Virtual Prototyping

Introduction to the Finite Element Method

Learning objectives:

1. Understand the basics of simulation using the finite element method (FEM) in NX Simcenter.
2. Learn to perform static analyses using one-dimensional (1D) elements.
3. Explore the applications of models using one-dimensional elements.

1. Simulation workflow

To perform a finite element simulation, complete the following steps:

1. Prepare the geometry of the structure.
2. Discretise the model by creating a finite element mesh.
3. Assign the required properties to the elements, such as the beam cross-section geometry in a 1D model or the thickness of thin-walled elements.
4. Define the supports by constraining the appropriate degrees of freedom.
5. Define the loads acting on the structure, such as a concentrated force.
6. Run the analysis. Use the Nastran solver for this exercise.

Once the analysis is complete, you can display results such as displacements and stresses. Each stage can be carried out in several ways. The simple beam example below introduces the basic functions of the NX interface.

Example data:

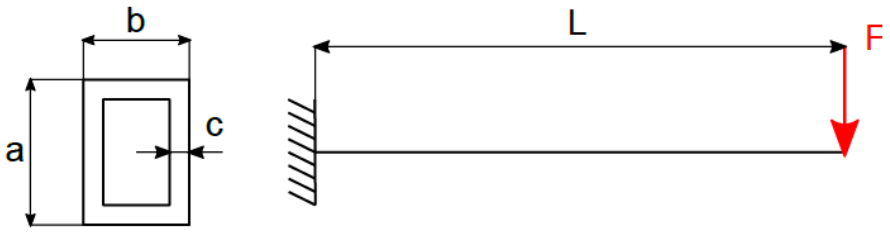
a = 60 mm

b = 50 mm

c = 6 mm

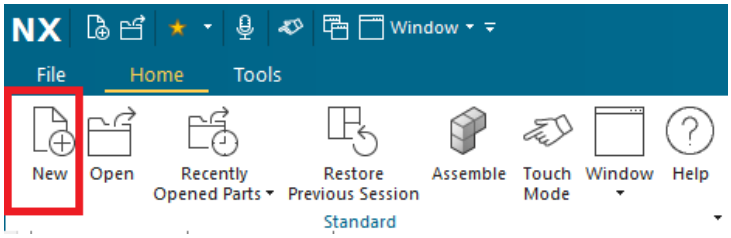
L = 1000 mm

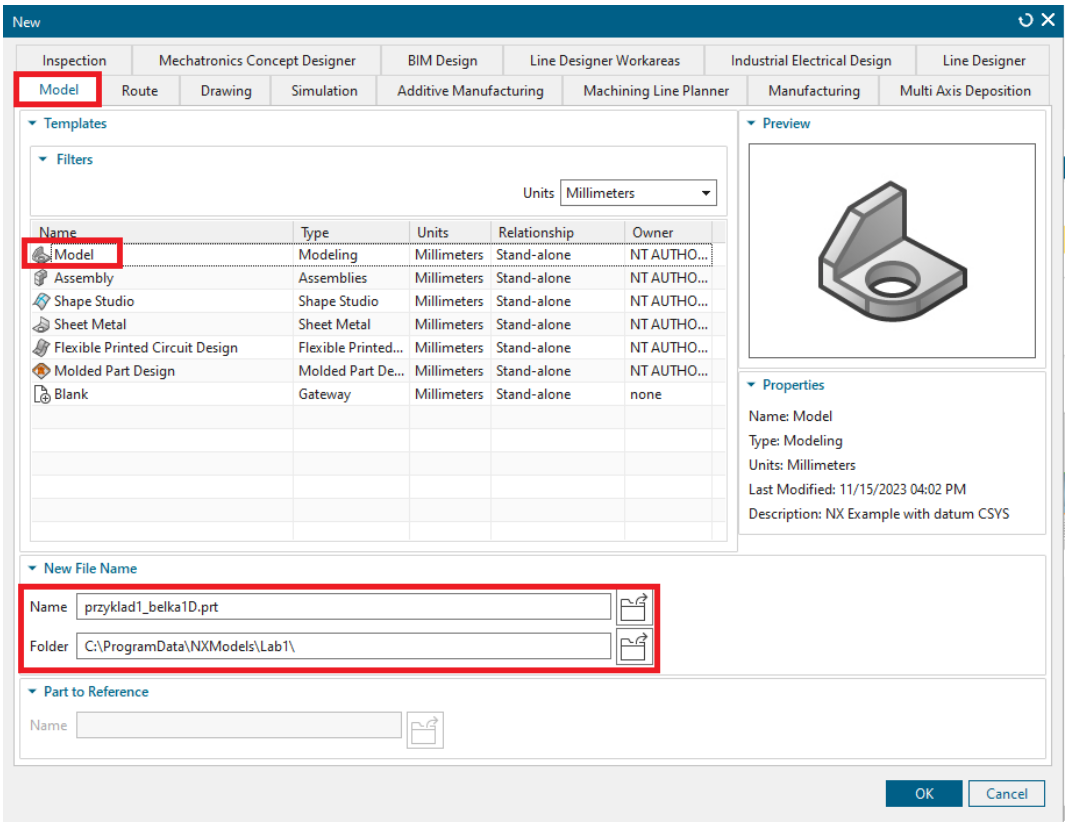
F = 1000 N



a) Creating the geometry

Create a new part file. Choose a save location and enter a file name.



