Statics – experimental validation, plastic deformation

Introduction

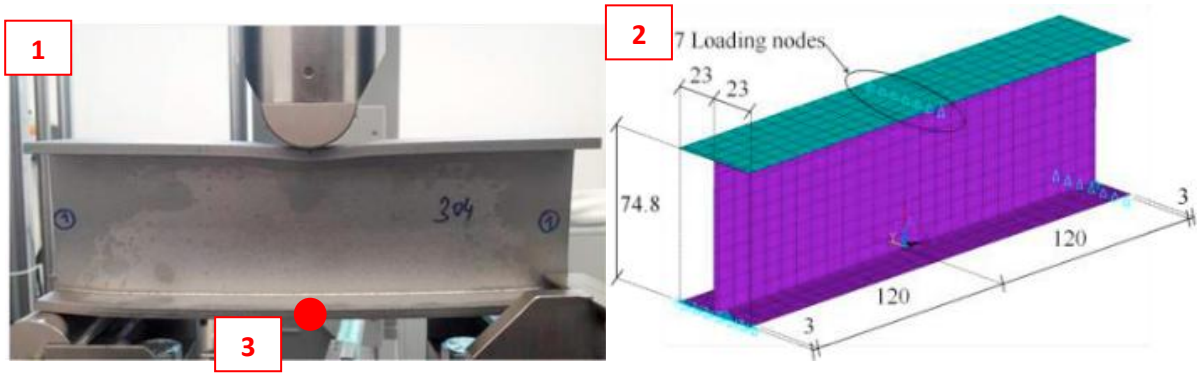
This class covers the problem of experimental validation of the FEA simulation model. For this purpose, a model representing a three-point bending test will be created. The model will be a recreation of a real-life experiment described in the scientific article. The model will use an elasto-plastic material definition and a nonlinear simulation will be performed.

Numerical model

The basis for the creation of the model is the following article:

D. Jindra, Z. Kala, J. Kala, S. Seitl, *Experimental and Numerical simulation of a Three Point Bending Test of a Stainless Steel Beam*, Transp. Res. Procedia. 55 (2021) 1114–1121. https://doi.org/10.1016/j.trpro.2021.07.183

The beam model representing a three-point bending test is shown below **(1),** together with the FEA model presented in the article **(2).** The beam was placed on the supports with a span of 240 mm. During the measurement the pressure load and the displacement was measured at the mid-span point in the bottom part of the beam. In the article several simplifications were made: the beam geometry is represented using 2D elements, the supports are assumed to be perfectly stiff, the loading of the beam was applied using displacement up to around 8 mm. The supports were shifted by around 3 mm from the beam edge.



Task I

Create a simplified model based on the article and compare the results obtained with those given in the article. Apply the load with a force distributed over a 6x24 mm area in the middle of the long axis of the beam. Assume load with a force of 20 kN. The supports should be applied at nodes 3 mm away from the lower edges of the beam. We will assume the material properties of 304/1.4301 steel. In the elastic range, the following properties are given:

