EXERCISE 2: THREE POINT BENDING USING ANSYS WORKBENCH

CONTENTS

THE CANTILEVER

We want to simulate a beam under three point bending with a force F applied at the center as shown in Figure 1.

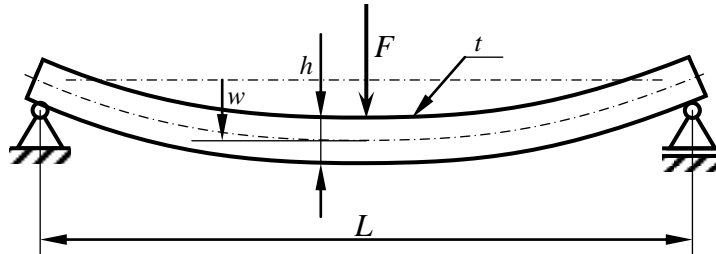**Figure 1: Beam under three point bending**

The following geometry and material data are required to model our problem:

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/mm²	Young's modulus
v	= 0.3	Poisson's ration
σ_{yield}	= 235 N/mm²	Allowable stress: yield stress of steel

Table 1: Geometry and material data.

Questions

With respect to this classic two-dimensional mechanical problem, we can state two questions:

1. Will the beam break? Where would it fail?
2. Assuming that it will *not* fail, 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 potential symmetries (Figure 2)!
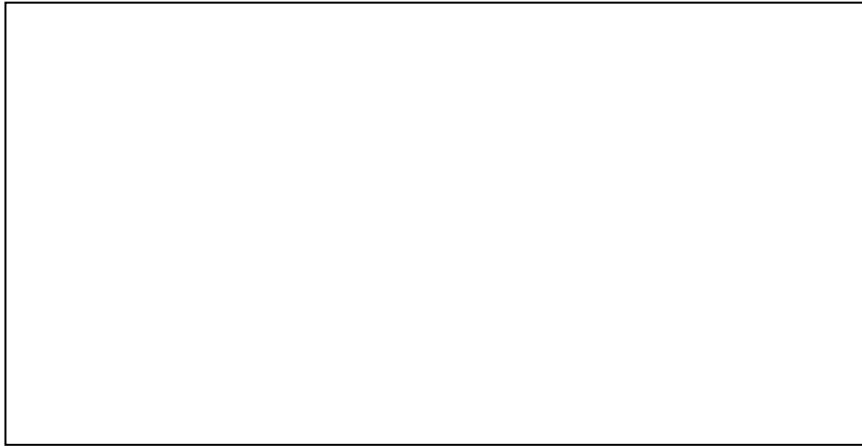
FIGURE 2: SPACE FOR DRAWING A SIMPLIFIED BEAM MODEL TAKING ADVANTAGE OF SYMMETRIES.

TAKING ADVANTAGE OF SYMMETRIES

STARTING AND CONFIGURING ANSYS WORKBENCH

First, log in using your credentials. From the Windows start menu select and run ANSYS Workbench (Figure 3), opening up ANSYS Workbench's project view (Figure 4).
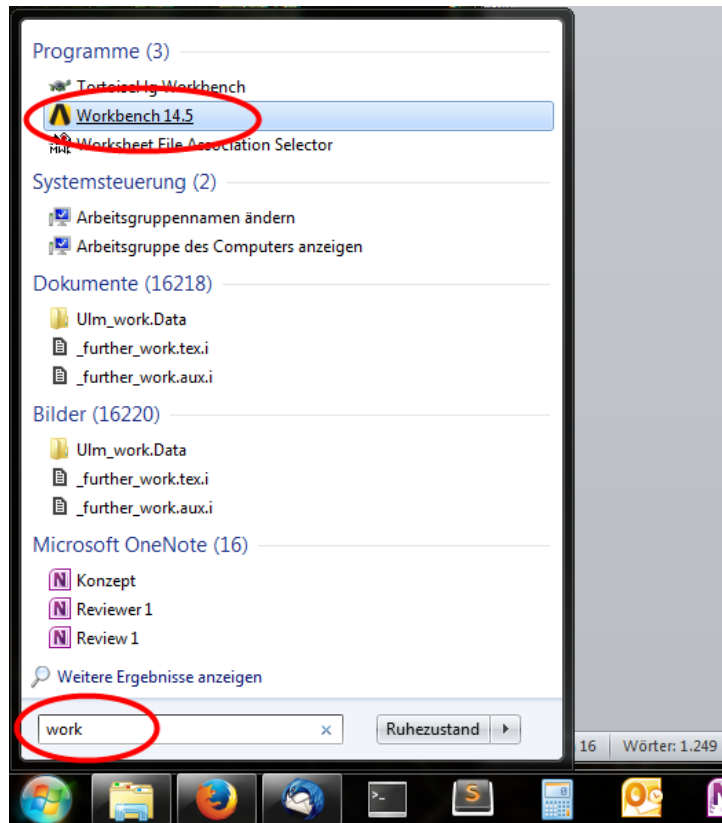

FIGURE 3: FIND AND RUN ANSYS WORKBENCH

Important: Please make sure to choose the right license type. After you have launched Workbench go to **Tools → License Preferences** and make sure that **ANSYS Academic Teach Advanced** is the default (i.e.: top-most) license option (Figure 5). Otherwise, use the **Move up** and **Move down** buttons to correct and finally **Apply** the settings.

Do **not** choose "ANSYS Academic Research" as the default license!

Be sure to change the language setting to **English**. Your instructor will show you how this is done.
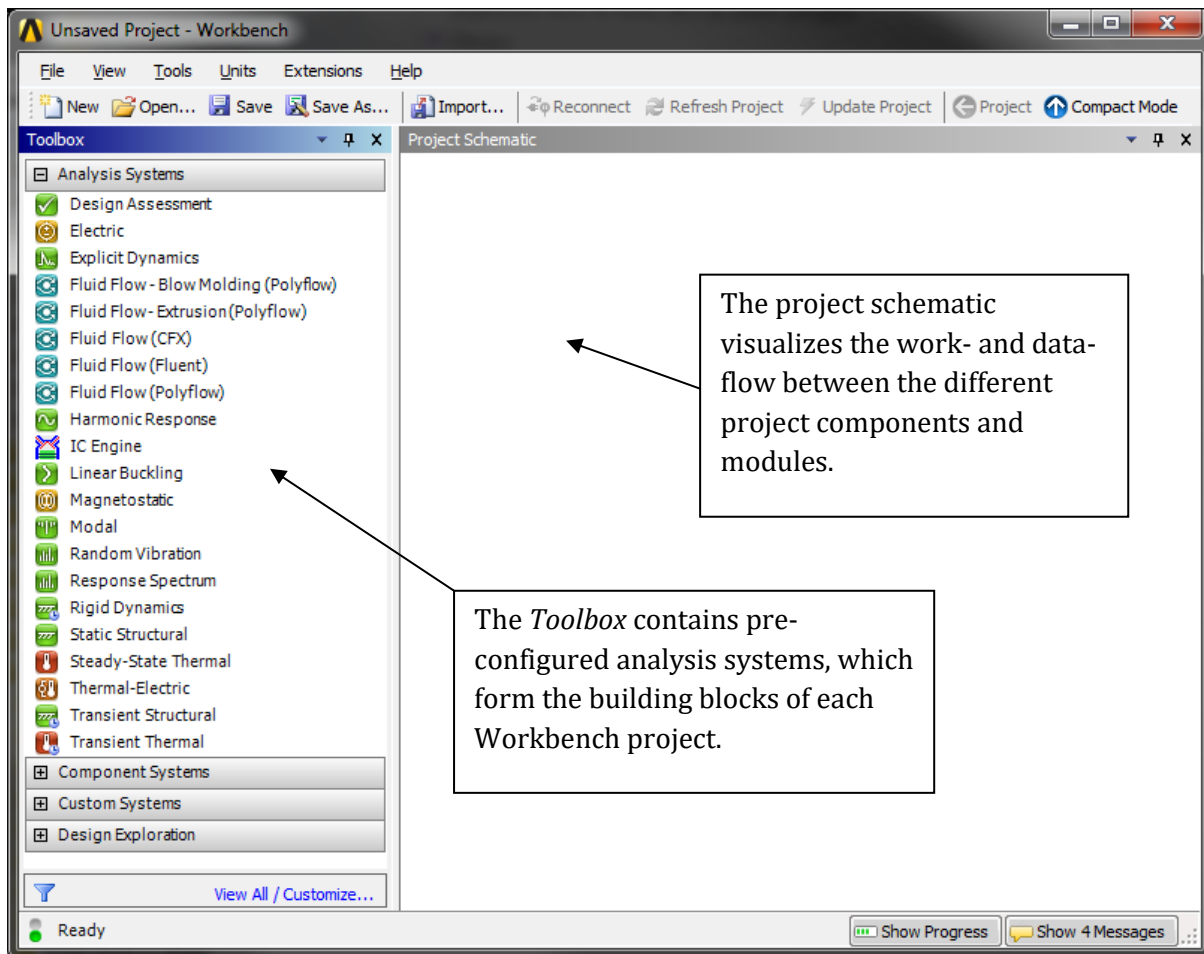
FIGURE 4: A NEW (AND YET EMPTY) ANSYS WORKBENCH SESSION

The project schematic visualizes the work- and data-flow between the different project components and modules.