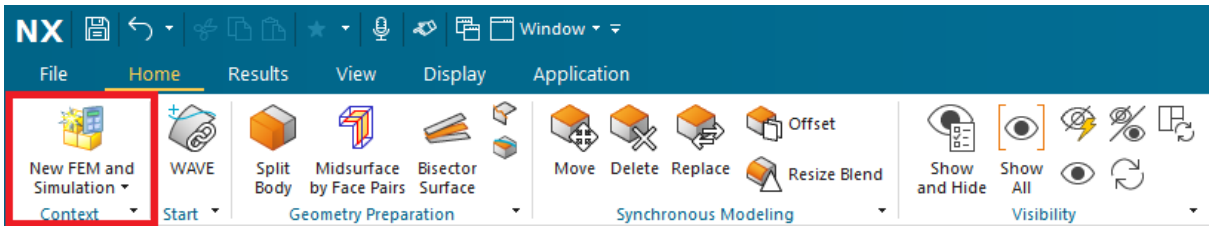


Create a sketch and draw a line segment of length L = 1000 mm. This geometry is sufficient to analyse a simple beam using 1D elements. In the next stage, you will create the finite element model and simulation files.

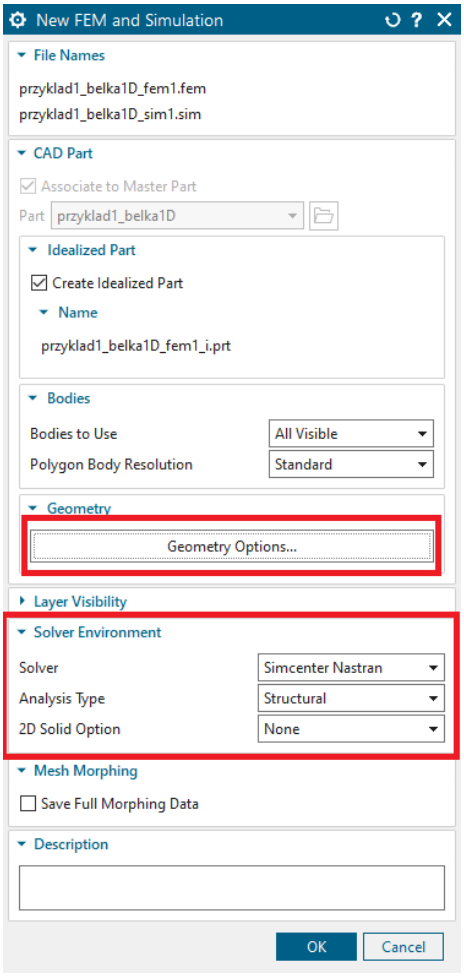
b) Discretising the model

Start with the part file containing the line segment open.

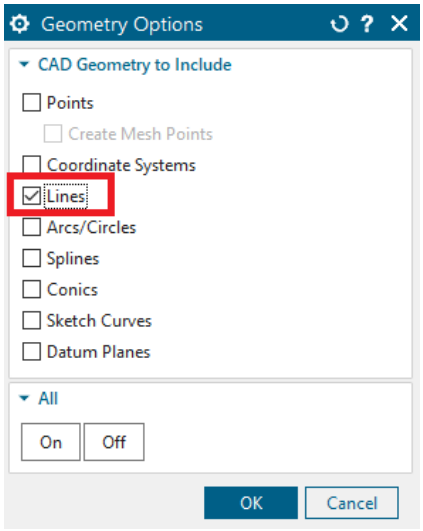
Select ***File → Pre/Post*** in the upper-left corner of the window, or go to the ***Application*** tab and select ***Pre/Post***. The simulation preparation and analysis environment will open, and the interface layout will change.



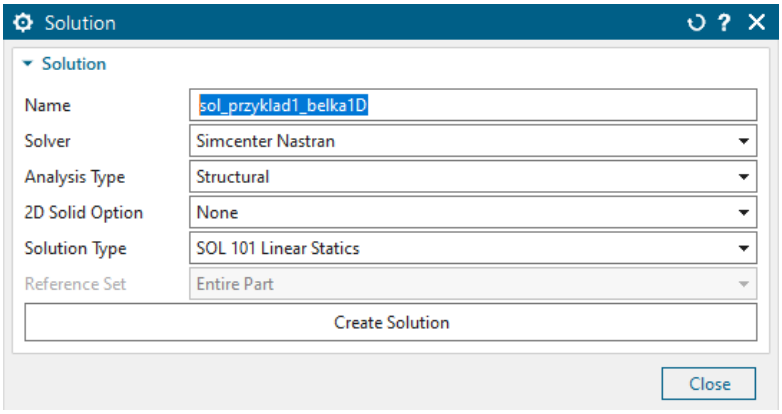
Select ***New FEM and Simulation***. A dialog will open with options for importing the geometry and selecting the solver.



Under ***Geometry Options***, select ***Lines*** because the geometry consists of a single line segment.



Confirm the settings using the ***OK*** buttons. The ***Solution*** window will open, with a smaller solver settings dialog in front of it.

