Exercise 1

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

Contents

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

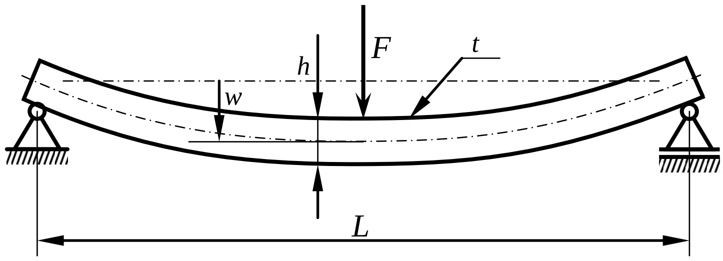

Fig. 1: Beam under 3-point bending.

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
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. 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).

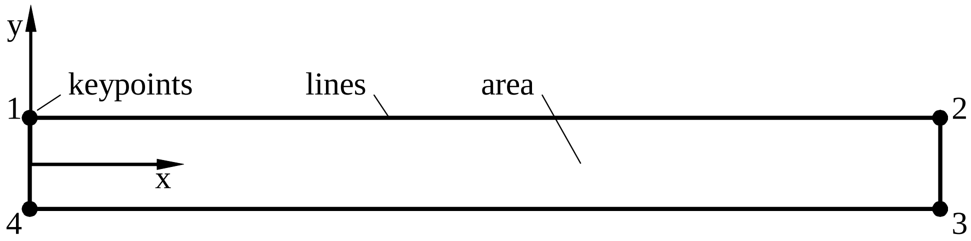

Fig. 3: 2D geometry of the beam

A.1.1 Create Keypoints

We need to create four keypoints:

Main Menu → Preprocessor → Modeling → Create → Keypoints → In Active CS:

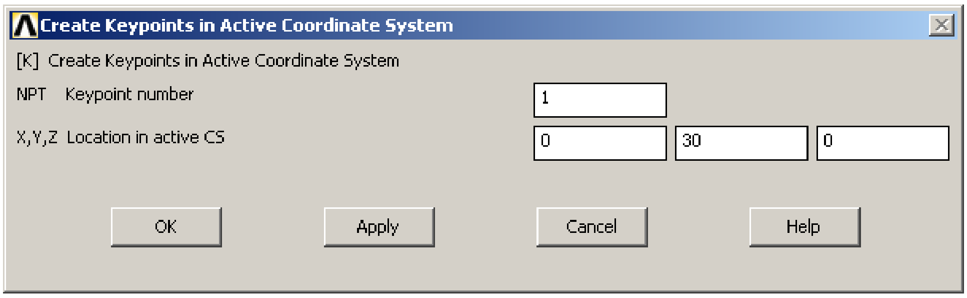

Fig. 4: Dialog box for the creation of keypoints

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 → Preprocessor → Modeling → Create → Lines → Straight Line

Connect the keypoints by clicking with the mouse pointer ↑ 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 → Preprocessor → Modeling → Create → Areas → Arbitrary → By Lines

Pick the four lines by the mouse pointer ↑ 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 → PlotCtrls

which brings up the following dialog box

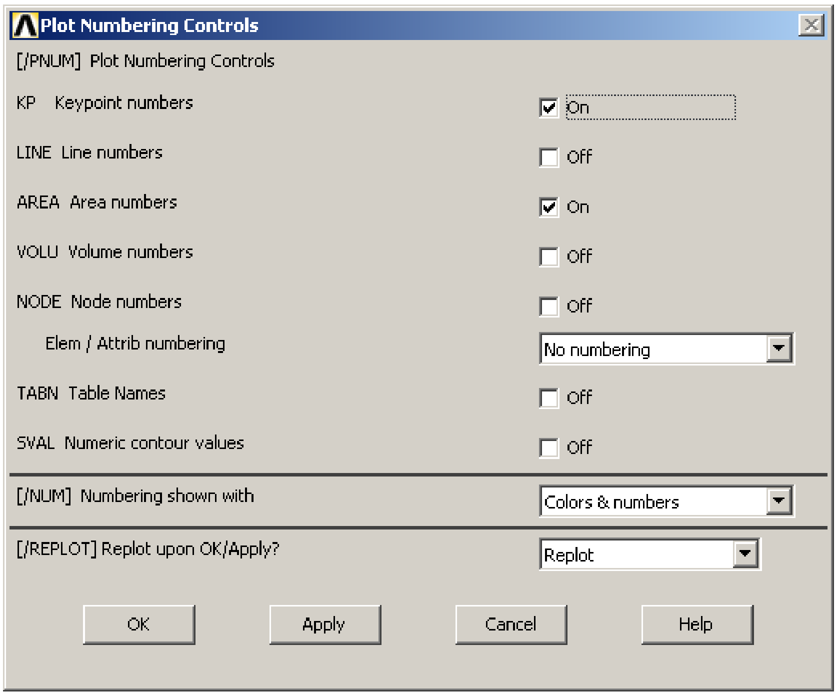

Fig. 5: Dialog box for plot controls, numbering

and switch on the numbering of required items (e.g. Keypoints, Areas). Then choose ...