The *Toolbox* contains pre-configured analysis systems, which form the building blocks of each Workbench project.
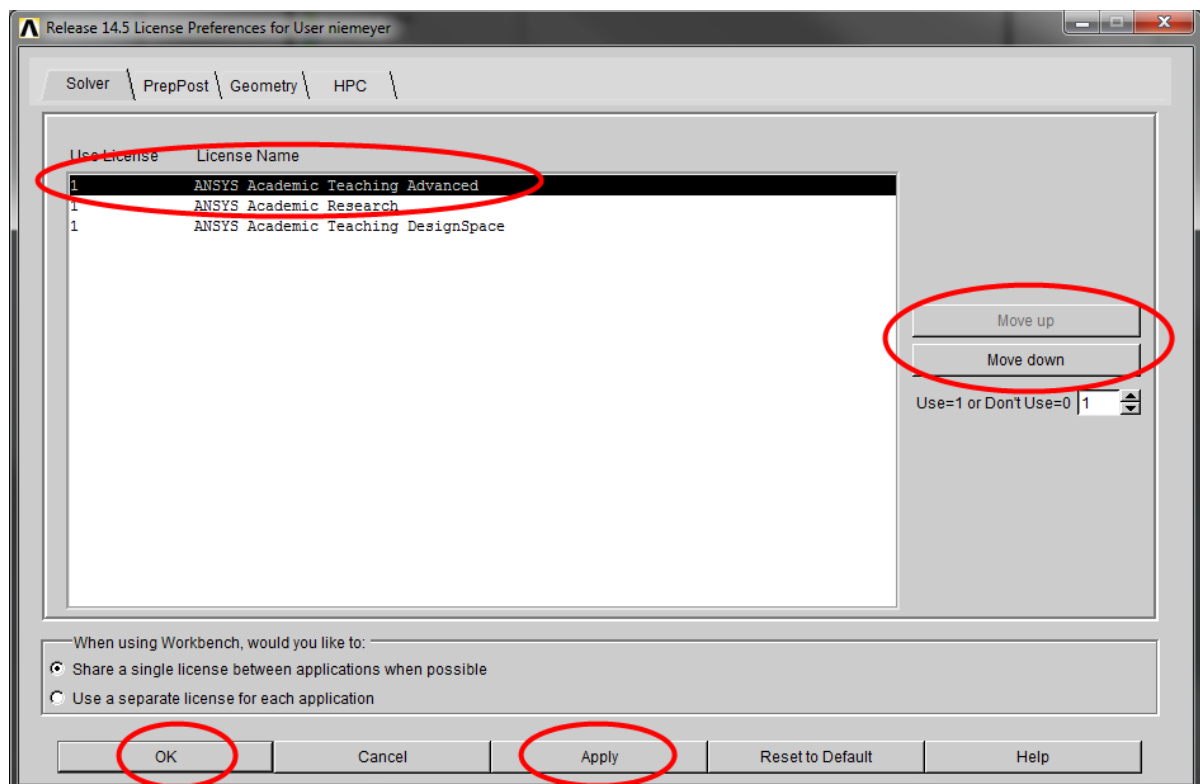

FIGURE 5: CONFIGURING THE LICENSE SETTINGS

A.   Pre-Processing: Setting up the Model

Before building the actual model, you need to create a new static-structural FE analysis by dragging and dropping the **Static Structural** analysis system onto the empty project schematic (Figure 6).
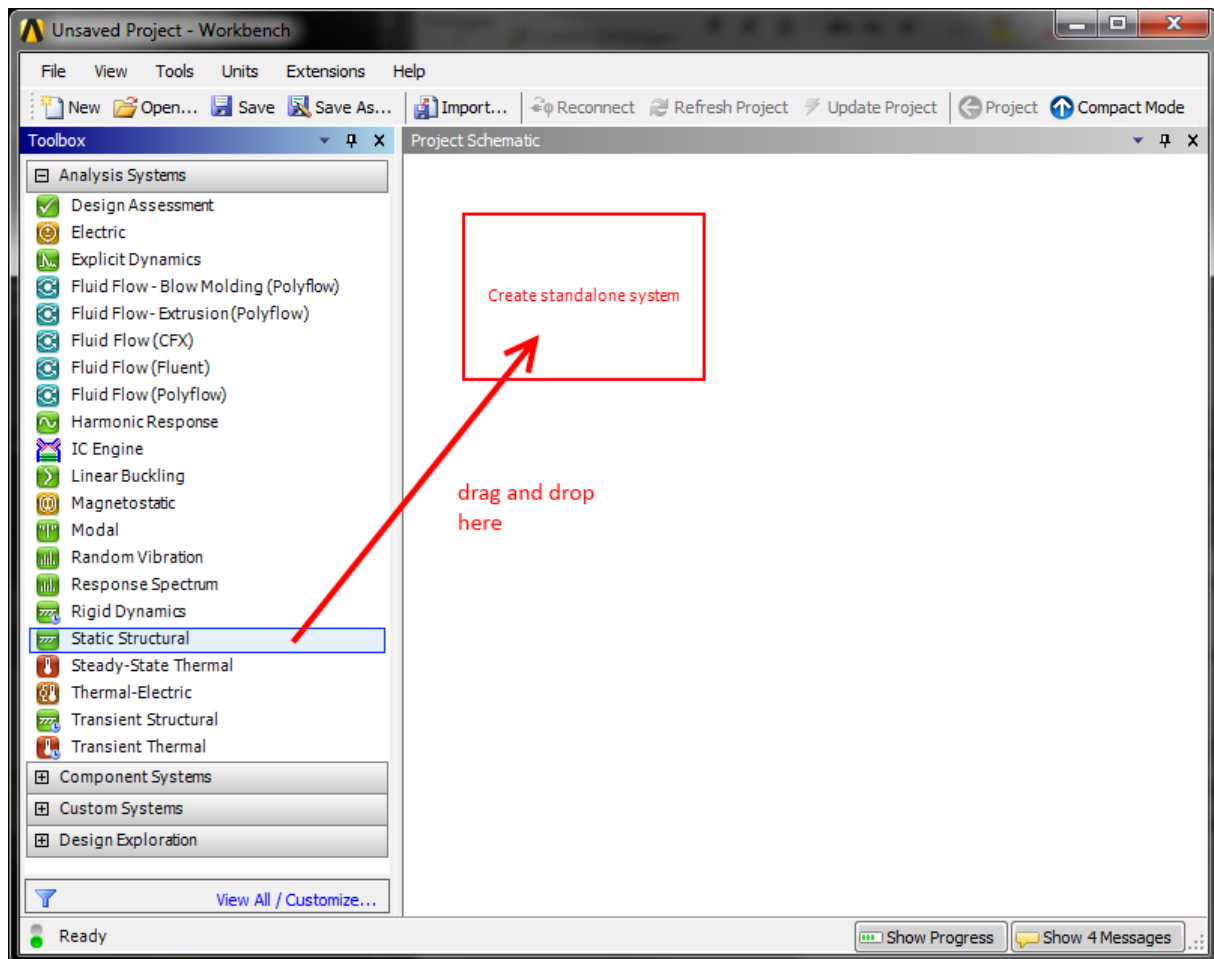

FIGURE 6: CREATING A NEW STATIC STRUCTURAL ANALYSIS SYSTEM

A.1   Defining the Geometry

In the newly created analysis system, double click the **Geometry** cell to start up the **DesignModeler** module; choose the desired units.

Create the solid beam by choosing (from the main menu) **Create -> Primitives -> Box**. Use the **Details** pane to specify the desired dimensions of the new primitive.