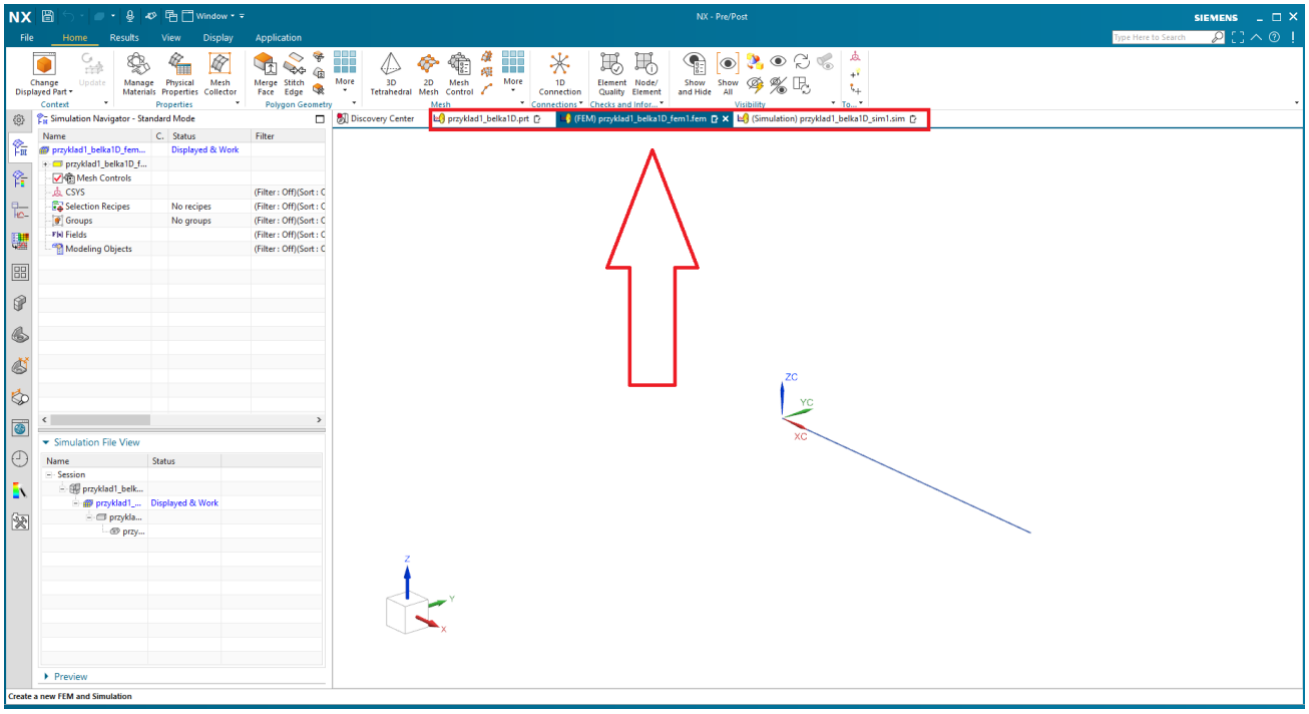


If necessary, change the solution name, then click ***Create Solution***.

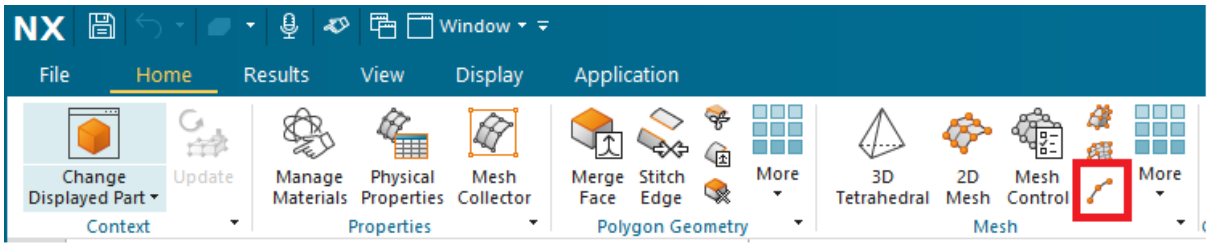
The open-file tab bar contains three files:

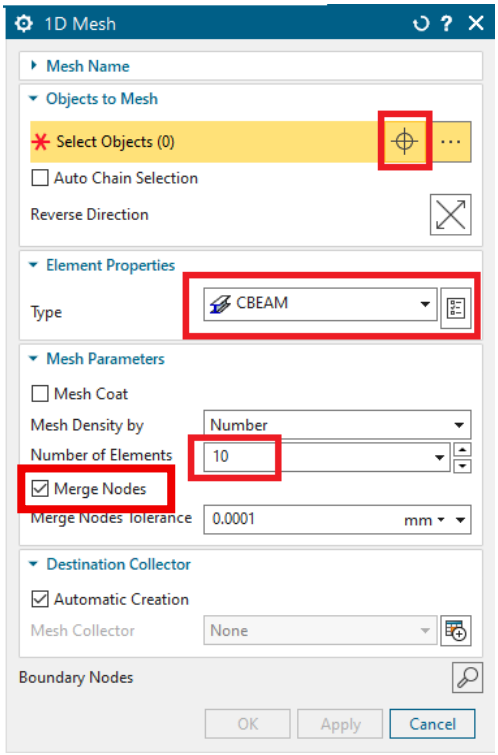
- the **.prt** part file, which contains the geometry;
- the **.fem** file, where you create the finite element mesh and assign element properties;
- the **.sim** file, where you define supports and loads and run the analysis.

Switching the active file changes the commands available on the top toolbar and the contents of the tree on the left side of the screen.



Switch to the **.fem** file. On the top toolbar, select the command for creating a ***1D*** mesh.





In the new dialog, click the crosshair button and select the line segment. The selection counter should change from ***Selected Objects (0)*** to ***Selected Objects (1)***.

