Virtual Prototyping

Finite Element Mesh Properties

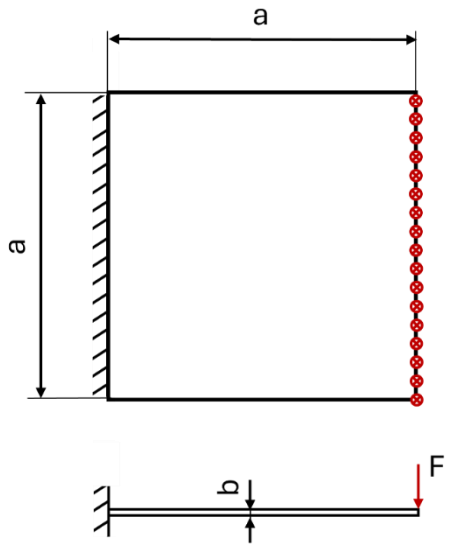
Learning objectives:

1. Understand the basics of simulation using two-dimensional (2D) and three-dimensional (3D) elements.
2. Learn to select finite elements based on their geometry and number of nodes.
3. Learn to connect different types of finite elements within a single structural model.

1. Analysing a thin plate using 2D elements

A square plate is fixed along one entire edge. A load perpendicular to the plate surface is applied to the opposite edge.

Example data: **a = 500 mm, b = 6 mm, F = 100 N**.



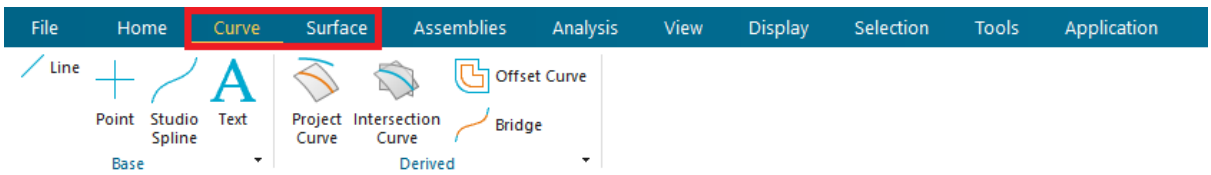
a) Creating the geometry

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

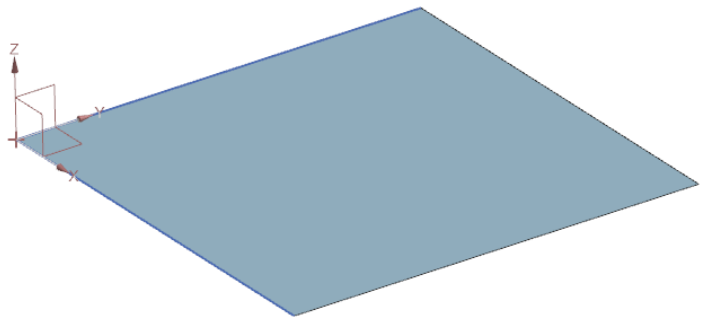
You can create the plate surface using commands on the ***Curve*** and ***Surface*** tabs (image below). One option is to use the ***Swept*** command on the ***Surface*** tab.

1. Use ***Curve*** → ***Line*** to draw two perpendicular line segments, each **500 mm** long, to form the edges of the surface.

2. Select ***Surface*** → ***Swept***, then select the line segments in the **Select Curve** fields.



The completed surface is shown in the image below.