Please ensure that the origin of the coordinate system is located on the plane and at the center of the cross section of the beam. The beam's long axis must be oriented along the global *x*-axis (Figure 7).
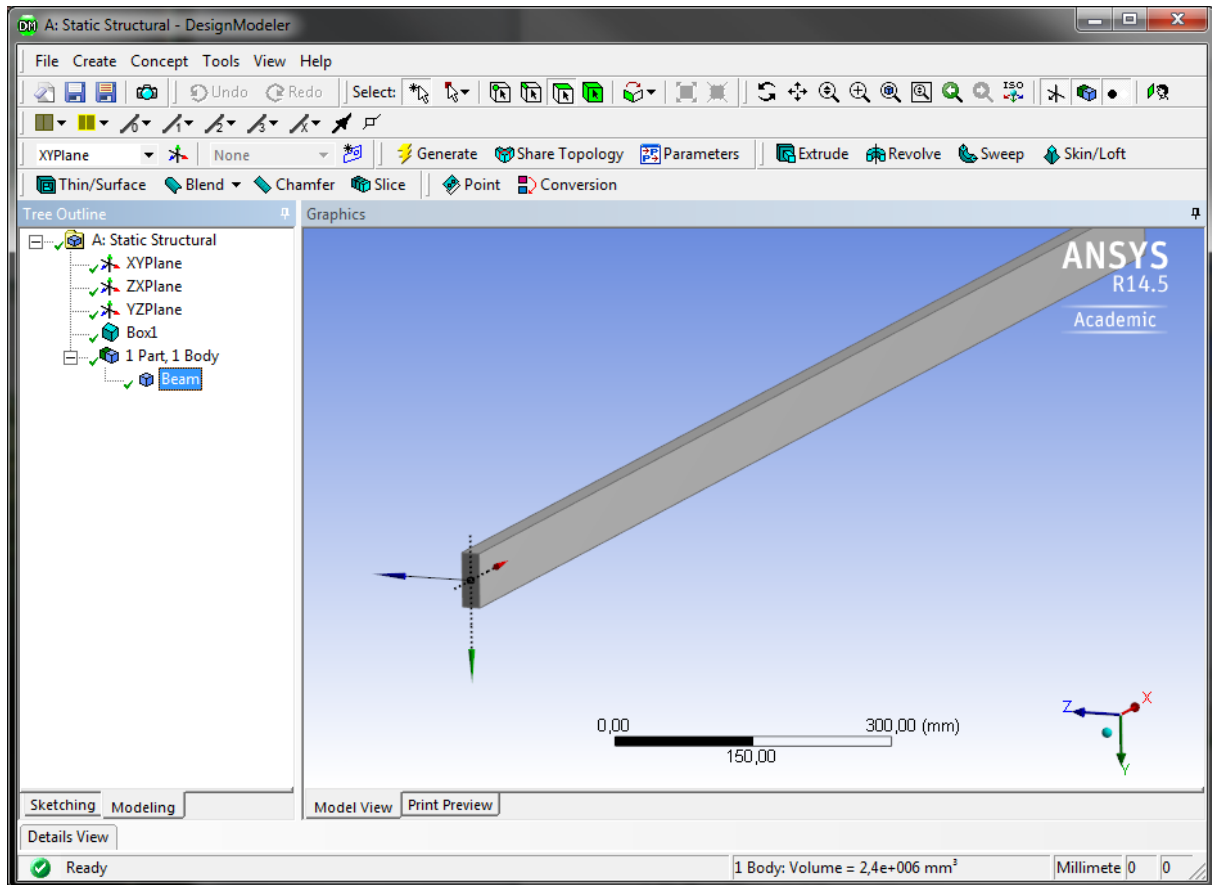

FIGURE 7: THE BEAM IN DESIGNMODELER

A.2   Material Properties

Material models define the mechanical behavior of the components of the FE model. We will use a simple linear-elastic and isotropic material model to represent the behavior of our steel beam.

In your static structural analysis system in the Workbench project view, double click the **Engineering Data** cell. This opens up a window titled **Outline of Schematic A2: Engineering Data**. "Structural steel" is the default material and is always predefined. Click the row beneath (where it says "Click here to add a new material") and enter any name for your new material.

In the **Toolbox** to the left, expand the **Linear Elastic** node and drag and drop **Isotropic Elasticity** onto the **Material** column of your material (Figure 8).

Enter the appropriate values into the **Properties** window (Young's modulus, Poisson's ratio), before clicking **Return to Project**.
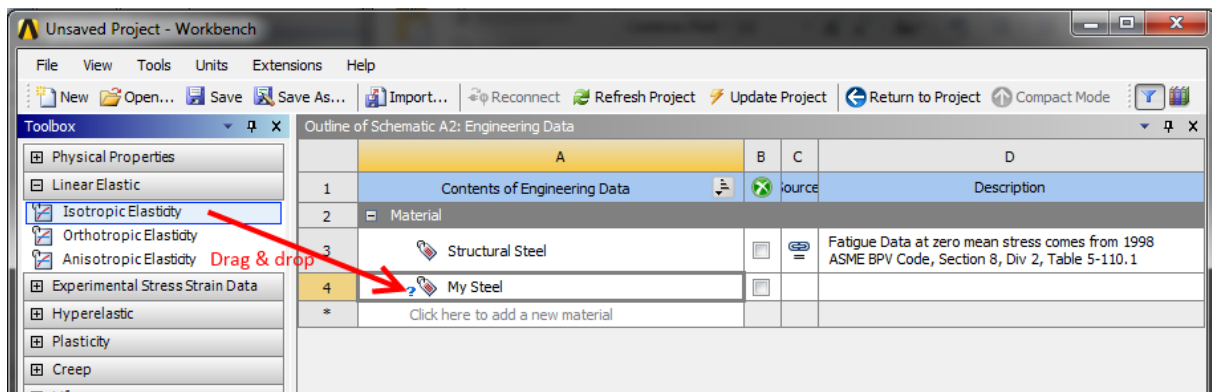

FIGURE 8: DEFINING A NEW LINEAR-ELASTIC MATERIAL

A.3   Meshing

The next pre-processing step is concerned with discretizing the continuous solid body geometry, also known as *meshing*.

In the project view, double click the **Model** cell to launch the **Mechanical** module. Your geometry should be imported automatically. Before meshing, make sure that the correct material model is assigned as well: In the Mechanical module's **Outline** pane (to the left) select the solid body representing the beam (under the **Geometry** node). Then select the material in the Details pane (**Details -> Material -> Assignment**).

The outline tree view also contains a node called **Mesh** with a little flash symbol. Right click on **Mesh** and select **Insert -> Mapped Face Meshing** from the context menu (Figure 9). Select all 6 faces of the beam and click **Scope -> Geometry -> Apply** in the details pane of the Mapped Face Meshing node.