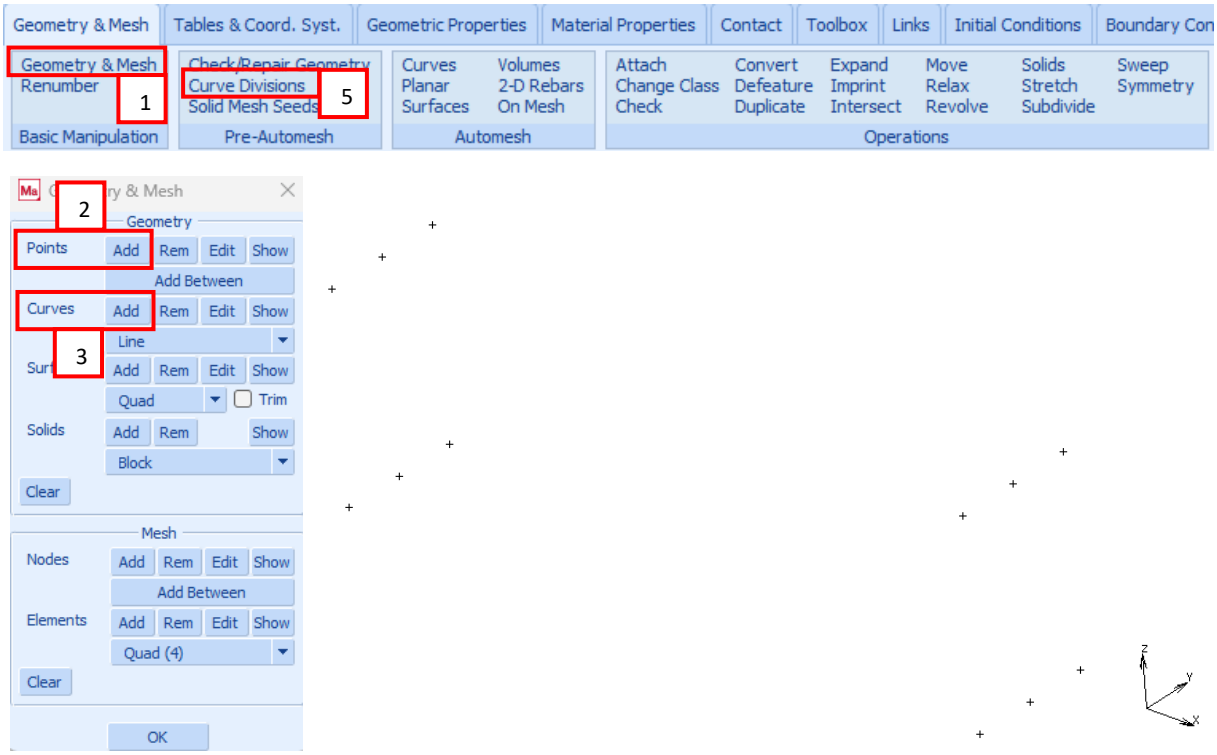
- Young's modulus: $E = 195$ GPa
- Poisson's ratio: $\nu = 0.27$
- density: $\rho = 7800$ kg/m^3

Model geometry

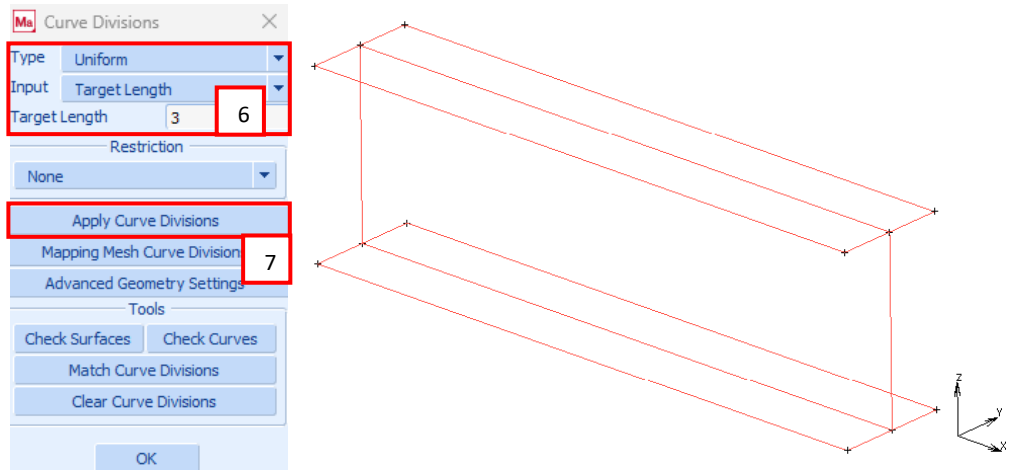
The geometry of the beam should be represented by three surfaces with the given thickness: two surfaces of 246 x 46 mm (in the XY plane) and one of 246 x 74.8 mm (in the XZ plane). Select the

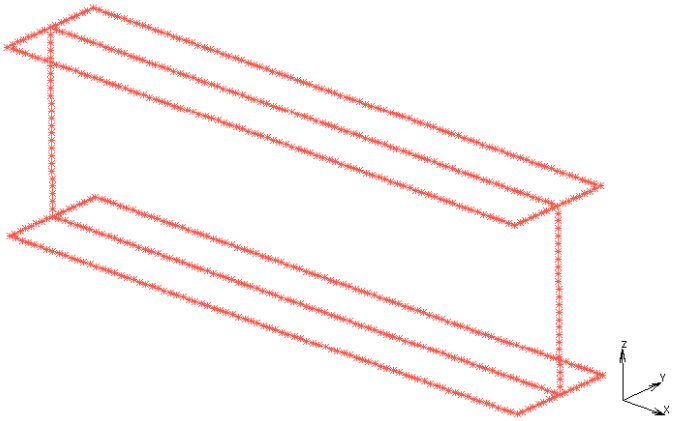
Geometry & Mesh - Basic Manipulation - Geometry & Mesh tab **(1)** and in the new window select **Points - Add (2).** Enter the coordinates of 12 points:

Lp	P1	P2	P3	P4	P5	P6
XYZ (mm)	(0,0,0)	(246,0,0)	(246,46,0)	(0,46,0)	(0,0,74.8)	(246,0,74.8)
Lp	P7	P8	P9	P10	P11	P12
XYZ (mm)	(246,46,74.8)	(0,46,74.8)	(0,23,0)	(246,23,0)	(246,23,74.8)	(0,23, 74.8)



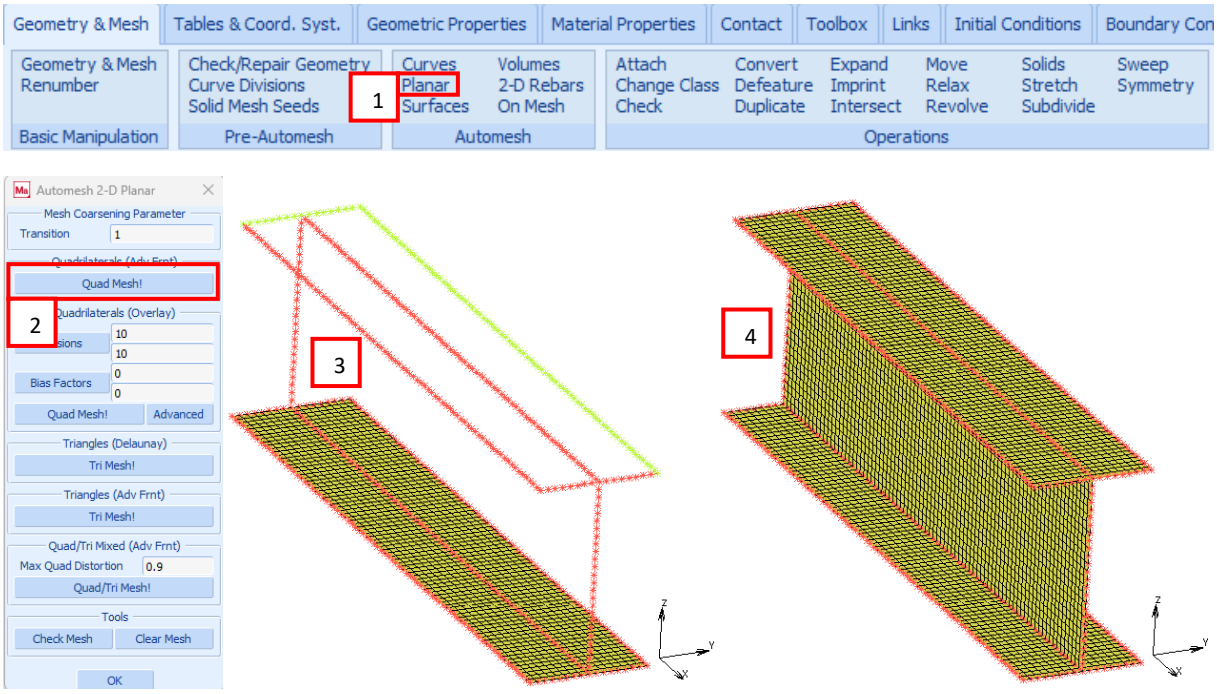
We choose **Curves - Line - Add (3)** and connect the points, creating 12 edges, as shown in the figure **(4).** Next, we choose **Geometry & Mesh - Pre-Automesh - Curve Divisions (5).** We select **Uniform (6), Target Length** and enter a value of 3. We click **Apply Curve Divisions (7),** select all curves and click RMB.