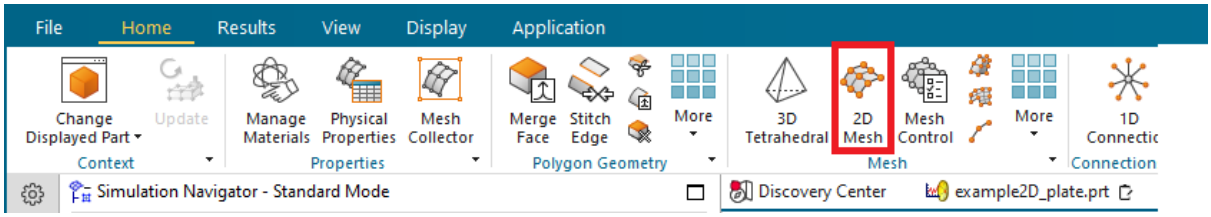


b) Discretising the model

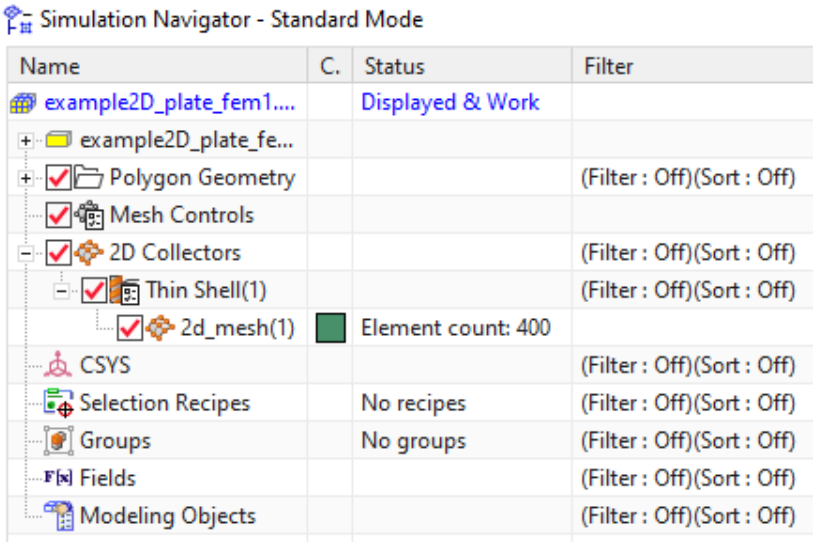
Once the geometry is ready, create a simulation as in the previous class. Select ***File → Pre/Post***, then ***New FEM and Simulation → New FEM and Simulation***.

Click ***OK*** in the simulation settings dialog. In this case, you do not need to select any additional geometry. In the next dialog, click ***Create Solution***.

Switch to the ***.fem*** file and select ***2D Mesh*** on the top toolbar. In the mesh settings dialog, select the surface, keep the default ***CQUAD4*** element type and set Element Size to **25 mm**.



An expandable **2D Collector** item will appear in the tree on the left (image below).



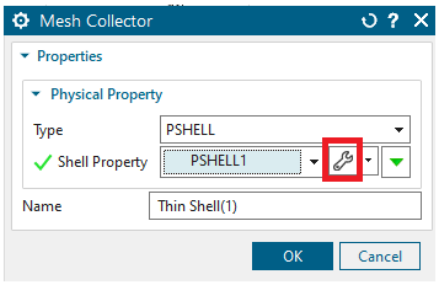
c) Assigning properties

1. In the tree on the left, double-click ***Thin Shell(1)***. The shell property selection dialog will open. The program has automatically created the ***PSHELL1*** property (image **1** below).

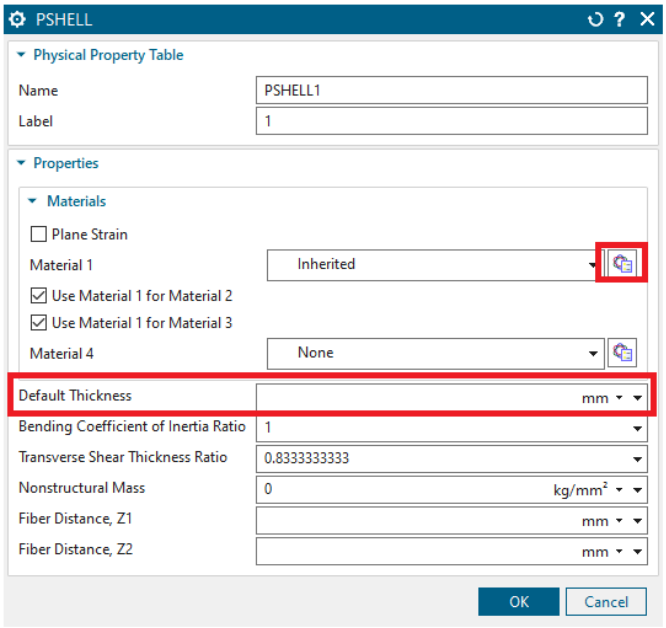
2. Click the wrench icon to open the property editing dialog (image **2**).

3. Select ***Steel*** from the material library and set the plate thickness to **6 mm**. Click ***OK*** to confirm the settings.