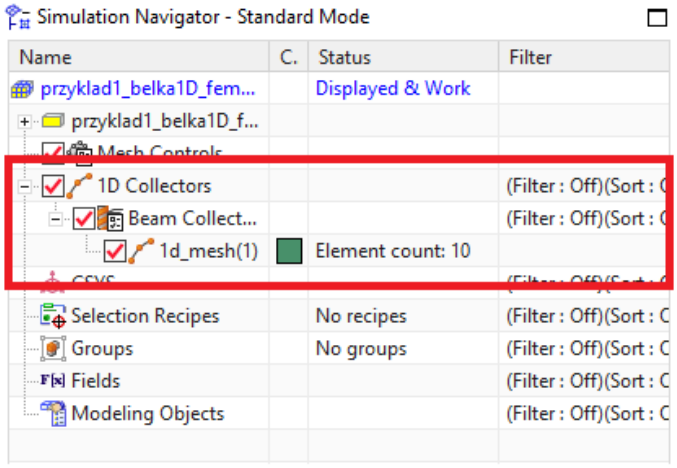
Select the ***CBEAM*** element type and set the number of elements to ***10***. Click ***OK*** to confirm the settings.

A new expandable item will appear in the tree on the left (image **1** below).

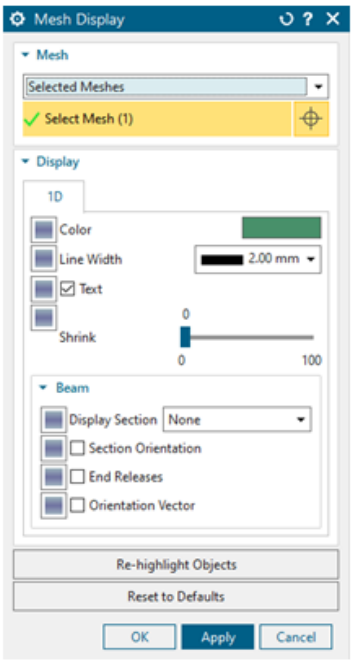
The mesh has been created, but the individual elements may not be visible. To change the display settings, right-click on ***1d_mesh(1)*** and select ***Edit Display***. The dialog shown in image **2** will open.

The ***Text*** option displays text next to each element. Setting ***Shrink*** to a non-zero value displays gaps between the elements.

1



2

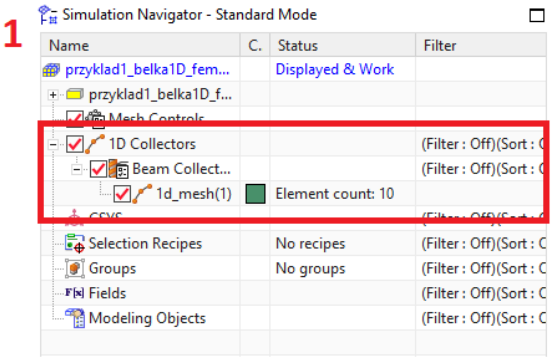


In the next stage, you will assign cross-section and material properties to the elements.

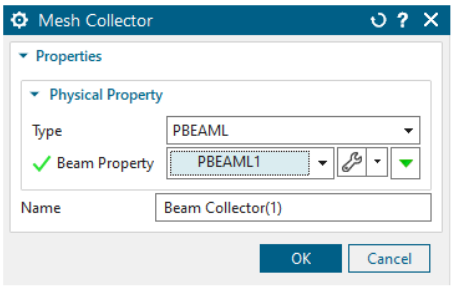
c) Assigning cross-section and material properties

In the tree on the left, double-click ***Beam Collector (1)***, as shown in image **1** below. The beam property selection dialog will open. The program has automatically created a property named ***PBEAML1*** (image **2**).