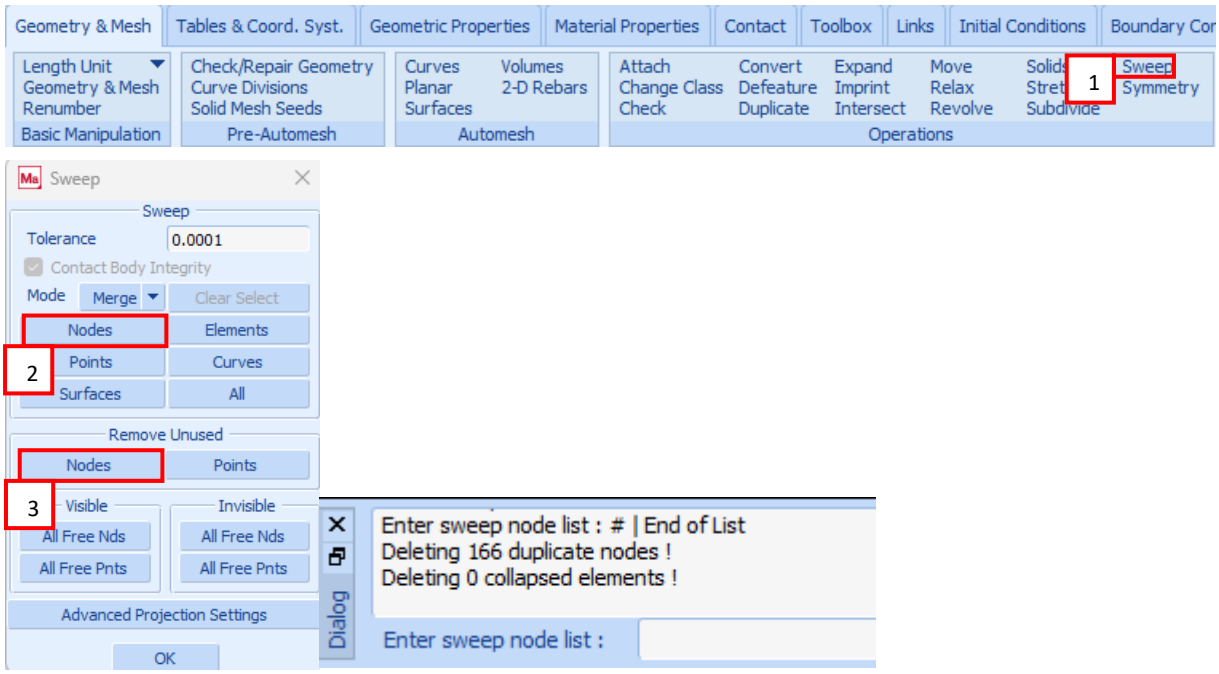


Meshing

We choose **Geometry & Mesh - Automesh - Planar (1).** We select **Quad Mesh!** and select the 4 edges corresponding to one of the three planes and click RMB **(3).** Repeat the operation for each plane **(4).**

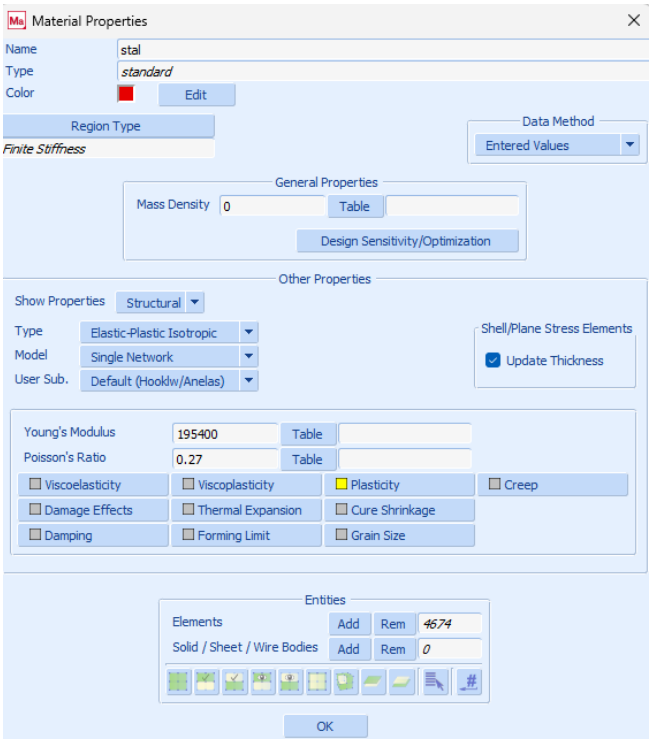


We select the **Operations - Sweep** function **(1).** We click **Sweep - Merge - Nodes (2),** select all nodes and click RMB. Then we click **Remove Unused - Nodes (3).** The program should remove 166 nodes.



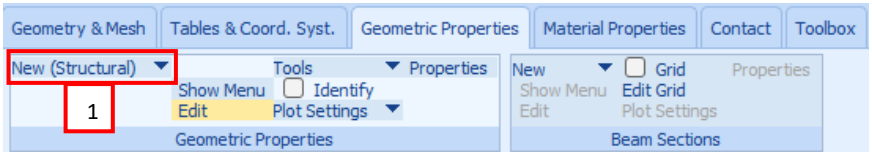
Material properties

The beam material should be defined in the **Material Properties - New - Standard** tab. In the **Input Properties** window, enter **Young's Modulus** and **Poisson's ratio**. Keep in mind the accepted convention of units. Next, assign material properties by selecting **Entities - Elements - Add** and select all elements.