1



2

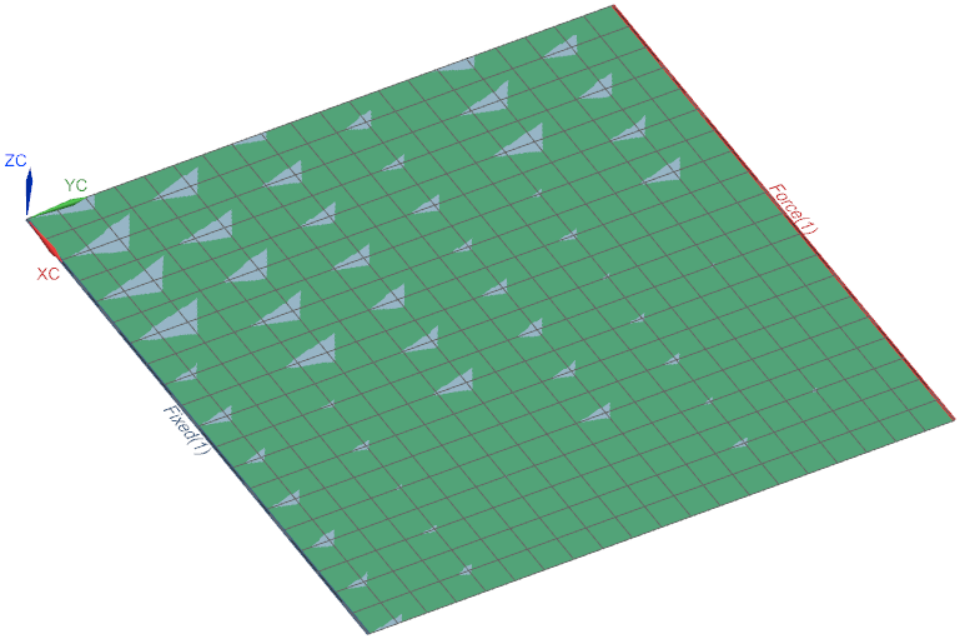


d) Adding supports and loads

Apply a fixed support to one edge of the plate using ***Constraint Type → Fixed Constraint***, constraining all degrees of freedom.

Then select ***Load Type → Force*** and apply the force to the opposite edge, perpendicular to the plate surface.

The model with the support and load is shown below.

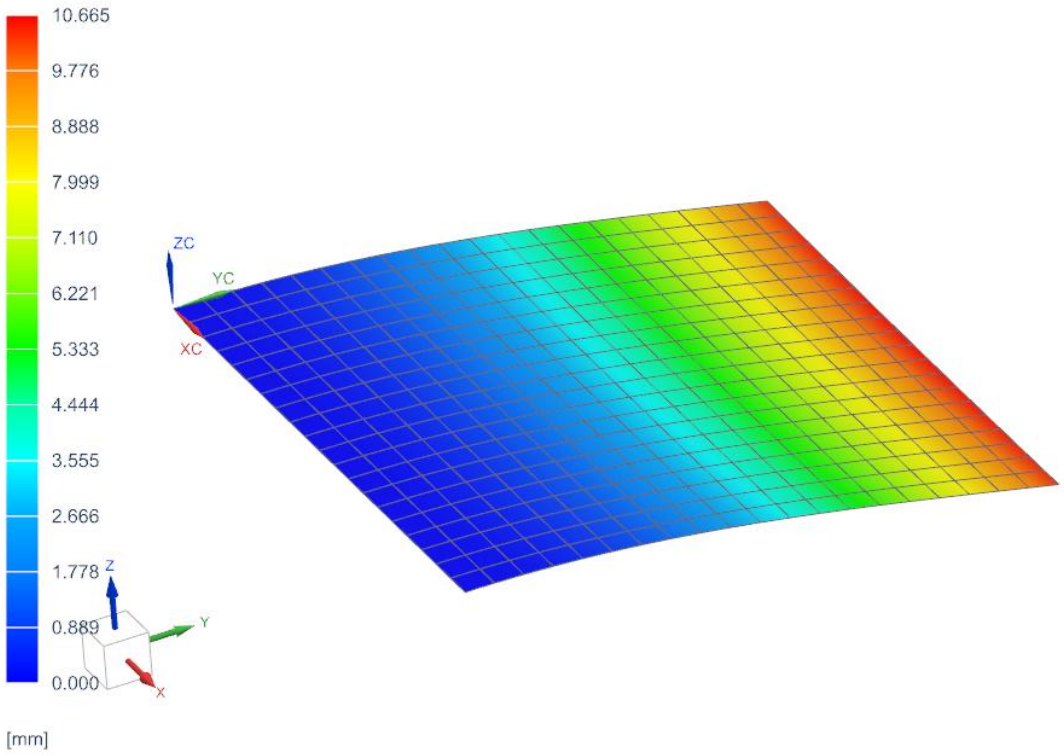


e) Running the analysis

On the top toolbar, go to the ***Solution*** group 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.