1



2



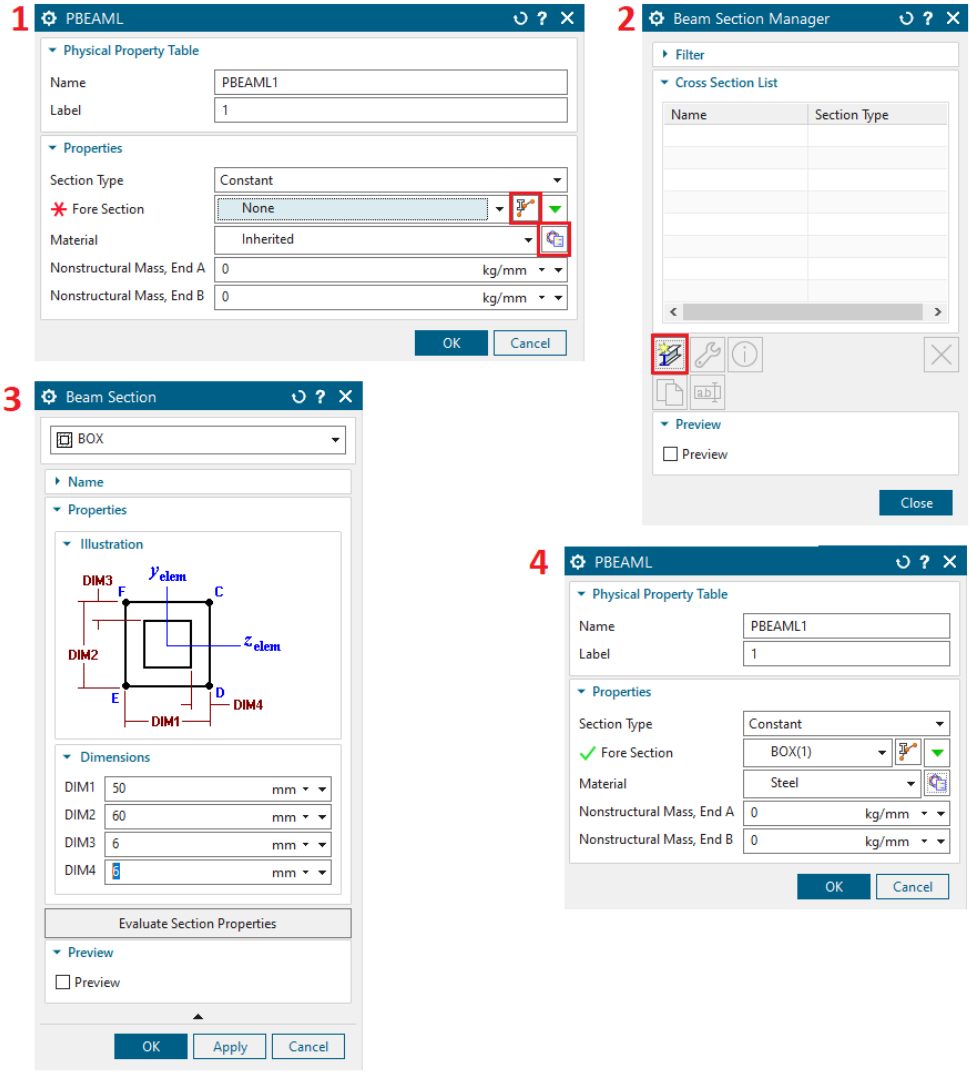
1. Click the wrench icon (***Edit***). Some property settings have not yet been specified (image **1** below). Use a constant beam cross-section for this example (***Section Type***). The ***Fore Section*** field is currently empty.

2. Click the icon highlighted in red to open the cross-section selection dialog (image 2). Then click the button for creating a new cross-section. The dialog shown in image **3** will open.

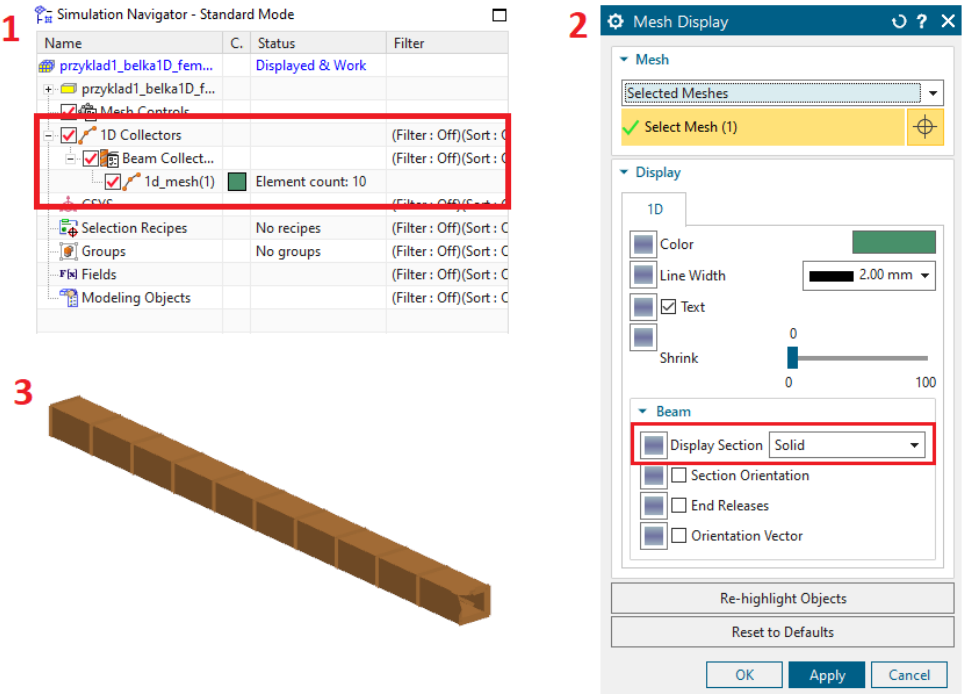
3. Select the ***BOX*** cross-section type. Set the width to 50 mm, the height to 60 mm and the wall thickness to 6 mm.

4. Confirm the cross-section and return to the properties dialog. Select steel (***Steel***) from the material library.

5. Compare your settings with image **4**, then click ***OK*** to confirm.



After assigning properties to the finite elements, display the beam in 3D. Check the cross-section orientation so that you can define the supports and load direction correctly.



d) Adding supports

Each node in a 1D beam model has six degrees of freedom, labelled **DOF1-DOF6** in NX. DOF1, DOF2 and DOF3 correspond to translations along the x, y and z axes, while DOF4, DOF5 and DOF6 correspond to rotations about these axes.

Supports constrain selected degrees of freedom. In this example, the beam is fixed at one end, so all six degrees of freedom must be constrained at the supported node. Other support arrangements will be used in the subsequent exercises.