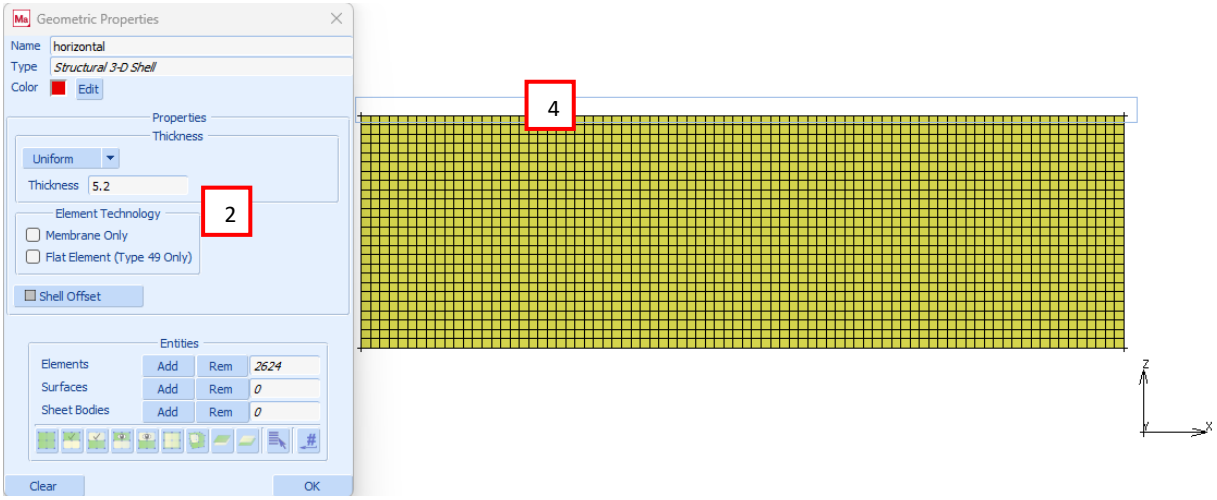


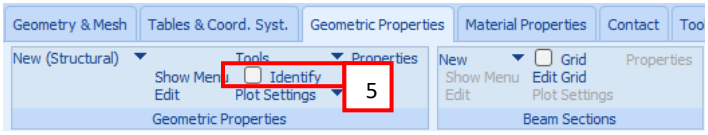
Geometric properties

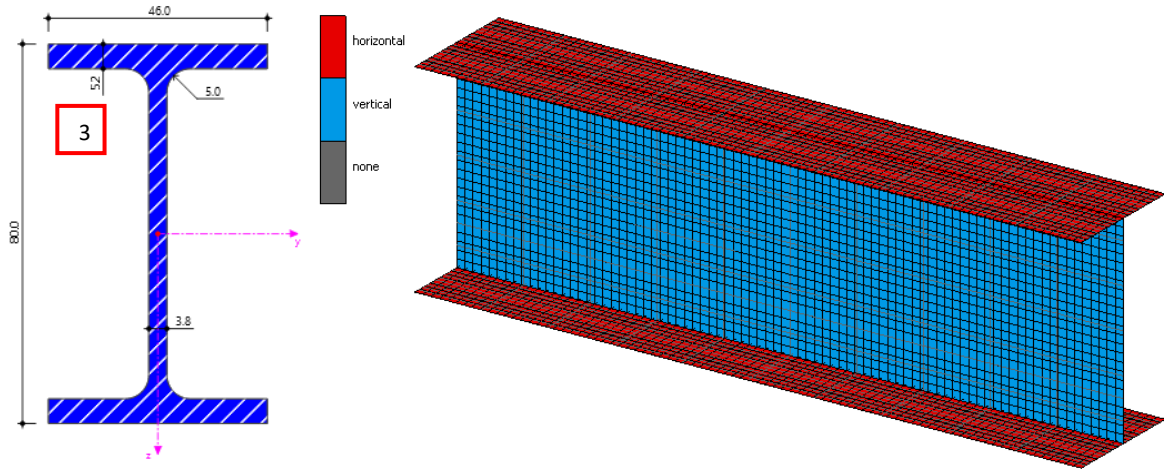
We assign material properties by selecting the **Geometric Properties - New - Shell tab (1).** We create two sets of properties, separately for vertical and horizontal geometry. We give them names and enter thicknesses of 5.2 mm **(2)** for the horizontal slabs and 3.8 mm for the vertical slab, as shown in the diagram below **(3)** (curves are omitted). To add properties, select the grid elements corresponding to the corresponding slabs.