f) Viewing the results

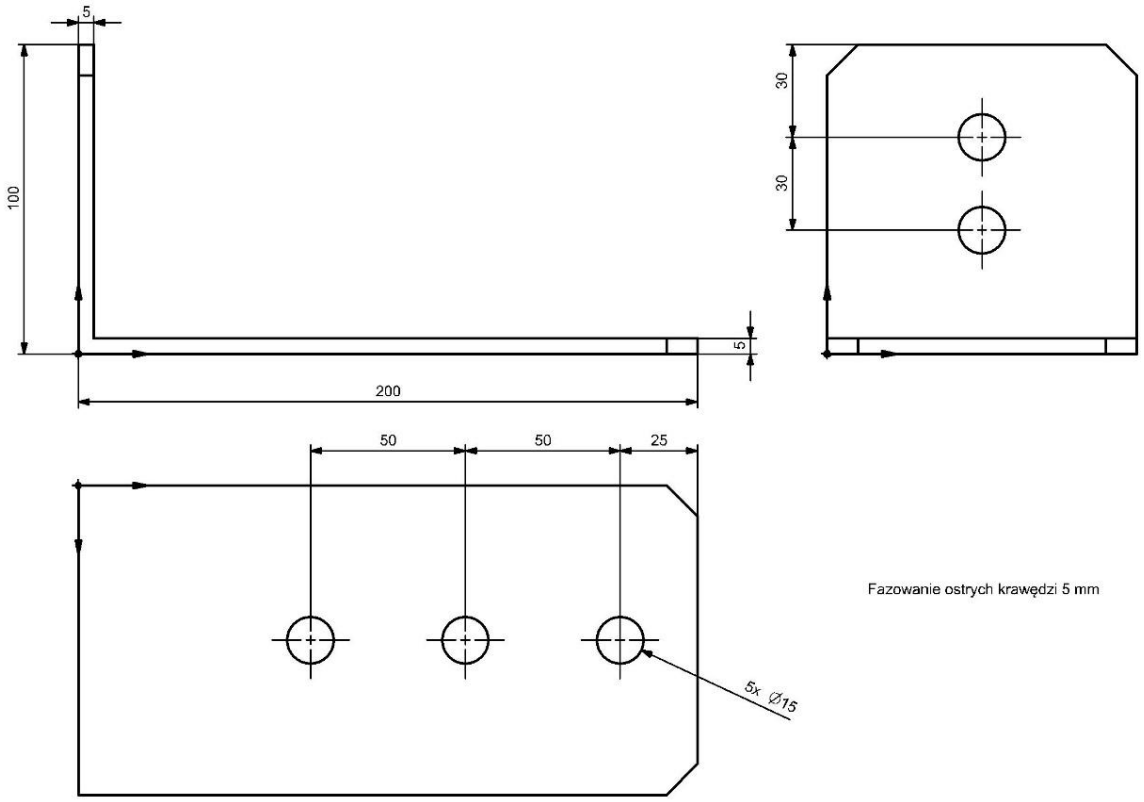
On the results tab, display the nodal displacements. An example results view is shown below.



2. Analysing a model using 3D elements

a) Creating the geometry

Create a new part file. Choose a save location and enter a file name. Create a 3D model according to the drawing below. You can use ***Extrude***, ***Shell*** and ***Hole*** on the ***Home*** tab in that order.

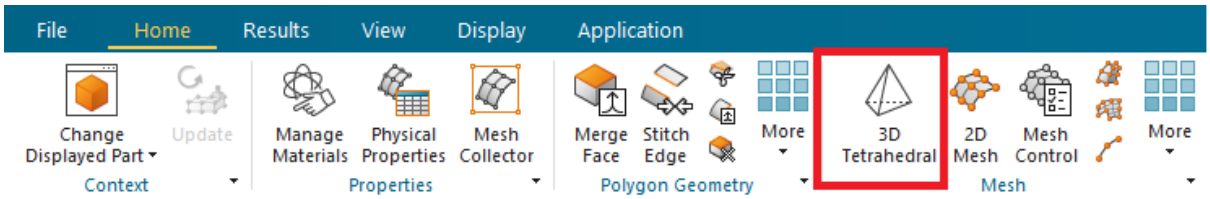


b) Discretising the model

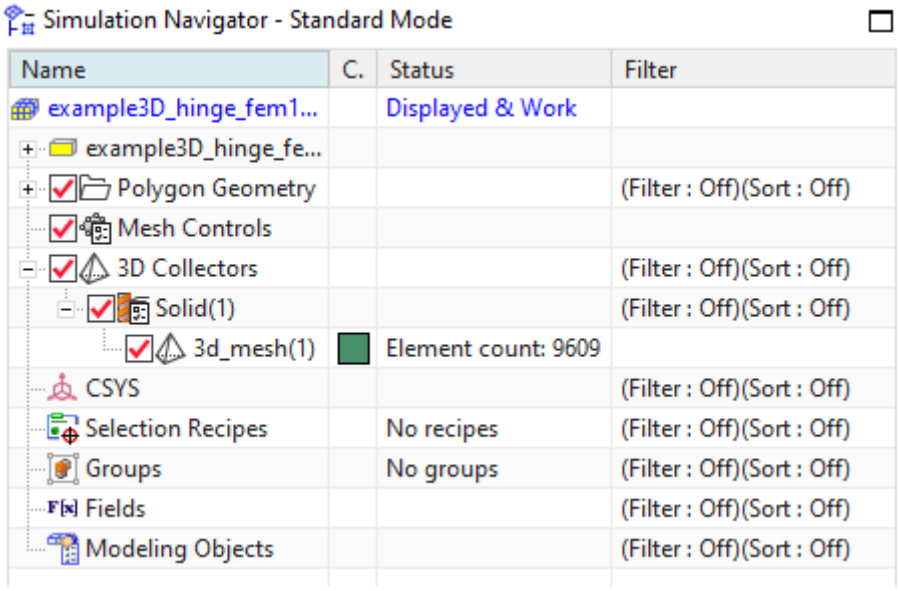
Create a simulation: select ***Application → Pre/Post***, then ***New FEM and Simulation → New FEM and Simulation***.