Utility Menu → Plot → ...

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 linear-elastic, isotropic material model.

Main Menu → Preprocessor → Material Props → Material Models

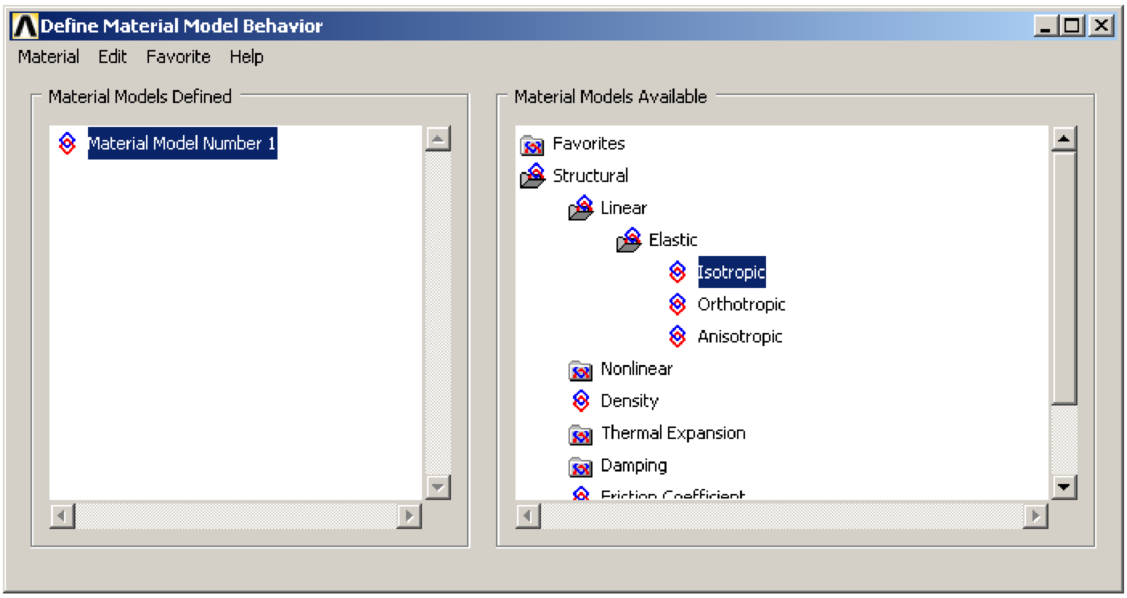

Fig. 6: Define the material properties.

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

Structural → Linear → Elastic → Isotropic

(see Fig. 14)

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

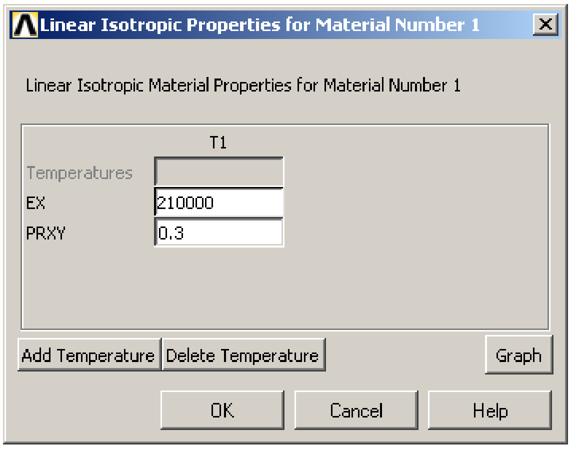

Fig. 7: Define material properties for material model number 1

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.