In the same way, add a **Sizing** sub-node to the Mesh node. This time, select *the whole body* and again apply your selection. In the **Details** pane of the (Body) **Sizing** node, select **Definition -> Type -> Element Size** and set the element size to 15 mm. In the **Outline**, right click on **Mesh -> Generate Mesh**. The result should resemble Figure 10.
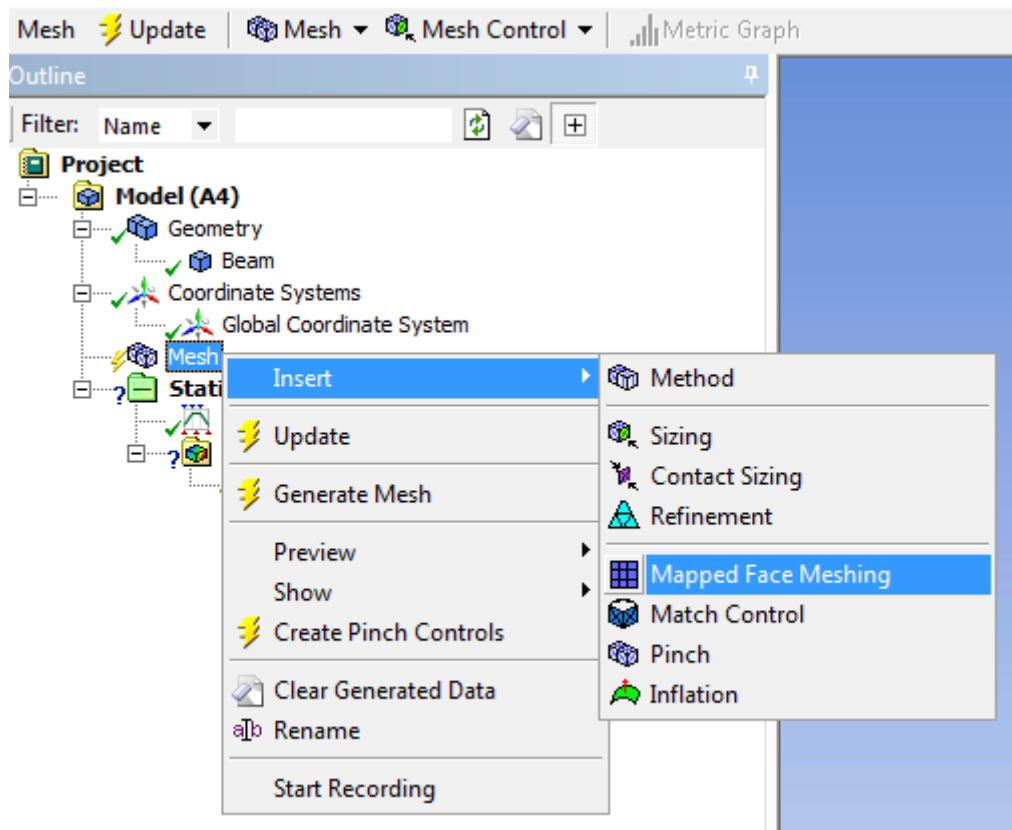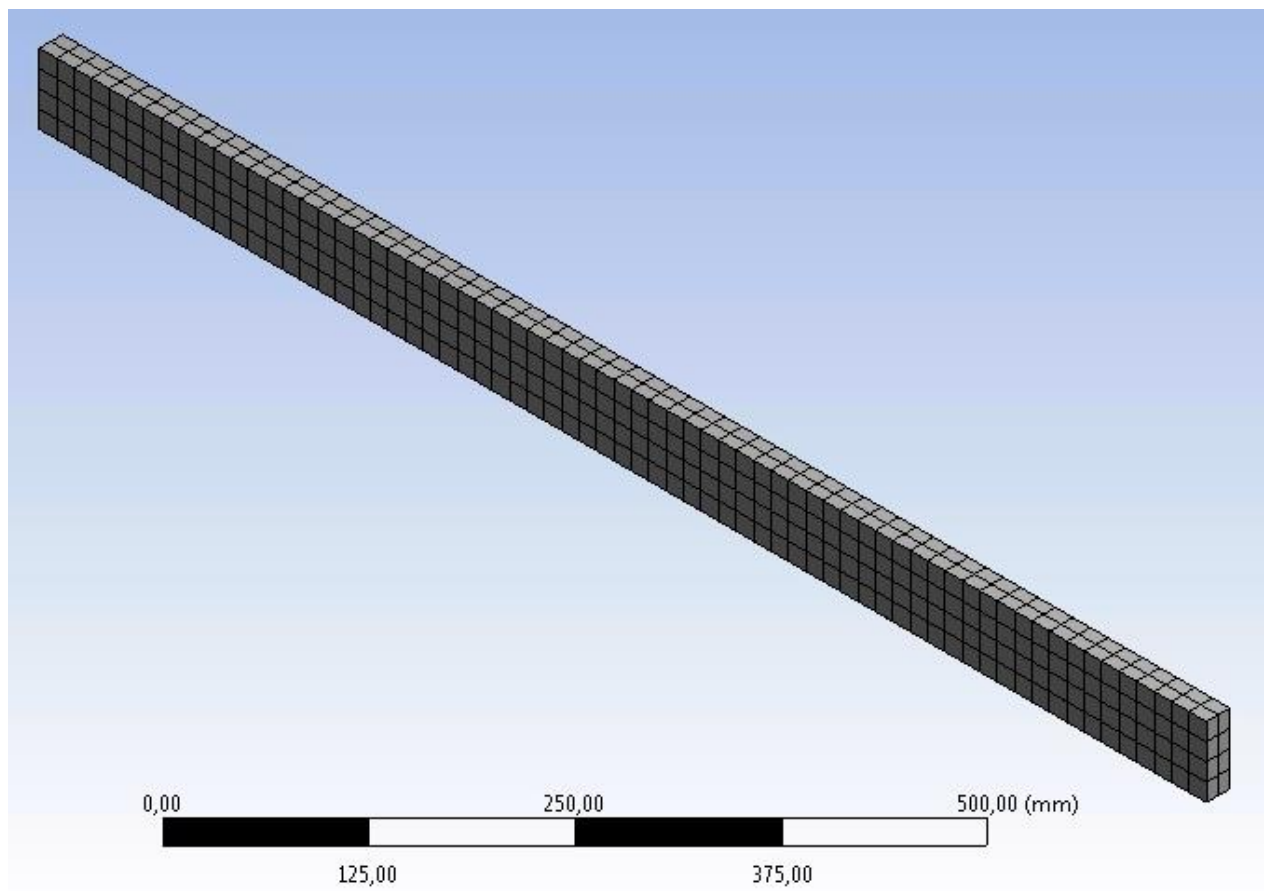
FIGURE 9: ADDING A MESHING METHOD

FIGURE 10: MESHED BEAM IN ANSYS MECHANICAL (536 ELEMENTS)

A.4   Applying Loads and Boundary Conditions

We now have to apply the loads and boundary conditions in such a way that the FE model represents our ideas from Figure 2.

We therefore fix all degrees of freedom of one end. In the **Outline** right click on the **Static Structural (A5)** node and select **Insert -> Fixed Support** from the context menu. Select an appropriate face of your solid body to be fixed.

Next, we need to apply the force to the other end of the beam. Again, right click on the **Static Structural (A5)** node, but this time **Insert -> Force.** Select the correct face and apply a force of the appropriate magnitude and direction. The result should resemble Figure 11.
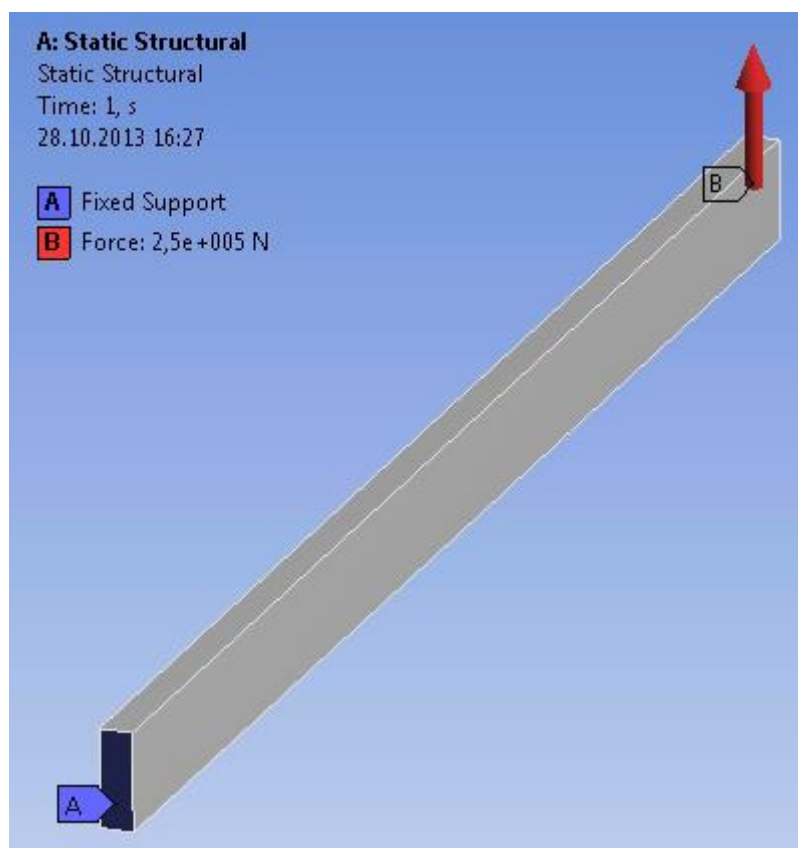

FIGURE 11: THE MODEL AFTER APPLYING LOADS AND BOUNDARY CONDITIONS

B.   Solving

Because this is a simple linear problem, we do not need to modify the solver options manually (**Analysis Settings** in the **Outline**). Instead, simply right click on the **Static Structural (A5)** node and select **Solve**. This will bring up a status window, which should disappear again after a few seconds of computing.