Click ***OK*** in the simulation settings dialog. In the next dialog, click ***Create Solution***.

Switch to the ***.fem*** file and select ***3D Tetrahedral*** on the top toolbar. In the mesh settings dialog, select the model, keep the default ***CTETRA10*** element type and set the element size to **5 mm**.



An expandable ***3D Collectors*** item will appear in the tree on the left (image below).

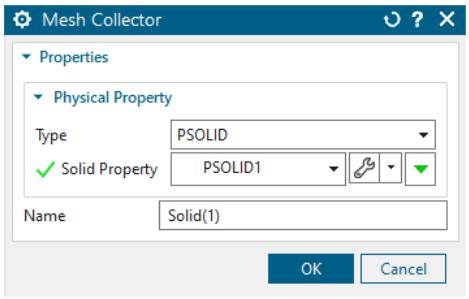


c) Assigning properties

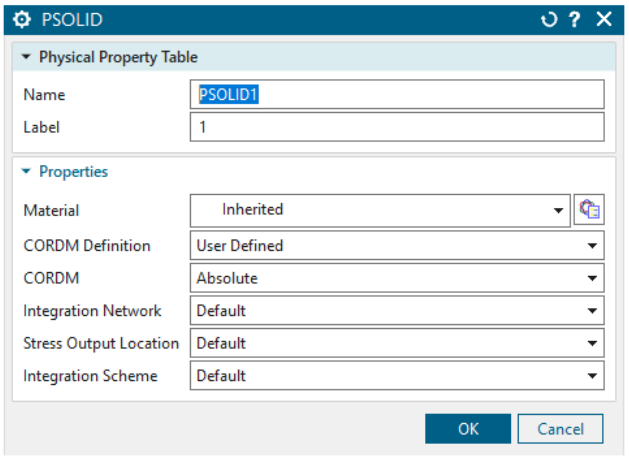
1. In the tree on the left, double-click ***Solid(1)***. The property selection dialog will open. The program has automatically created the ***PSOLID1*** property (image **1** below).

2. Click the wrench icon to open the property editing dialog (image **2**). Select ***Aluminum_6061*** from the material library.

1



2

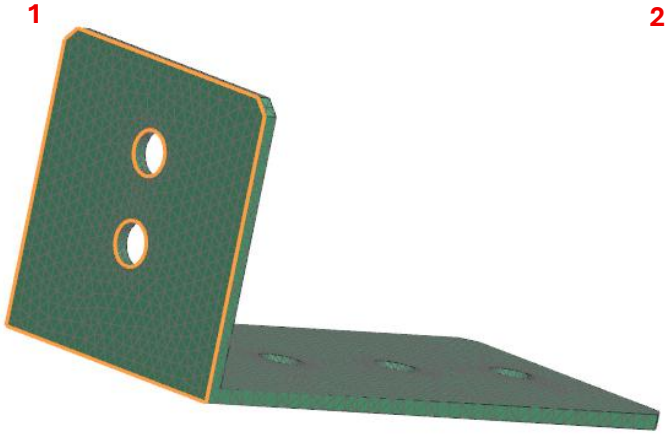


d) Adding supports and loads

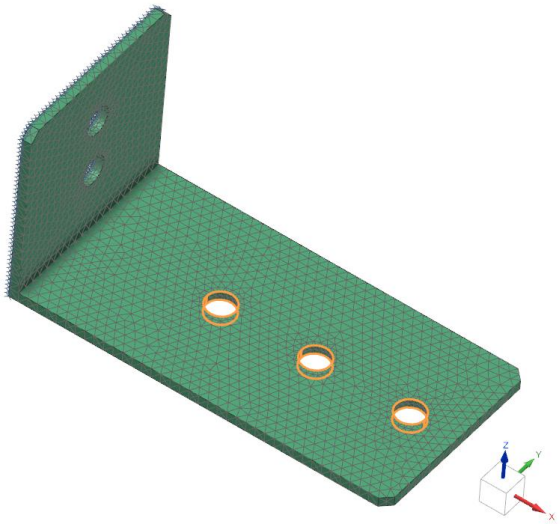
Use ***Constraint Type → Fixed Constraint*** to apply a fixed support to one face of the shorter leg of the angle bracket (image **1** below).

Then select ***Load Type → Force*** and apply a force of **2000 N** to the cylindrical surfaces of the holes. Set the force direction as shown in image **2**, along the ***y*** axis.