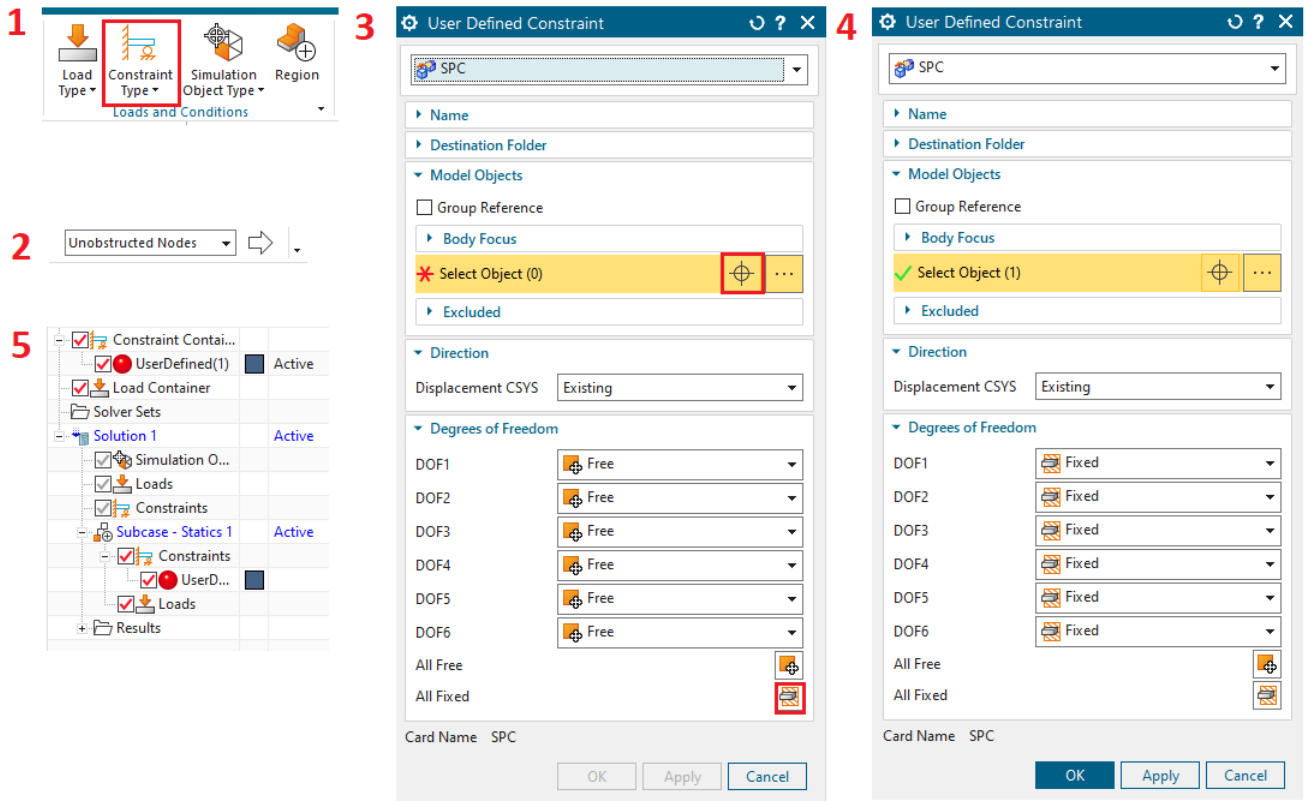
1. Switch to the **.sim** file. On the top toolbar, find ***Loads and Conditions*** and select ***Constraint Type → User Defined Constraint*** (image **1**). The support definition dialog will open (image **3**).

2. On the selection bar above the model area, select ***Unobstructed Nodes*** (image **2**). This makes it easier to select the mesh nodes to which the support will be assigned.

3. In the support dialog, click the selection button in the ***Select Object*** row and select the node at one end of the beam.

4. Click ***All Fixed*** at the bottom of the dialog to constrain all degrees of freedom of the selected node. Compare your settings with image **4**.