Preprocessor → Element Type → Add/Edit/Delete

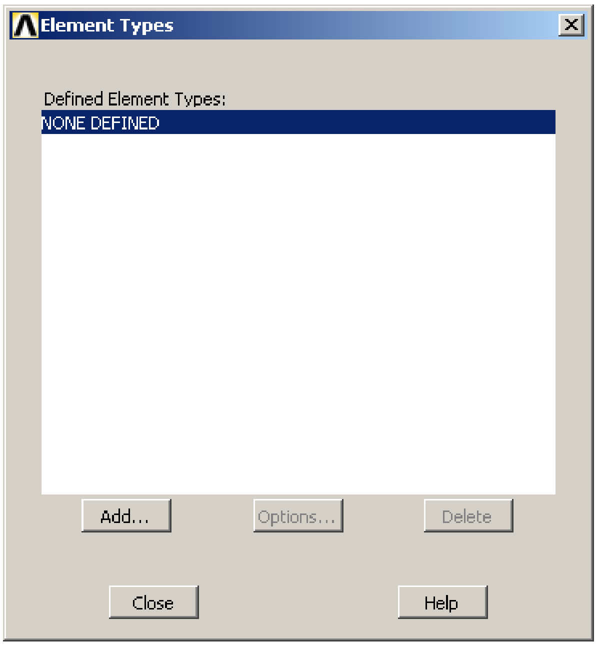

Fig. 8: Empty list of defined element types.

Press the button **[Add]** and select

Structural Mass → Solid → Quad 4node 42

and confirm with **[OK]**.

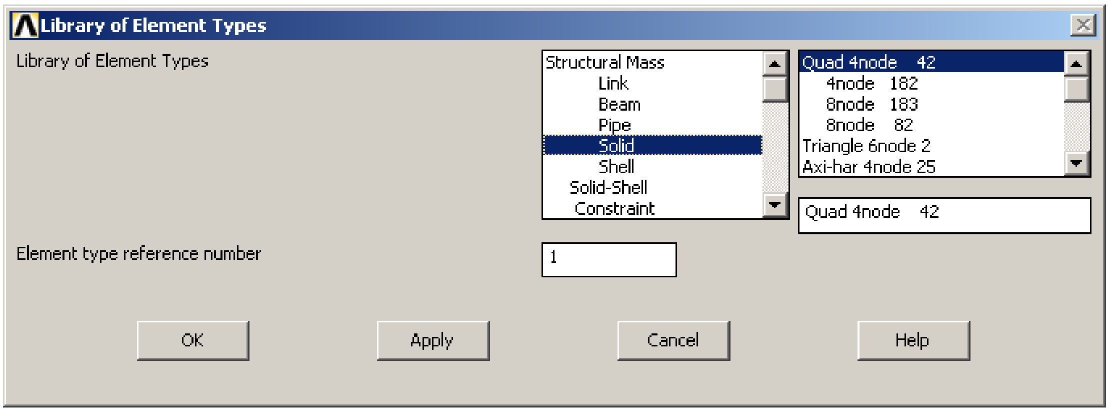

Fig. 9: Selection of finite element type (type number = 42).

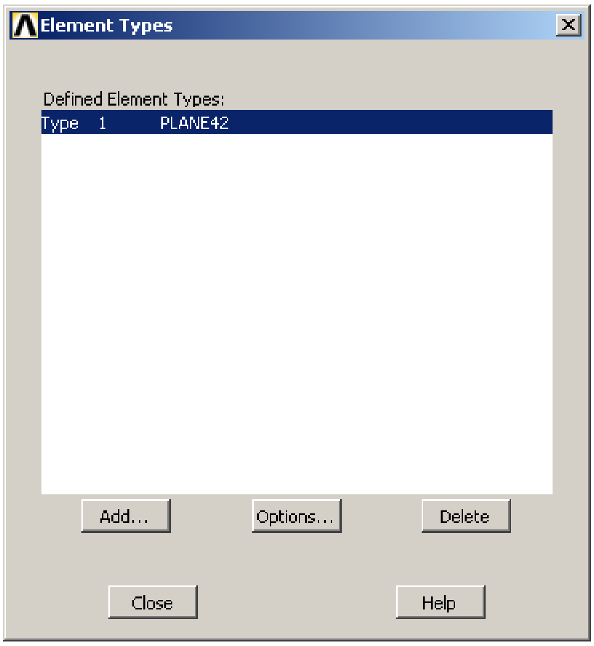

Fig. 10: List of defined element types including our previous choice (Plane42).

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.