1



2

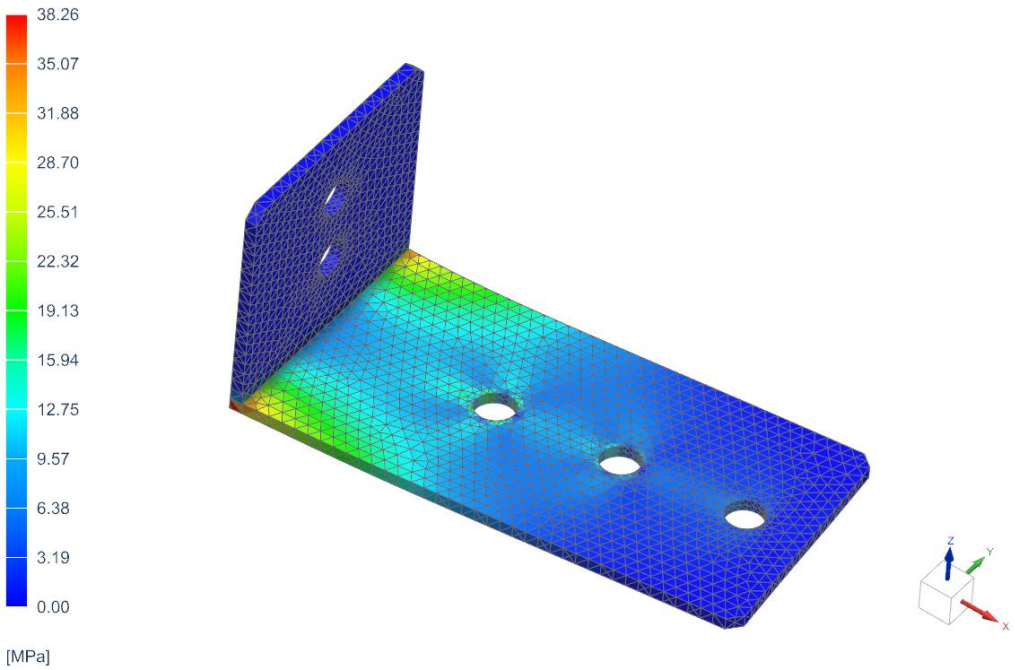


e) Running the analysis

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.

f) Viewing the results

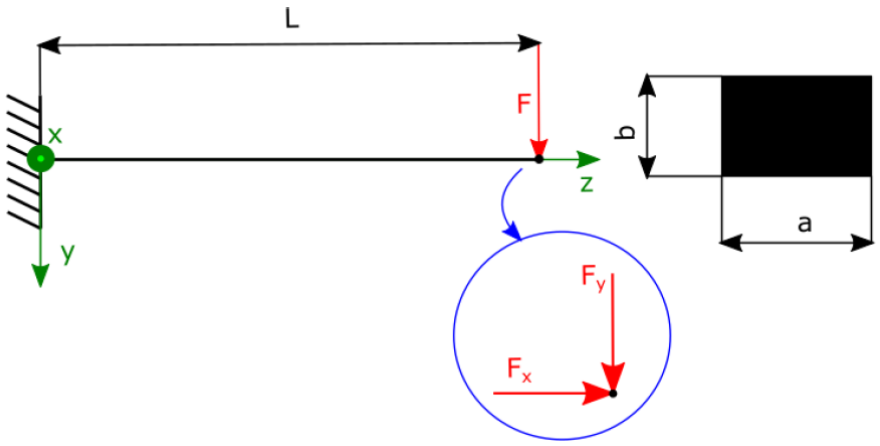
On the results tab, display the ***von Mises*** equivalent stress. An example results view is shown below.



3. Exercise 1

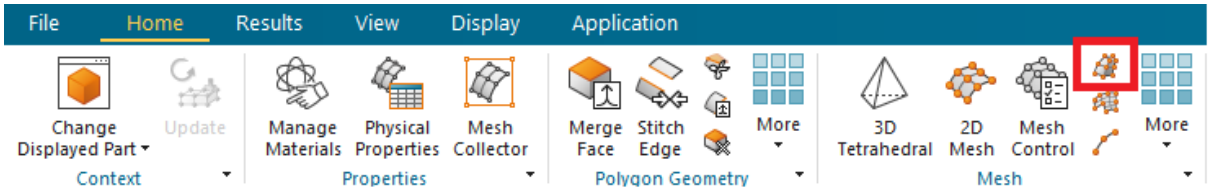
Create three beam models, using 1D, 2D and 3D elements, according to the diagram below. For the 2D model, distribute the load over the entire edge at the free end of the beam; for the 3D model, distribute it over the entire end face. At the other end, apply a fixed support to the entire edge (2D) or end face (3D). Force F has two perpendicular components in the XY plane: **Fx** and **Fy**.
For each model, run analyses using different finite element types and mesh sizes, as specified in the table below the diagram. After each analysis, read the displacement (***Displacement***) at the free end of the beam, where force **F** is applied, and enter the result in the table. For the 2D and 3D models, read the displacement at one selected node at the free end.