C.   Post-Processing: Evaluating the Solution

The primary results of an FEA are nodal displacements. Strains and stresses are computed on demand as a post-processing step based on the determined displacement field.

C.1   Specifying Result Items

Up until now, ANSYS only offers the solver log under **Solution (A6) -> Solution Information**. To visualize the results we are interested in, add the following Items to the Solution (A6) node (Figure 12):

- Total Deformation
- Normal Elastic Strain
- Normal Total Stress
- Equivalent (von Mises) Stress

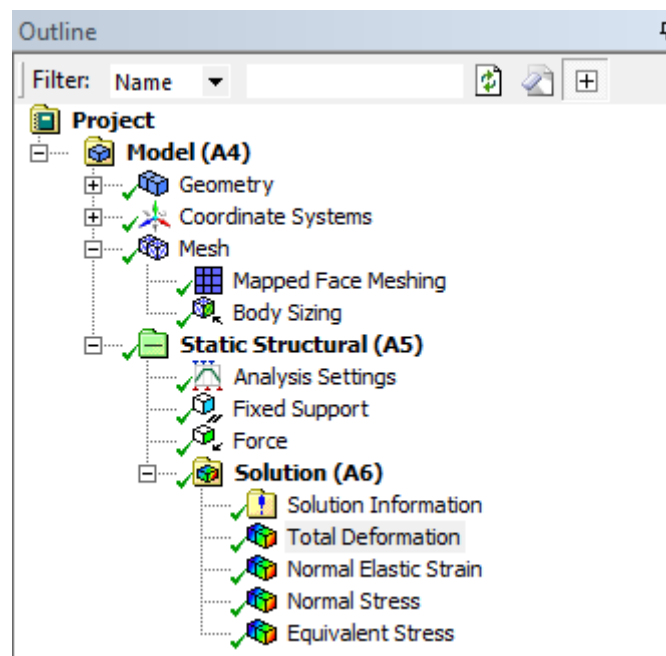Right click on **Solution (A6)** and select **Evaluate All Results**.

FIGURE 12: ANALYSIS OUTLINE WITH ADDED POST-PROCESSING ITEMS

C.2   Total Deformation

When performing FE analyses, it is always wise to first perform some plausibility checking. Create a contour plot of the total deformation (Figure 13).
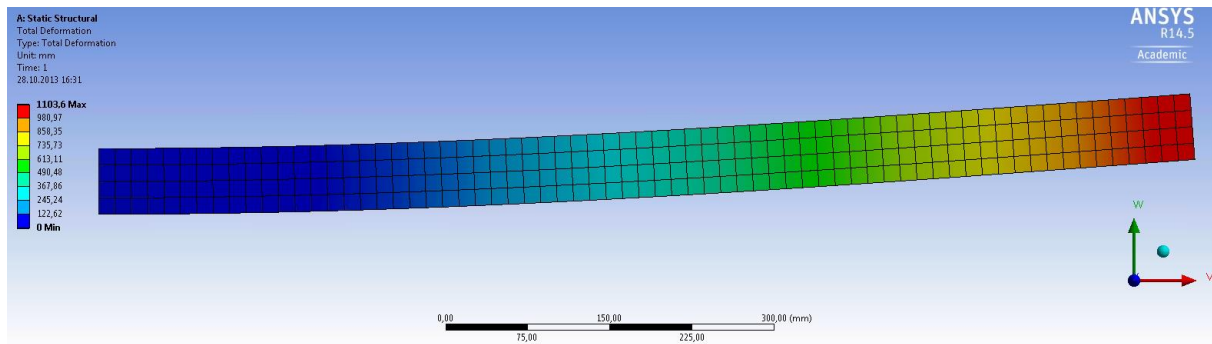

FIGURE 13: (SCALED) CONTOUR PLOT OF THE TOTAL DISPLACEMENTS.

The predicted displacements seem to be totally fine at first glance. On closer look the maximum total displacement is more than 1.0 m, according to the scale to the left!

The issue with this plot is that by default ANSYS automatically scales the displayed deformations so that they are "easily visible." For very small displacements this behavior is totally fine, as they wouldn't be visible at all otherwise. In our case, however, this setting is deceptive. Changing the scaling factor to 1.0 (**Results** toolbar) yields a completely different picture, making it crystal clear that something has gone wrong – awfully wrong:
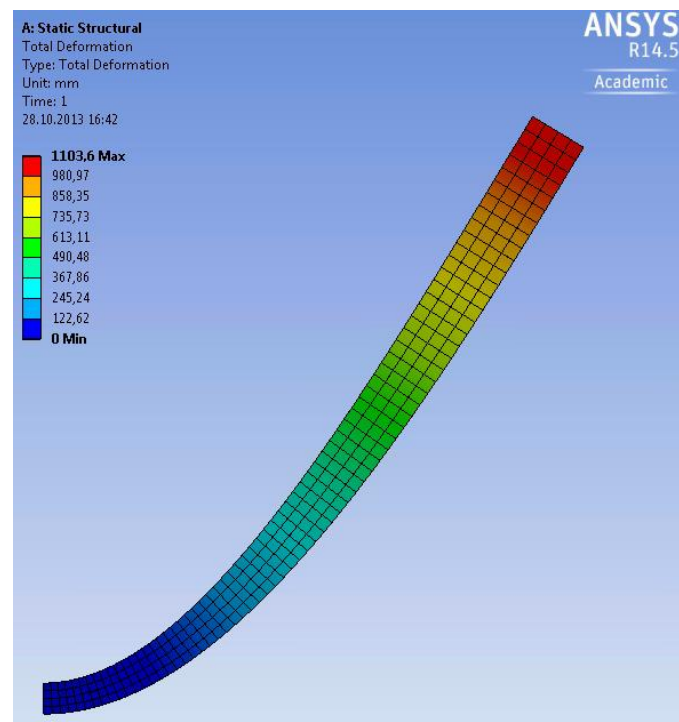

FIGURE 14: (UNSCALED) CONTOUR PLOT OF THE TOTAL DISPLACEMENTS

The reason for this huge displacement is that in Table 1, we (deliberately) assumed a much too high force. If we correct the force to be $F = 5000$ N instead, we get the following (unscaled) deformation plot:
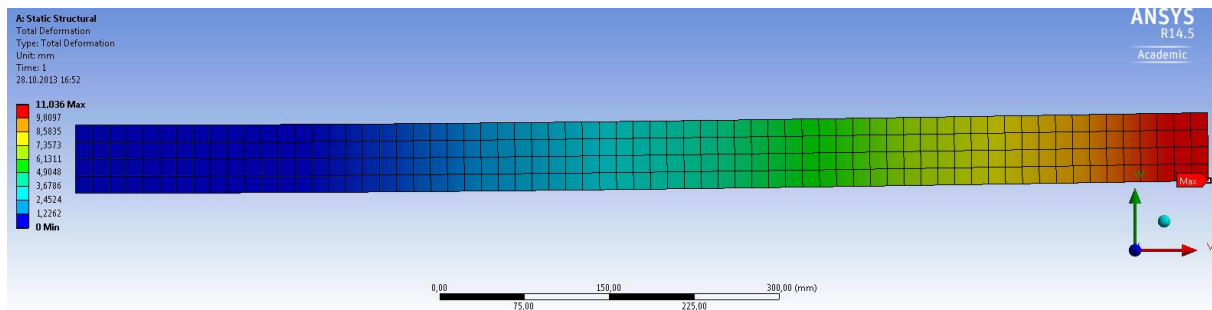