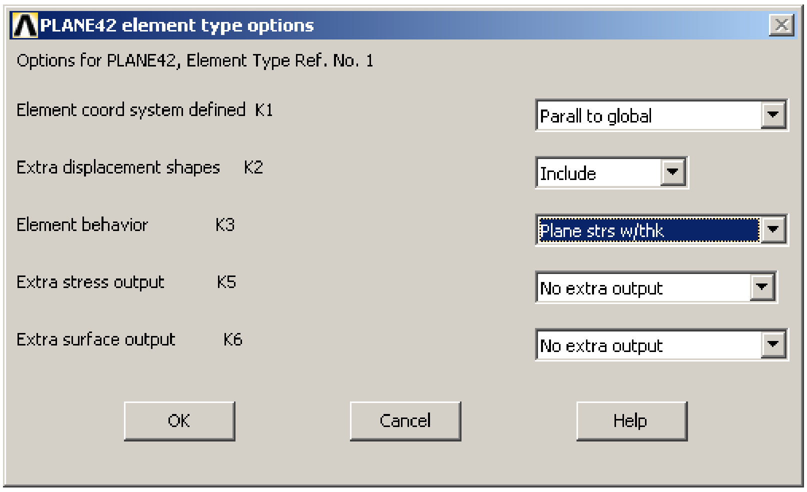

Fig. 11: Selection of the finite element behavior.

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.

Main Menu → Preprocessor → Real Constants → Add/Edit/Delete

First push the **[Add...]** button in ... ... then choose Type 1 and confirm with **[OK]**

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

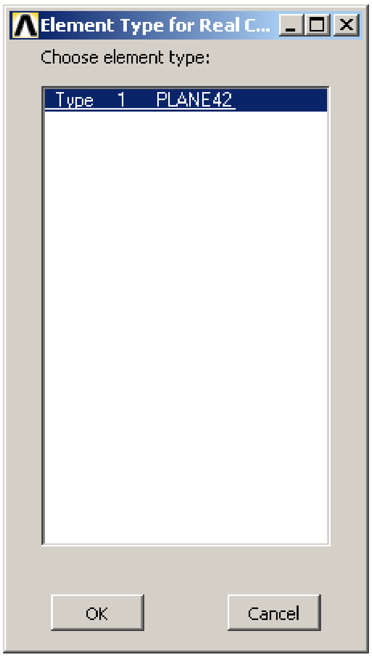

Fig. 13: Choose the element type.

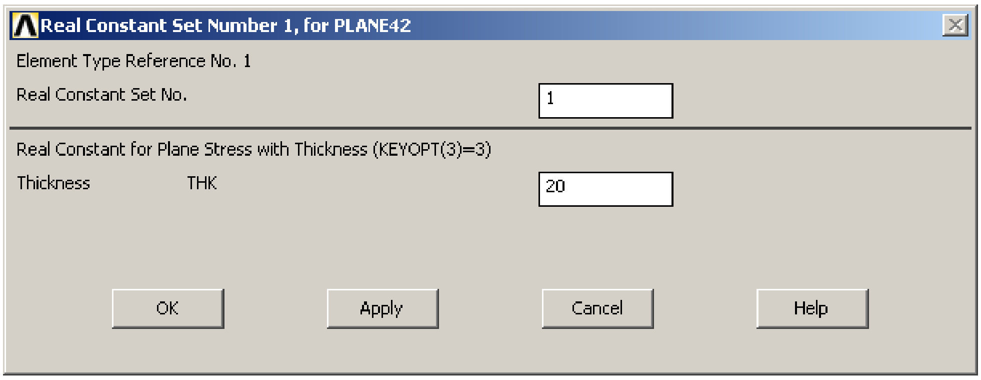

Fig. 14: Define the real constants.

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 → Meshing → Mesh → Areas → Mapped → 3 or 4 sided

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

Fig. 15: Meshed beam model

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 → Loads → Define Loads → Apply → Structural → Displacement → On Nodes

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

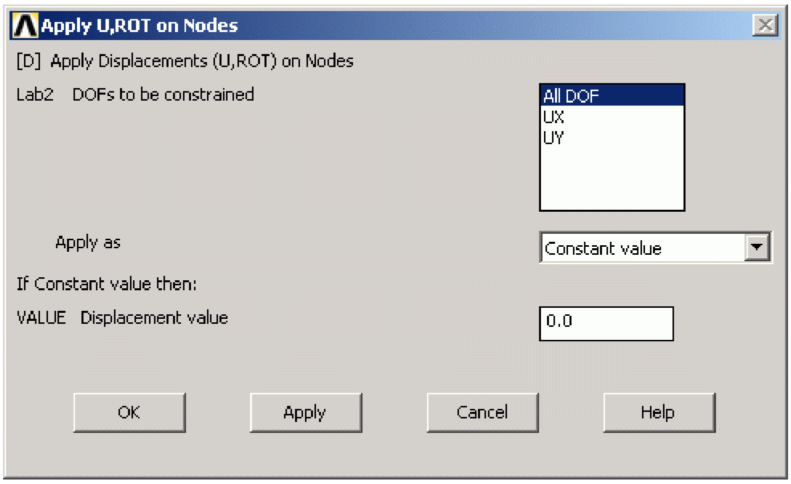

Fig. 16: Applying boundary conditions

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 → Loads → Define Loads → Apply → Structural → Force/Moment → On Nodes