5. Click OK to confirm the support. The new support will appear in the tree on the left (image **5**).



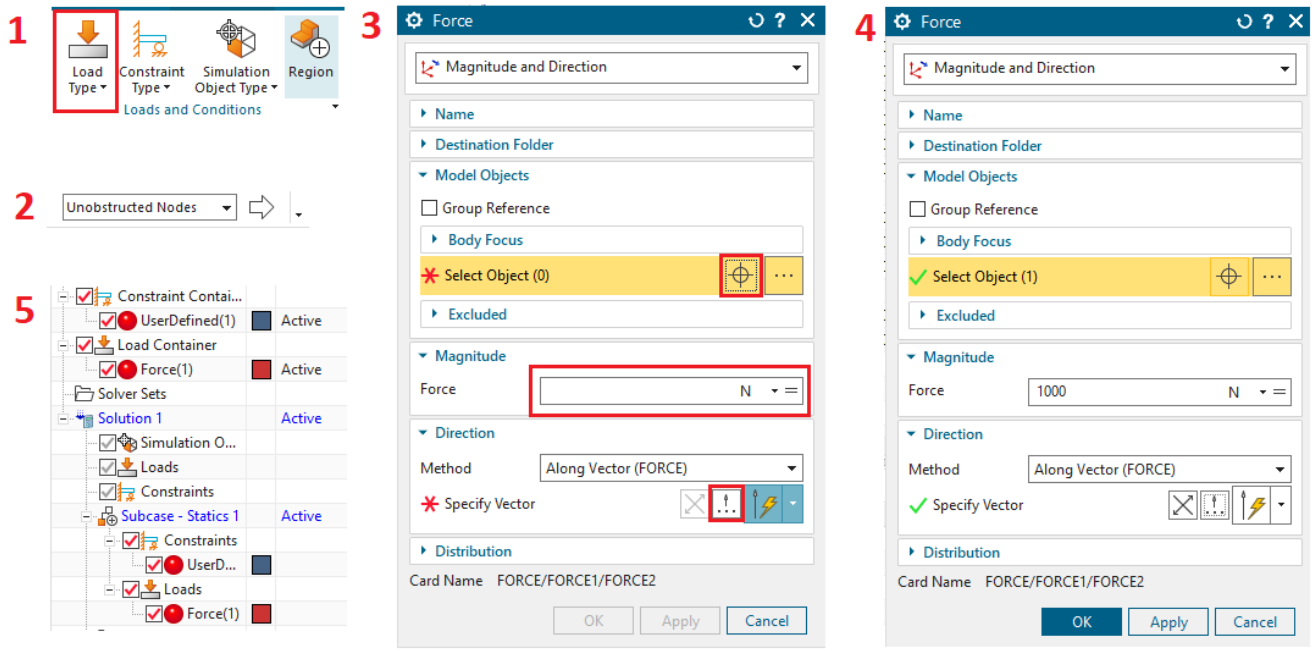
e) Adding a load

1. Under ***Loads and Conditions***, select ***Load Type → Force***. If necessary, set the node selection filter again (image **2**).

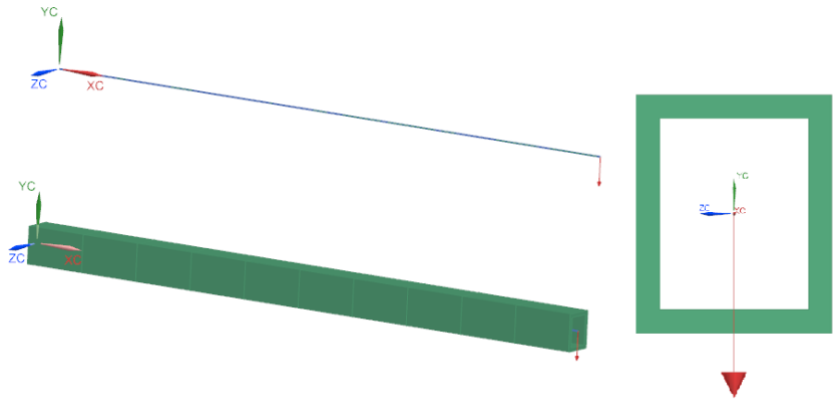
2. In the force definition dialog, select the node at the free end of the beam, opposite the supported end.

3. Set the force magnitude to 1000 N and choose its direction according to the example diagram. Make sure the load acts on the shorter edge of the cross-section.

4. Confirm the load. The force will appear in the tree on the left (image **5**).



Use the 3D view of the beam to check that the force is applied at the correct location and points in the correct direction.



f) Running the analysis

On the top toolbar, go to the ***Solution*** tab and select ***Solve***. Click ***OK*** in the new dialog. The analysis will start and several additional windows will open. Once a message confirms that the analysis is complete, you can close these windows.