FIGURE 15: (UNSCALED) CONTOUR PLOT OF THE TOTAL DEFORMATIONS AFTER APPLYING THE CORRECT LOAD

C.3   Visualizing Strains & Stresses

Now that we have corrected our model, we can try to answer the question, whether the beam will be able to resist the given load or if it will fail. For that, create contour plots of the component strain and stress along the *x*-axis to investigate tensile and compressive stresses (Figure 16, Figure 17). For ductile materials like steel, the von Mises yield criterion can be used to predict, whether the material is likely to deform plastically. We therefore also plot the von Mises (equivalent) stresses (Figure 18).
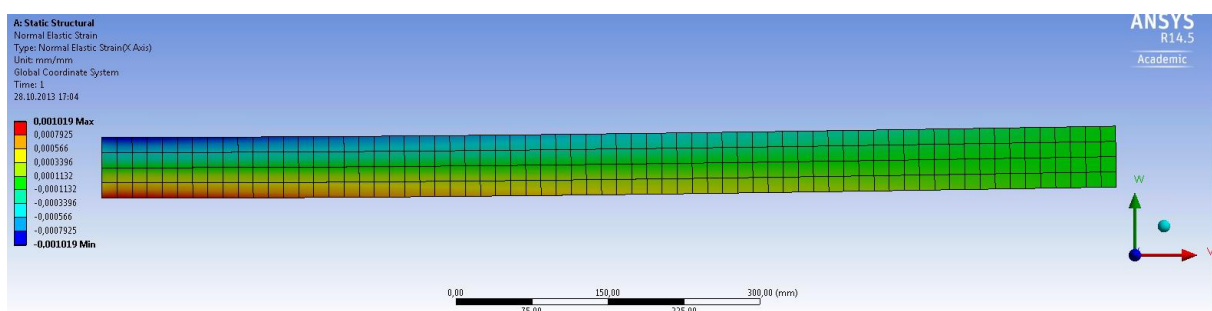

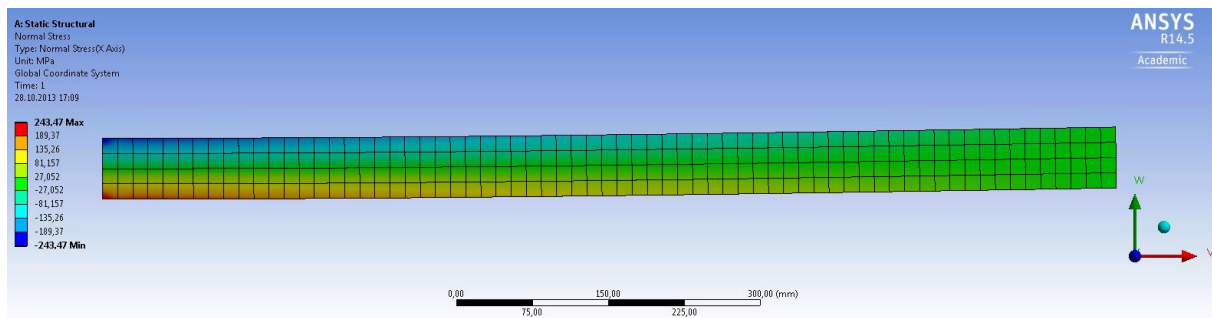FIGURE 16: ELASTIC NORMAL STRAIN ALONG *X*
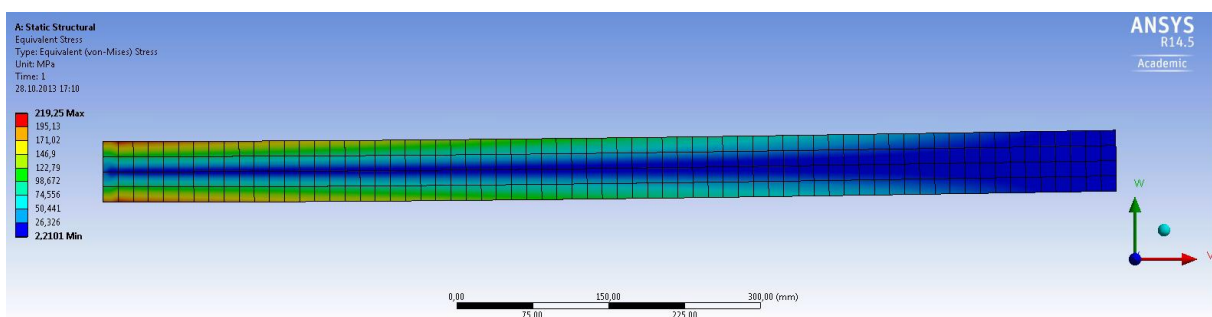
FIGURE 17: ELASTIC NORMAL STRESS ALONG *X*

FIGURE 18: VON MISES STRESS

Answering the Questions:

1 Will the beam break? Where would it fail?

With the corrected force (F = 5,000 N) the beam will not break. The maximal predicted von Mises stress reaches values of $\sigma_{\mathrm{pred}} \approx 220$ N/mm^2, and thus less than the ultimate yield stress of $\sigma_{\mathrm{yield}} = 235$ N/mm^2. That means the failure criterion $\sigma_{\mathrm{pred}} > \sigma_{\mathrm{yield}}$ is not fulfilled. The difference between the two values however is small (σ_{pred} reaches 94 % of σ_{yield}). Many technical applications require a safety factor (SF) of 2.0 or higher. In our example, the safety factor SF $= \sigma_{\mathrm{yield}}/\sigma_{\mathrm{pred}}$ is *much* smaller.

The critical region where we would expect the beam to start failing is located at the left end of the half beam, at the location of maximum stresses. For the full length beam the critical region would therefore be located in the middle of the beam where the force was applied.

2 Assuming that it will not fail, what would be the maximum deflection *w*?

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

D.   Deriving the Solution Analytically

For this cantilever problem we can also derive our solution analytically; we can calculate the maximum deformation, as well as maximum material stress. The formulae you need can be found in the accompanying MMSM1 notes on page 37 and 38.

D.   Deriving the Solution Analytically

E. BONUS 1: Thermal Loads

Now we can look at a somewhat more complicated geometry with a different set of loads.

E.1 Modelling a simple bimetallic strip

A bimetallic strip is a beam consisting of two different metallic materials. These beams are usually welded together lengthwise, and are found in devices such as toasters. Create a Workbench Model for a bimetallic strip that exhibits the following properties (material properties for Aluminum and Steel are available through the ANSYS Material Databank):
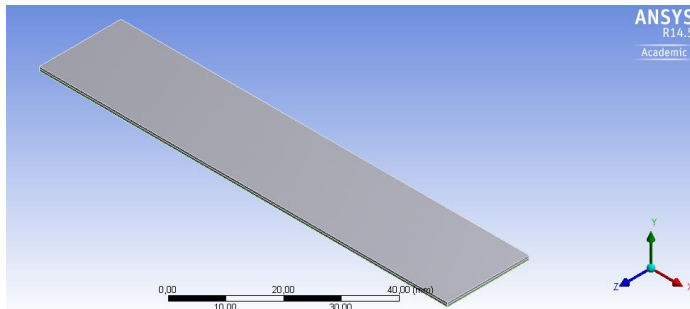
Length	l = 100 mm
Height	h = 1 mm
Width	w = 20 mm
Material 1	Aluminum
Material 2	Structural Steel
ΔT	100 K

E.2 Tasks

Try to answer the following questions and complete the tasks.

- a. What happens to the strip under a thermal load?
- b. What does the internal stress along the X-axis look like? Where do you suspect is it the greatest (apart from the fixture)?
- c. Create this model in ANSYS and test your theories. A thermal load can be assigned by choosing **Load -> Thermal Condition.**
- d. We want to increase the displacement of the strip. What changes can we make to reach this result?
- e. Are there changes other than material, length (l) or width (w) which will result in larger displacements?
- f. How large is the force exerted by the strip at its end? (Tip: Fix the other end of the strip in Y-direction)

F. BONUS 2: Analogous Model in Lower Dimensions

It is possible to create the same model in a simplified, lower dimensional form. Let's look at how this is done. Create a new **Static Structural** Workbench project.