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

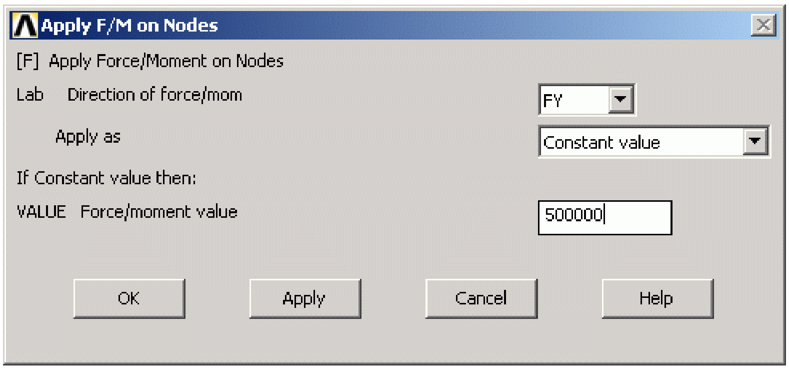

Fig. 17: Apply forces on nodes.

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 → Solution → Analysis Type → New Analysis

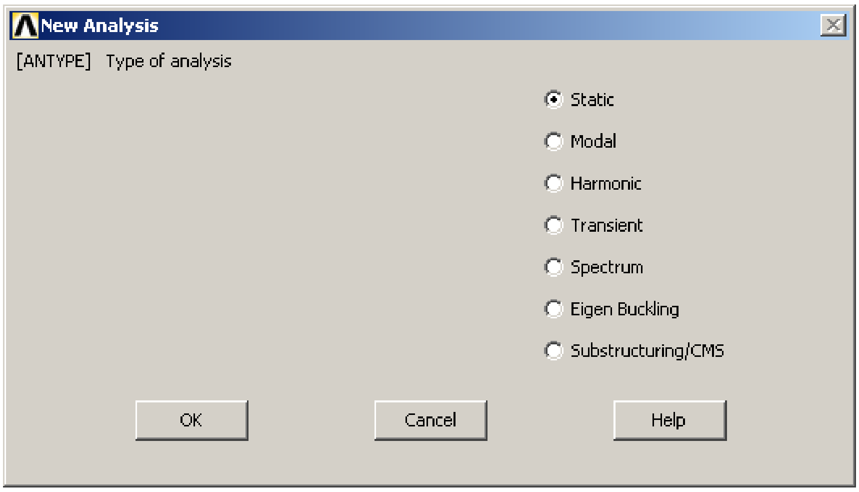

Fig. 19: Choose type of analysis.

Then select the option *“Static”.*

B.2 Start solving

Main Menu → Solution → Solve → 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 ...