Use **a = 60 mm, b = 20 mm, L = 1000 mm, Fx = Fy = 100 N**. Select **Steel** from the NX material library.

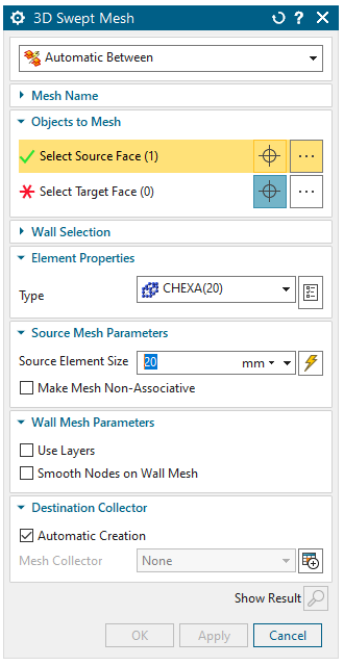


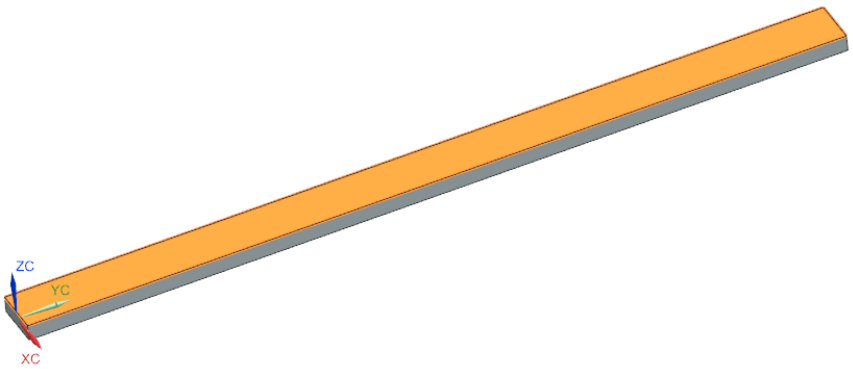
Grid size [mm]	Mesh topology					
	1D	2D		3D		
	CBEAM	CTRIA3	CQUAD4	CTETRA4	CTETRA10	CHEX8
20						
5						
3						

For the 3D model in this exercise, also use a **Hex8** mesh. This is created differently from tetrahedral **Tet4** and **Tet10** meshes. Select ***3D Swept Mesh***. Note: an existing tetrahedral mesh (TET) cannot be converted directly into a hexahedral mesh (CHEX8). Generate the hexahedral mesh from the solid geometry as a new mesh.



Select two parallel faces of the beam: the first in ***Select Source Face*** and the second in ***Select Target Face***. Enter the element size in ***Source Element Size***.

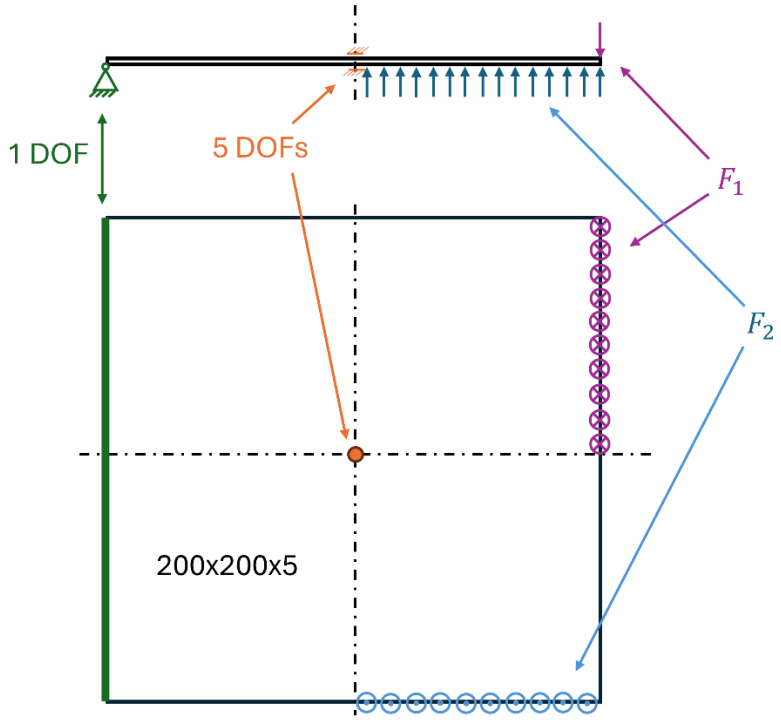




4. Exercise 2

Analyse the plate shown below. You can create the geometry as one large surface or four smaller, adjacent surfaces. Your choice affects subsequent work in the **.fem** and **.sim** files. The four-surface option is recommended; you can try the single-surface option as an additional exercise. If you choose four surfaces, you **must** join them using ***Sew*** on the ***Surface*** tab in the part file.

