... and then choose: Nodal Solution \rightarrow DOF Solution \rightarrow Y-Component of displacement

Contour Nodal Solution	Data					
Titem to be contoured						
Ravorites						-
Modal Solution						
DOF Solution						
🧭 X-Compo	onent of displa	icement				
🕜 Y-Compo	onent of displa	icement				
🍘 Displace	ment vector s	IW				
🛃 Stress						
🛜 Total Strain						
🛜 Elastic Strain						
🛜 Plastic Strain						
🛜 Creep Strain						
🛜 Thermal Strain						
🛜 Total Mechanic	al and Therma	al Strain				
🌮 Swelling strain						-
1						Þ
Undisplaced shape key —						
ondisplaced shape key						
Undisplaced shape key	Deformed sh	ape only				-
Scale Factor	True Scale				• 1	
Additional Options						8
		OK	A	pply	Cancel	Help

Fig. 20: Select the type of contour plot.

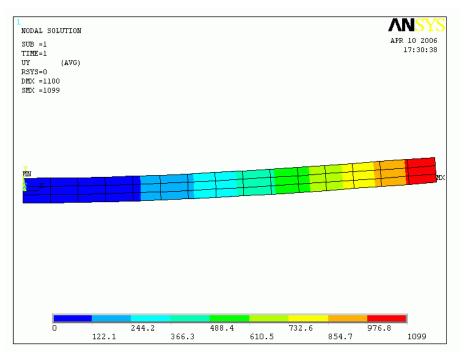


Fig. 21: Deformed shape of the beam (scaled!). Colors represent displacements u_y

In the command line, first enter **/dscale**, **1**, **1.0** to turn off automatic scaling of the displacement plot. Press **[Enter]**, then enter **/replot** and again confirm by pressing **[Enter]**.

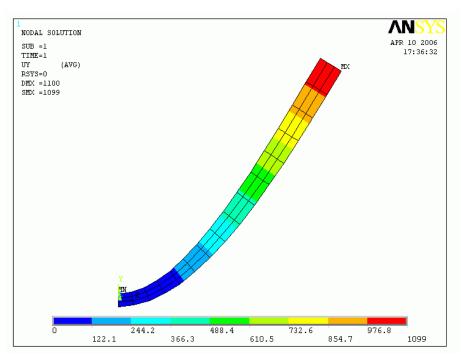


Fig. 22: Deformed shape of the beam (not scaled!).

Discuss the results which can be seen on the two figures above!

Correcting the Model:

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

 $Preprocessor \rightarrow Loads \rightarrow Define \ Loads \rightarrow Delete \rightarrow All \ Load \ Data \rightarrow All \ Forces \rightarrow On$ All Nodes

... to delete the wrong loads. Perform the step earlier described in chapter A.4.2 to apply the correct force onto the nodes. Also, carry out a new solution step (see chapter B.2) and replot the new results (see chapter C.1).

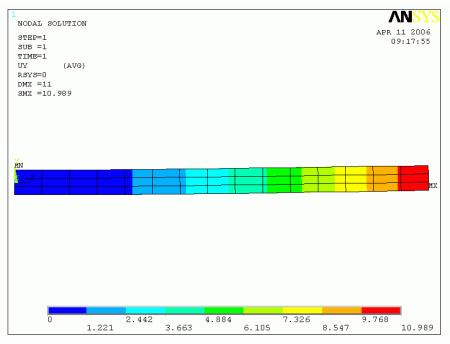


Fig. 23: Deformed shape of the beam (not scaled!) resulting from corrected force.

C.2 Contour Plot: X-Component of Total Strain

General Postproc \rightarrow Plot Results \rightarrow Contour Plot \rightarrow Element Solu

In the appearing window choose: Element Solution \rightarrow Total Strain \rightarrow X-Component of total strain

STEP=1 SUB =1 TIME=1 EPTOX RSYS=0 DMX =1 SMN =- SMX =.	(NOA 1 .970E-0 970E-03	.vg)				APR :	NSYS 11 2006 9:19:55
M	X						·
	970E-03		539E-03	108E-03		.754E-03	

Fig. 24: Contour plot of x-component of total strain.

C.3 Contour Plot X-Component of Stress

General Postproc \rightarrow Plot Results \rightarrow Contour Plot \rightarrow Element Solu

In the appearing window choose: Element Solution \rightarrow Stress \rightarrow X-Component of total stress

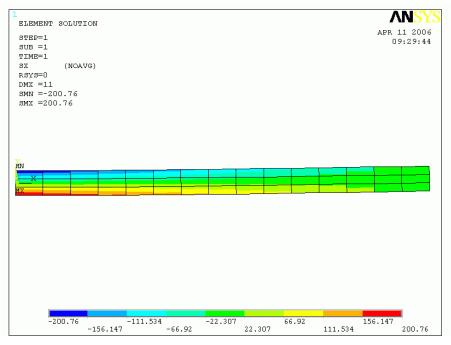


Fig. 25: Contour plot of x-component of stress.

C.4 Contour Plot: Von-Mises Equivalent Stress

General Postproc \rightarrow Plot Results \rightarrow Contour Plot \rightarrow Element Solu

In the appearing window choose the options: Element Solution \rightarrow Stress \rightarrow von Mises stress

L ELEMENT SOLUT STEP=1 SUB =1 TIME=1 SEQV (NOA DMX =11 SMM =3.357 SMX =205.688								APR 1: 10:	
Y NX									MI
3.357	25.839	48.32	70.801	93.282	115.764	L38.245	160.726	183.207	205.688

Fig. 26: 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 (Fig. 26), 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 (Fig. 26, 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 (Fig. 23). The full length beam under 3-pointbending (Fig. 1) will show a maximum deflection of the same amount in the middle.