Main Menu → General Postproc → Contour Plot → Nodal Solu

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

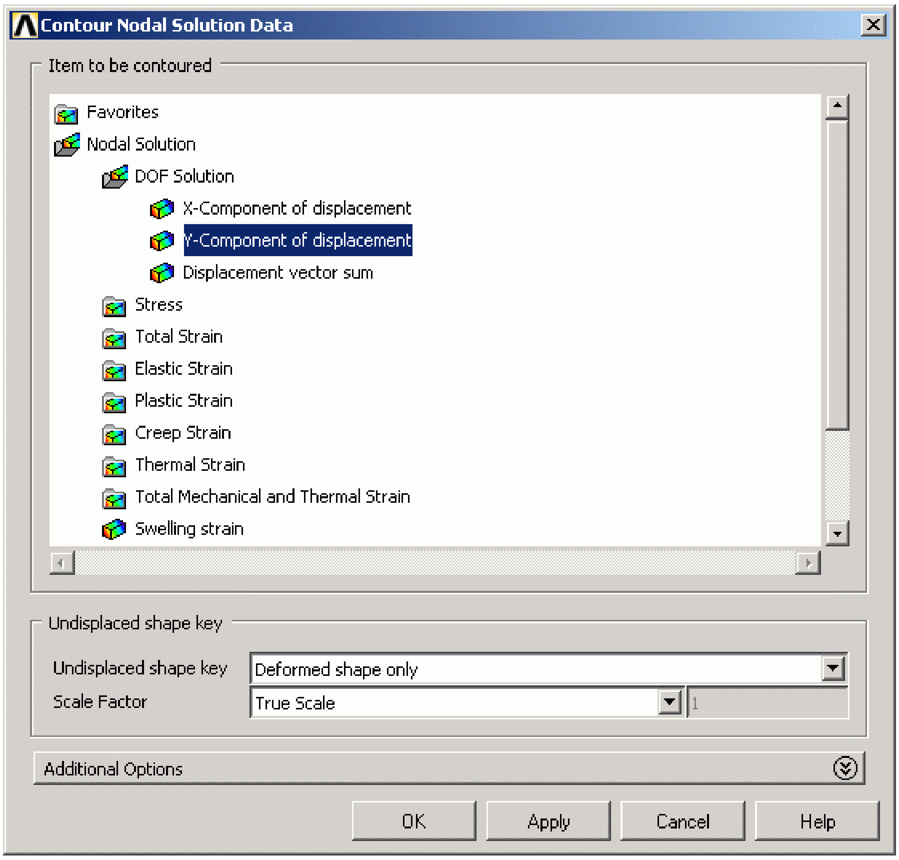

Fig. 20: Select the type of contour plot.

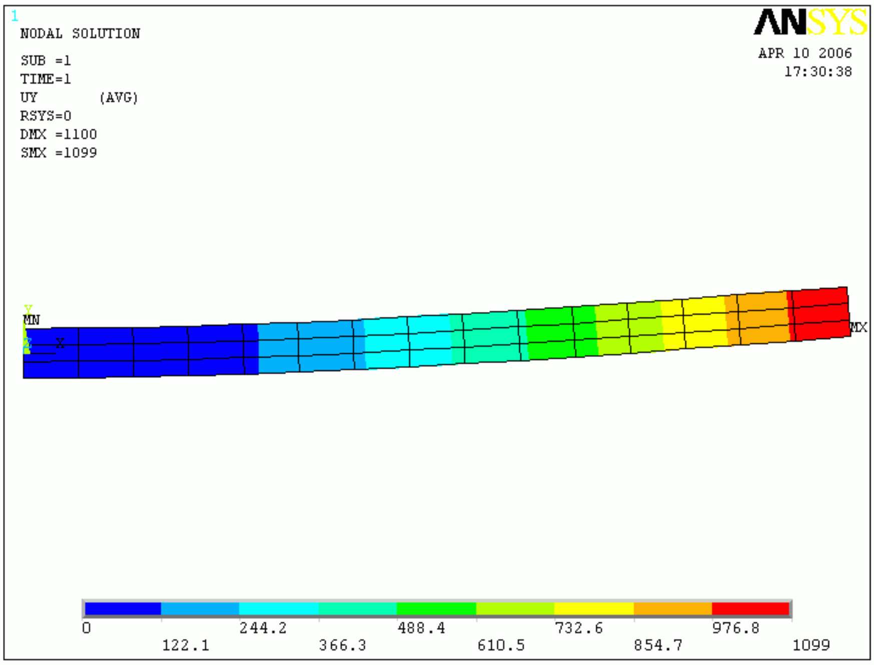

Fig. 21: Deformed shape of the beam (scaled!). Colors represent displacements u_y

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]**.

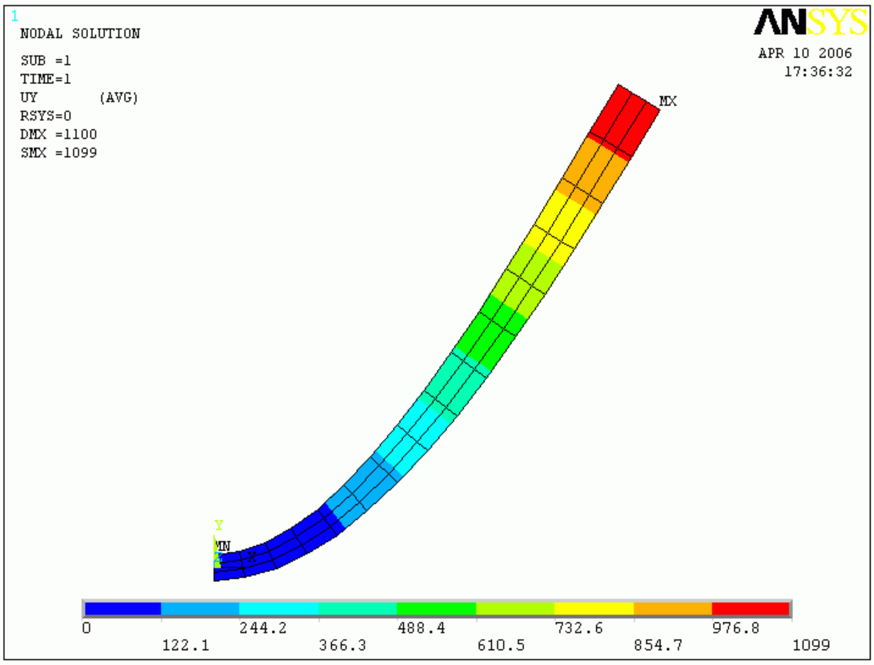

Fig. 22: Deformed shape of the beam (not scaled!).