Use forces $\mathbf{F_1 = F_2 = 500\ N}$ and select **steel** from the NX material library.

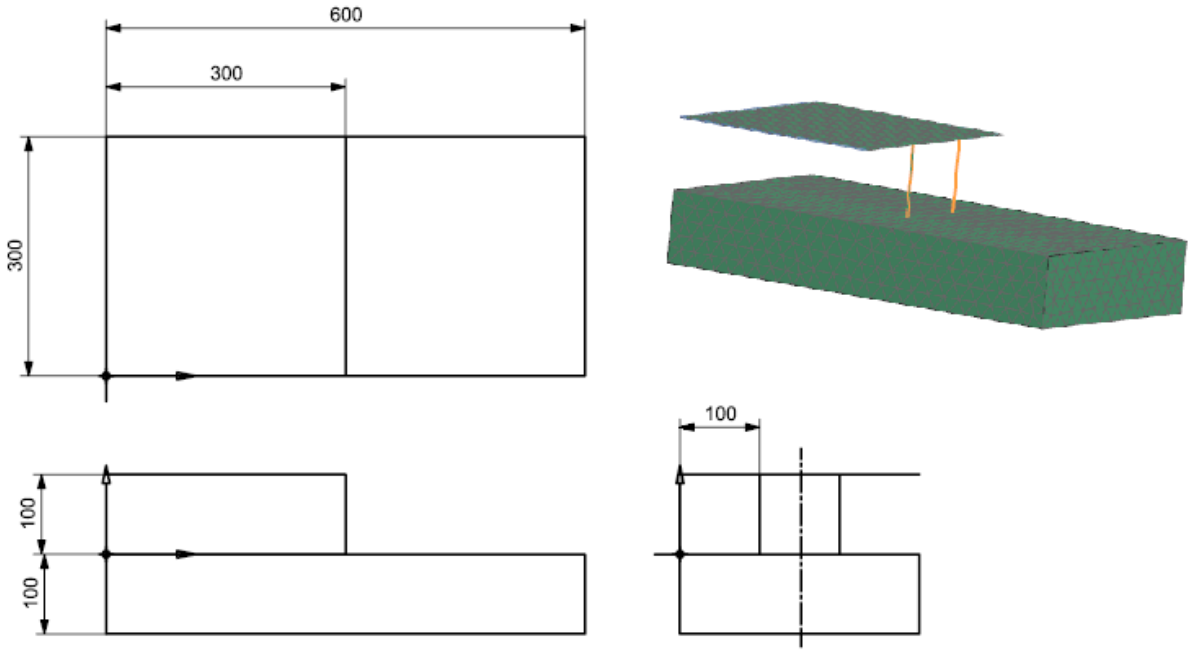


5. Exercise 3

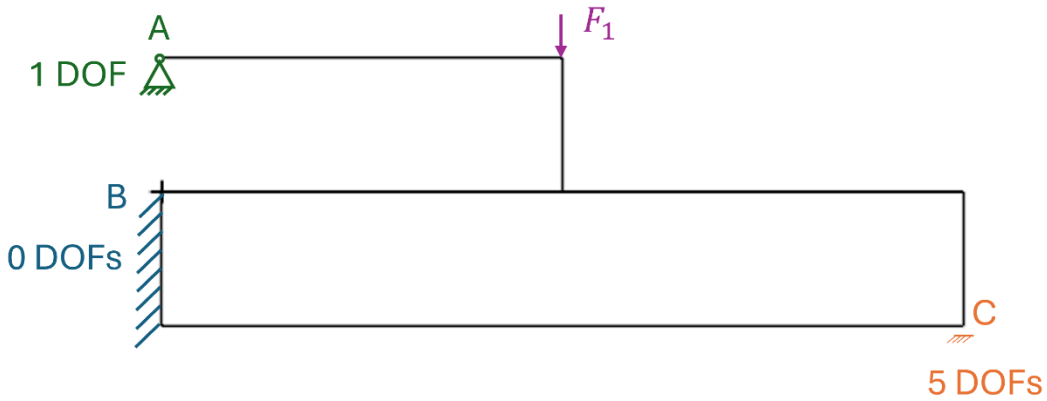
Analyse the model shown below. You can create all the geometry in a single part file. Model the top plate as a surface.

The model contains three types of finite elements. Connect them using ***1D Connection*** by selecting the 1D mesh and the edge/surface to which it is to be connected (see image below). **Creating a separate 1D mesh for each beam is recommended.**

Select **steel** for the plate and beams, and **ABS** for the three-dimensional support. The boundary conditions are given below.



Define the boundary conditions according to the diagram below. **A**pply supports A and **C** along the entire indicated edges, and support **B** over the entire indicated face. Use a force of $\mathbf{F_1 = 5\ kN}$.



1D Connection