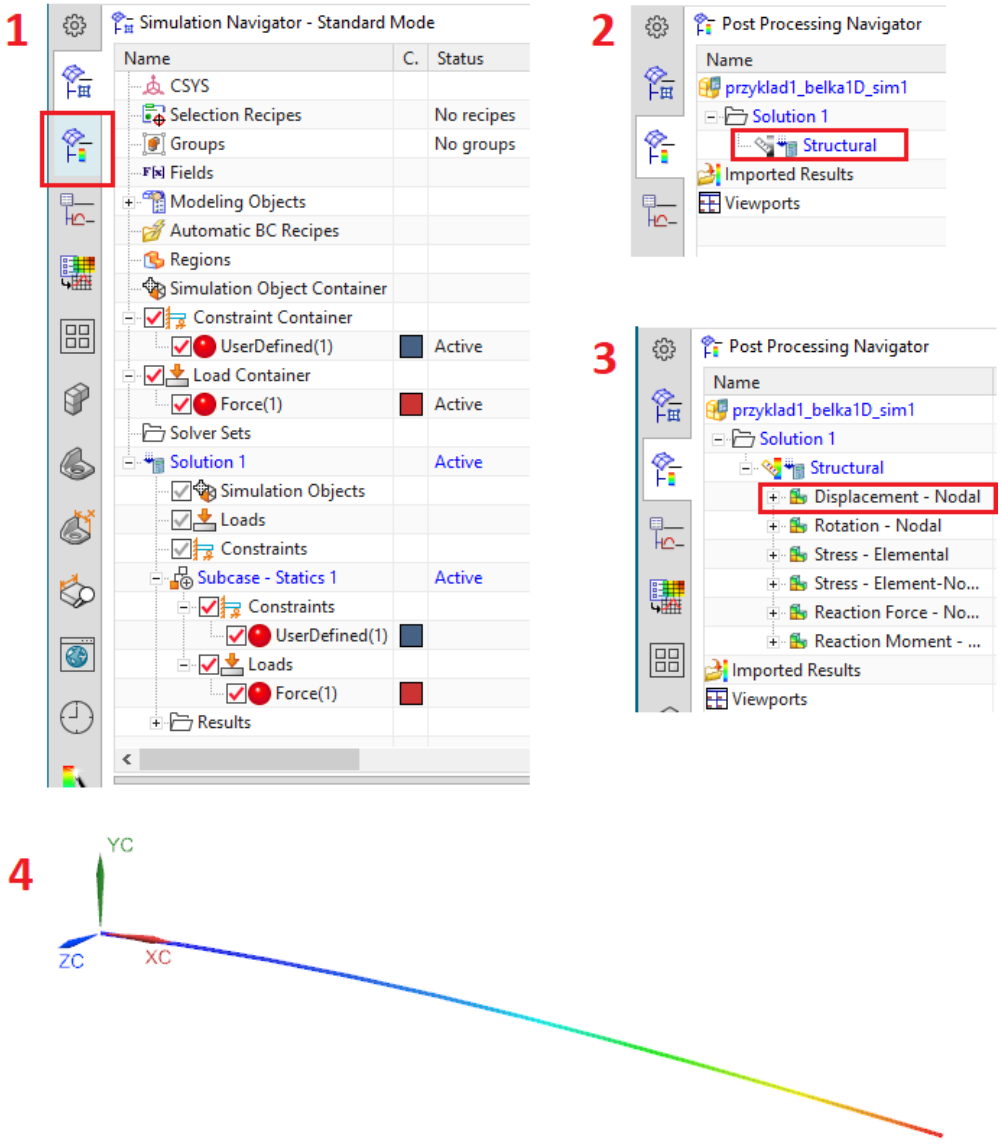
g) Viewing the results

1. Select the second tab next to the project tree (image 1). The results tree shown in image **2** will appear.

2. Double-click ***Structural*** with the left mouse button. New items will appear in the tree (image **3**).

3. Select ***Displacement – Nodal*** to display nodal displacements. Stress and reaction force results are also available.

4. Read the maximum beam displacement and compare it with the reference value of 2.951 mm. A matching result indicates that you have reproduced the example correctly.

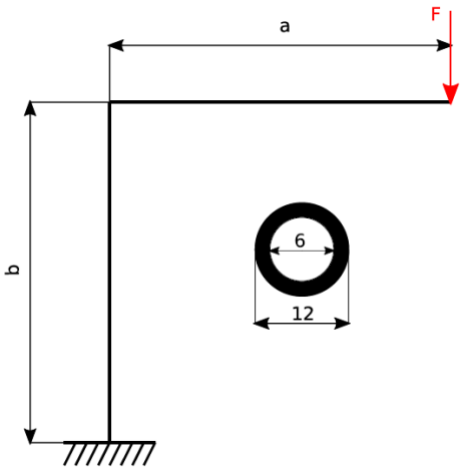


2. Exercise 1

Create a model of the structure shown in the diagram below. Use the specified cross-section for both members, **a** and **b**.

Use **a = b = 100 mm**, an element length of **20 mm** and a force of **F = 1000 N**. In the material selection dialog, create a custom material with the properties of steel: Young's modulus **E = 206 GPa** and Poisson's ratio **v = 0.3**.

Read the displacement at the free end of the beam, where the force is applied.



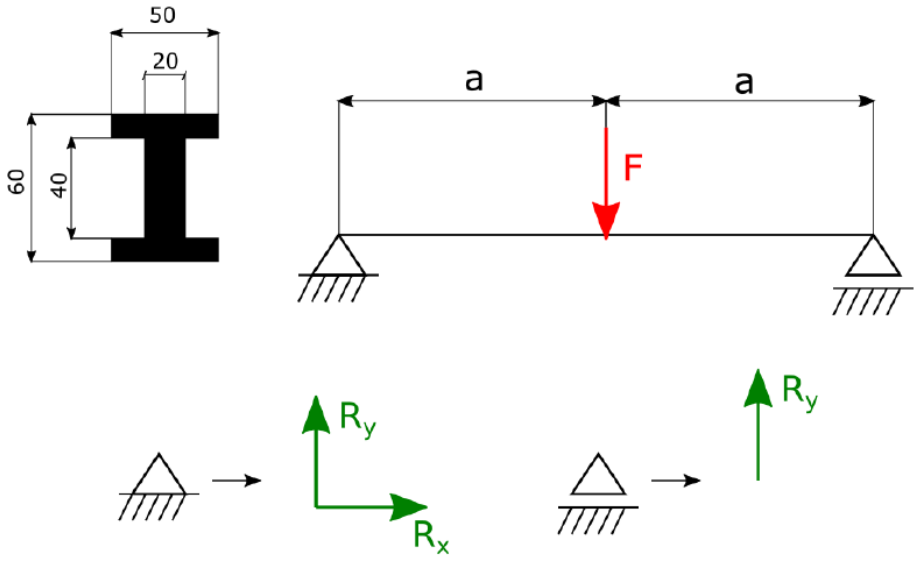
3. Exercise 2

Perform a static analysis similar to the introductory example. Create a new beam model according to the diagram below.

Use **a = 500 mm**, an element length of **50 mm** and the properties of aluminium: Young's modulus **E = 70 GPa** and Poisson's ratio **v = 0.33**. A force of **F = 1000 N.**

At the supports, define boundary conditions corresponding to the reactions shown in the diagram. Also constrain the appropriate degrees of freedom at all nodes so that the structure can move only in the plane of the diagram.

Read the displacement at the point where the force is applied.



4. Exercise 3

Determine the approximate maximum frame height a for which the maximum stress in the structure shown below does not exceed **50 MPa**. Use **b = 500 mm** and **F = 1000 N**. Select steel from the NX material library.

Use cross-section **I** for the vertical beams of length a and cross-section **II** for the horizontal beams of length b. Use the following cross-section dimensions: **x = 50 mm**, **y = 25 mm** and **c = 3 mm**.