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 → Loads → Define Loads → Delete → All Load Data → All Forces → 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).

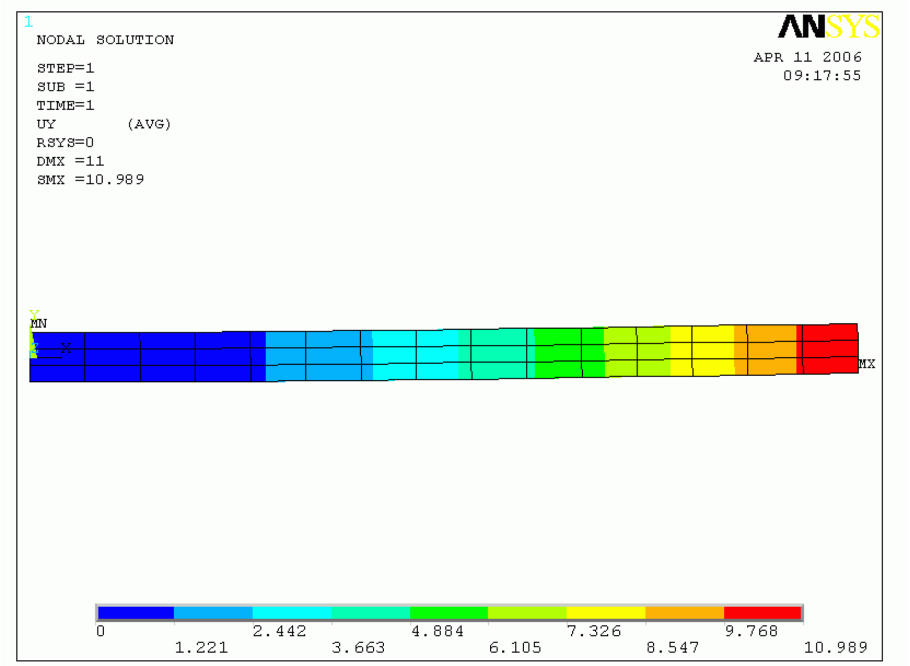

Fig. 23: Deformed shape of the beam (not scaled!) resulting from corrected force.

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

General Postproc → Plot Results → Contour Plot → Element Solu

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

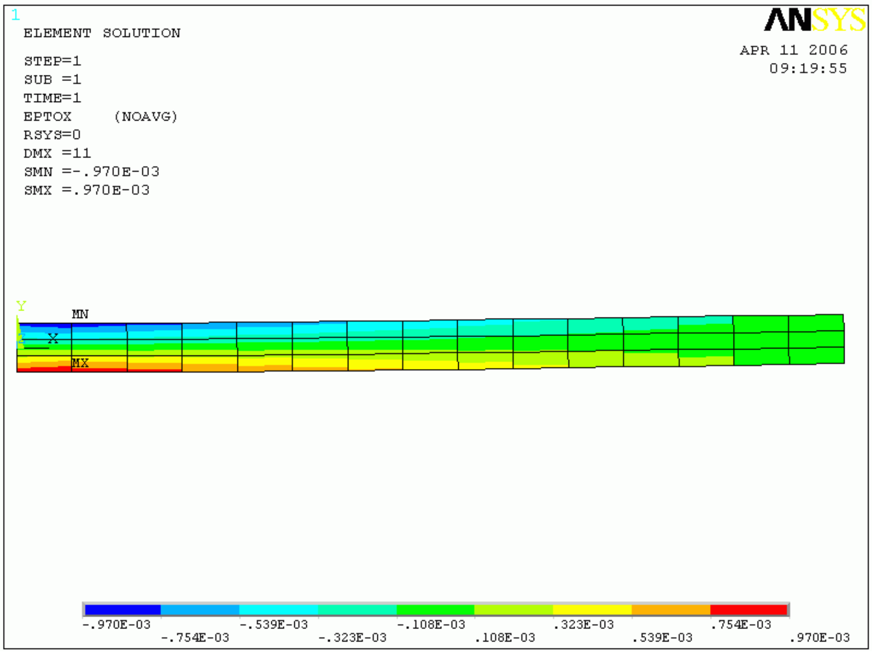

Fig. 24: Contour plot of x-component of total strain.