F.1 2-Dimensional with Plane Stress

In order to create a 2-Dimensional version of our previous analysis, we must first tell ANSYS to restrict itself to only two dimensions. This is done by accessing the properties in the Geometry section of our model. Right click on the **Geometry** section, and change the dimensionality as shown:
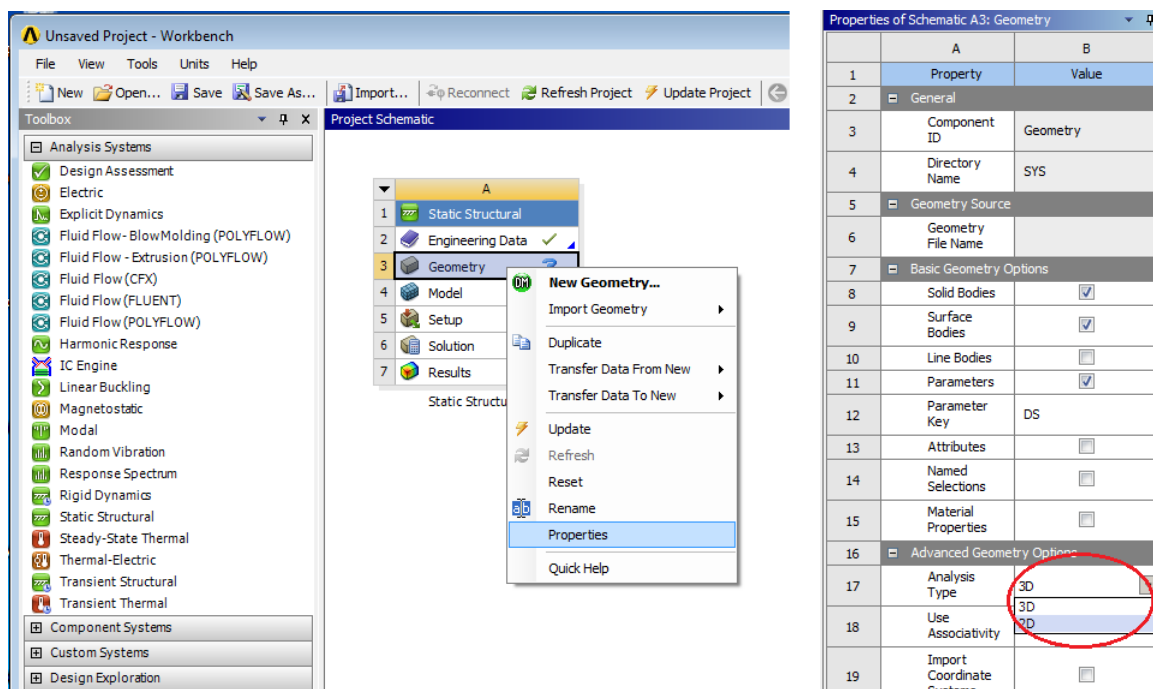

FIGURE 19 CHANGE MODELLING TO 2D

Ensure that your material properties are defined as in the previous section, by double clicking on **Material Properties** and entering the parameters; then start the **Geometry** section. Now, we need to sketch a rectangle using the ANSYS CAD-feature. First, create a **Sketch**. You can find the command as a tab on the left hand side of your window. Select the **Rectangle** shape and place it on the X-Y plane. Then, change to the **Dimension** subsection, and assign the correct values to your rectangle.
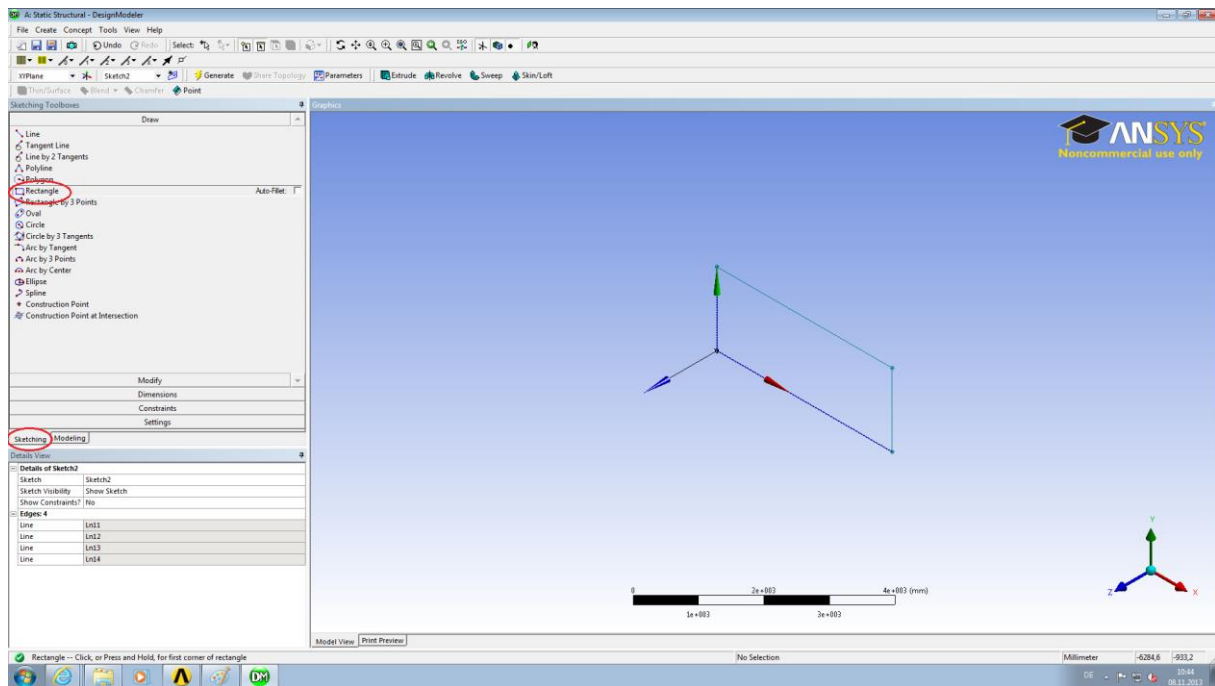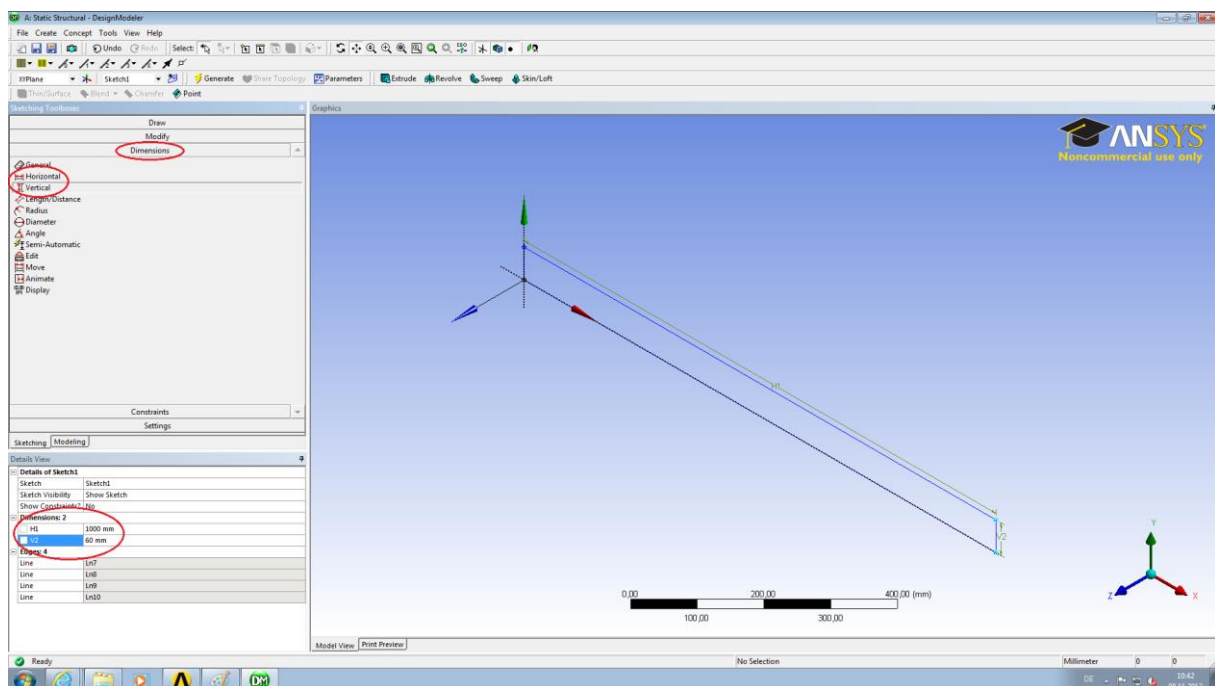
FIGURE 20 CREATING A RECTANGLE SKETCH