C.3 Contour Plot X-Component of Stress

General Postproc → Plot Results → Contour Plot → Element Solu

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

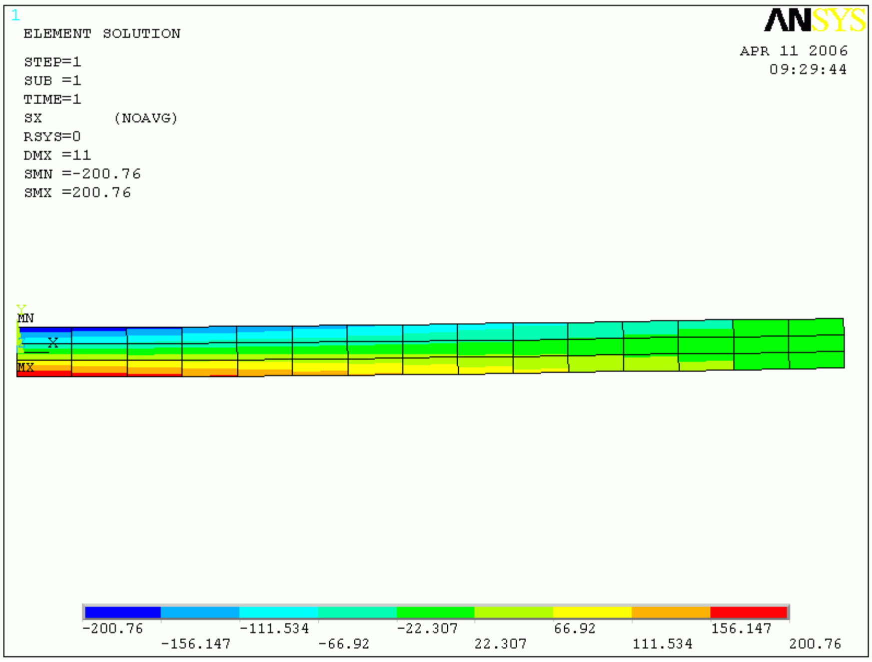

Fig. 25: Contour plot of x-component of stress.

C.4 Contour Plot: Von-Mises Equivalent Stress

General Postproc → Plot Results → Contour Plot → Element Solu

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

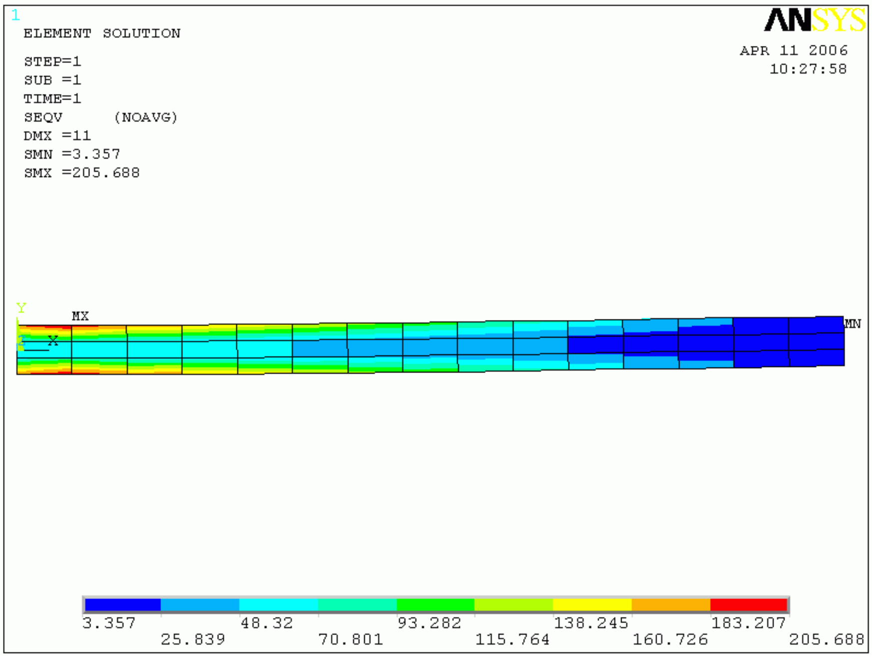

Fig. 26: Contour plot of Von-Mises equivalent stress.

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 σ_{pred} = 206 N/mm² (Fig. 26), 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 $\sigma_{\text{yield}}/\sigma_{\text{pred}}$ 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-point-bending (Fig. 1) will show a maximum deflection of the same amount in the middle.