FIGURE 21 ASSIGNING DIMENSIONS TO RECTANGLE

Create the Surface Body by selecting **Concept -> Surfaces From Sketches** and highlighting your sketch. Generate the body, and, in the Surface Body detail view, change the **Thickness** appropriately. Close the Geometry window and open Model window. It is important to perform our analysis as **Plane Stress**. Ensure this is selected under as **2D Behavior** in the Geometry detail view.
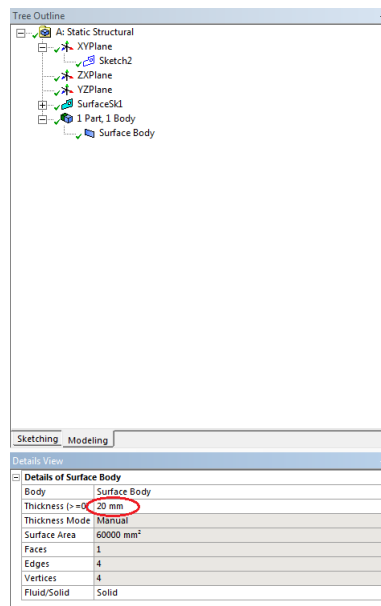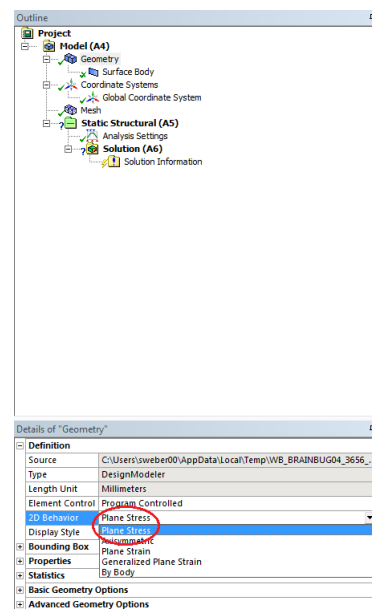
FIGURE 22 THICKNESS ASSIGNMENT

FIGURE 23 PLANE STRESS ASSIGNMENT

Complete the model with appropriate boundary conditions and evaluations. How does it compare to our 3D model?

F.2   1-Dimensional Line Object

It is also possible to create this model as a 1-dimensional entity. Once again, start a new Workbench project. As before we define our material properties, and enter the Geometry section. This time we don't have to switch the Geometry Properties; instead we want to create a Line Segment from points. Choose **Create -> Point**, and, in the detail view, select Manual Input in the Definition section. Clicking **Generate** will create the point. Place the first point at the origin, and create another at (1000, 0, 0).
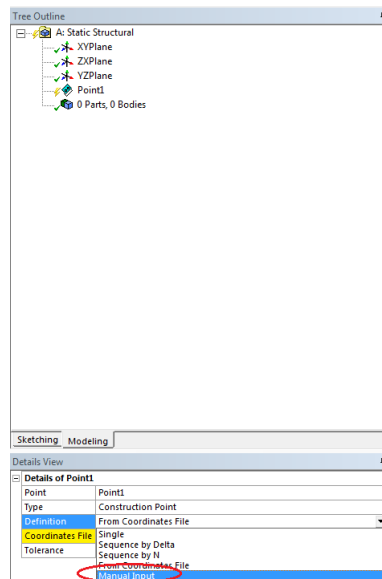
FIGURE 24 CREATING A POINT

Choose **Concept -> Line From Points** and select both your points (you can select multiple entities by holding down Ctrl), and press **Generate** once more. Now we have a body, but no way to tell the program about its thickness or height. Remedy this by choosing **Concept -> Cross Section -> Rectangular**, and entering the appropriate values. Assign the newly created Cross Section to your Line Body by clicking on the Line Body and making the assignment in the detail view.
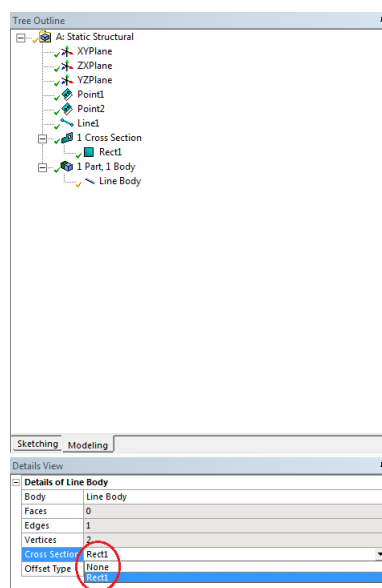

FIGURE 25 CROSS SECTION ASSIGNMENT

Complete your analysis as in the previous sections. Chances are, your answer is wrong; the reason is simple, though not obvious.

You may have noticed that, once we generated our Line Body, that there is a strange vector pointing out from it. This vector is used to determine the alignment of our Cross Section. Open the **Geometry** section, right click on **Line Body** and choose **Select Unaligned Line Edges**. Choose **Vector** in the Alignment Mode section, and enter a vector such that the Y-Axis alignment of your Cross Section corresponds with the direction of your vector. That is, if you entered a

height H1 of 60mm for your cross section, you must define a vector of (0, 1, 0) in the Alignment Mode.
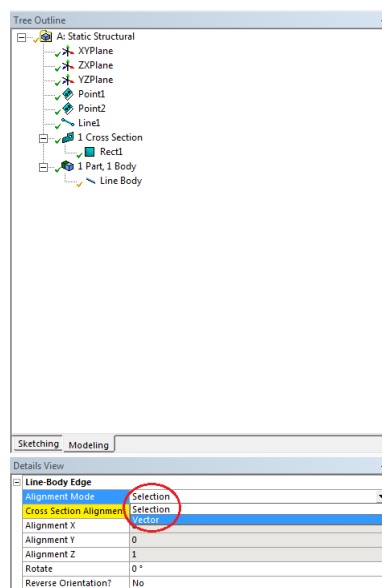

FIGURE 26 CROSS SECTION ALIGNMENT

You can verify whether your alignment was correct by entering the Model viewer and choosing **View -> Cross Section Solids**. Your answer should now coincide with our previous calculations.

G.   Additional Notes on Sketching

It is, perhaps, helpful to look directly on to the body you are constructing. You can easily do this in the Design Modeler; simply have a body or sketch highlighted, and press the **Look At Face/Plane/Sketch** button found in the upper right under the toolbar.
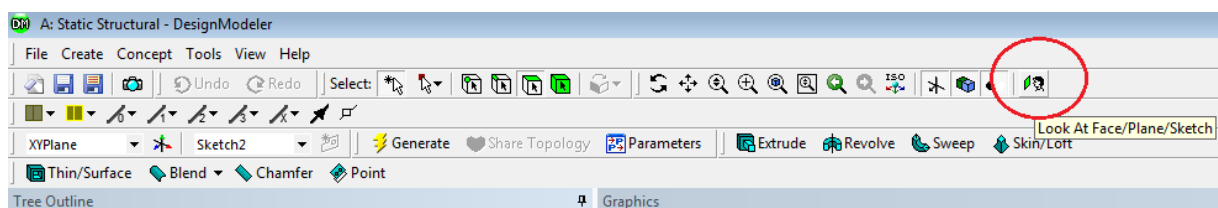

FIGURE 27 THE LOOK AT BUTTON

It is also possible to enable grid lines in Workbench. In the sketching tab, select **Settings**. Here you will find several useful options, including snap-to-grid and the overall grid sizes.
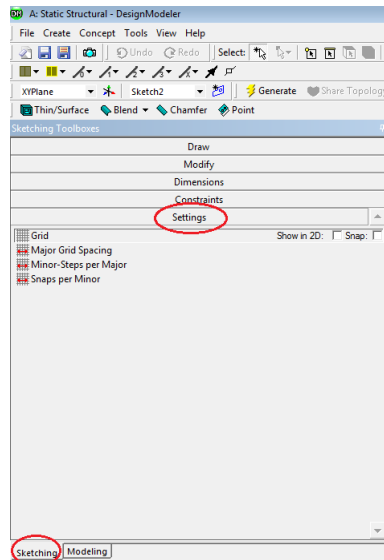
FIGURE 28 ENABLING SKETCHING GRID

Finally, you can enable displaying the values of dimension axes in the dimension section. It is often more helpful seeing the dimension value, rather than an arbitrarily assigned name variable.
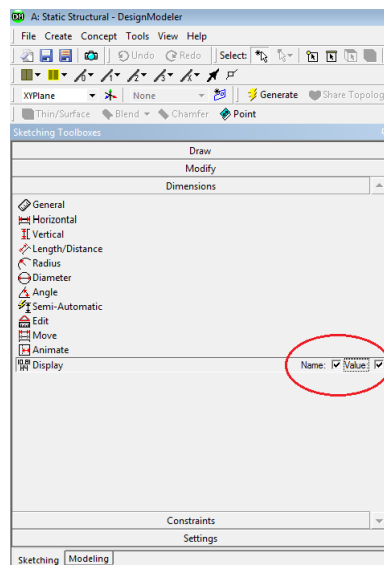

FIGURE 29 ADDING DIMENSION VALUES