You can make it easier to select elements by positioning the camera perpendicular to the XZ plane and first select all elements for one set (e.g., vertical), and for the other set select only selected elements (e.g., by dragging with the mouse as shown below) **(4)**. Finally, the added properties can be visualized by clicking **Geometric Properties - Geometric Properties - Identify (5).**

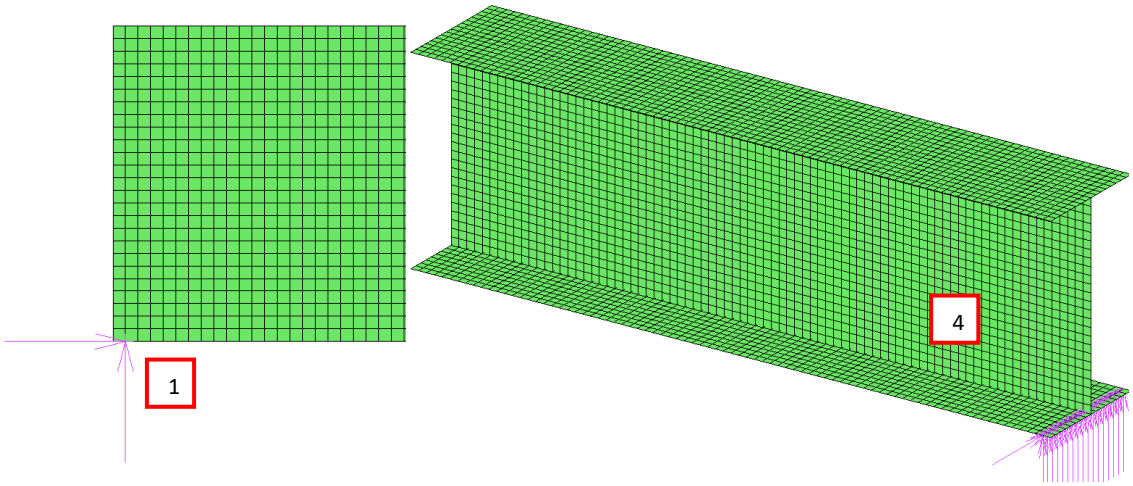


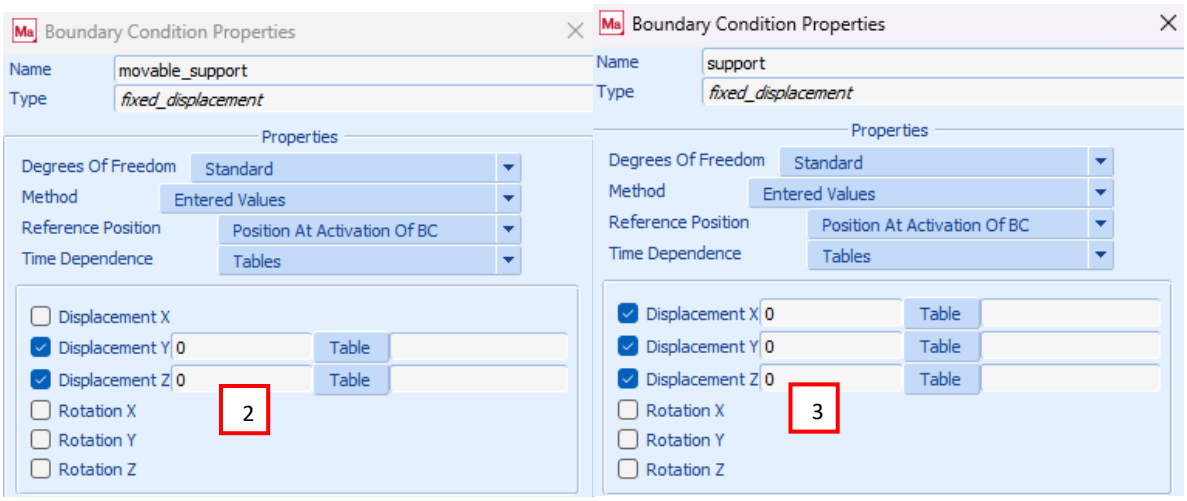




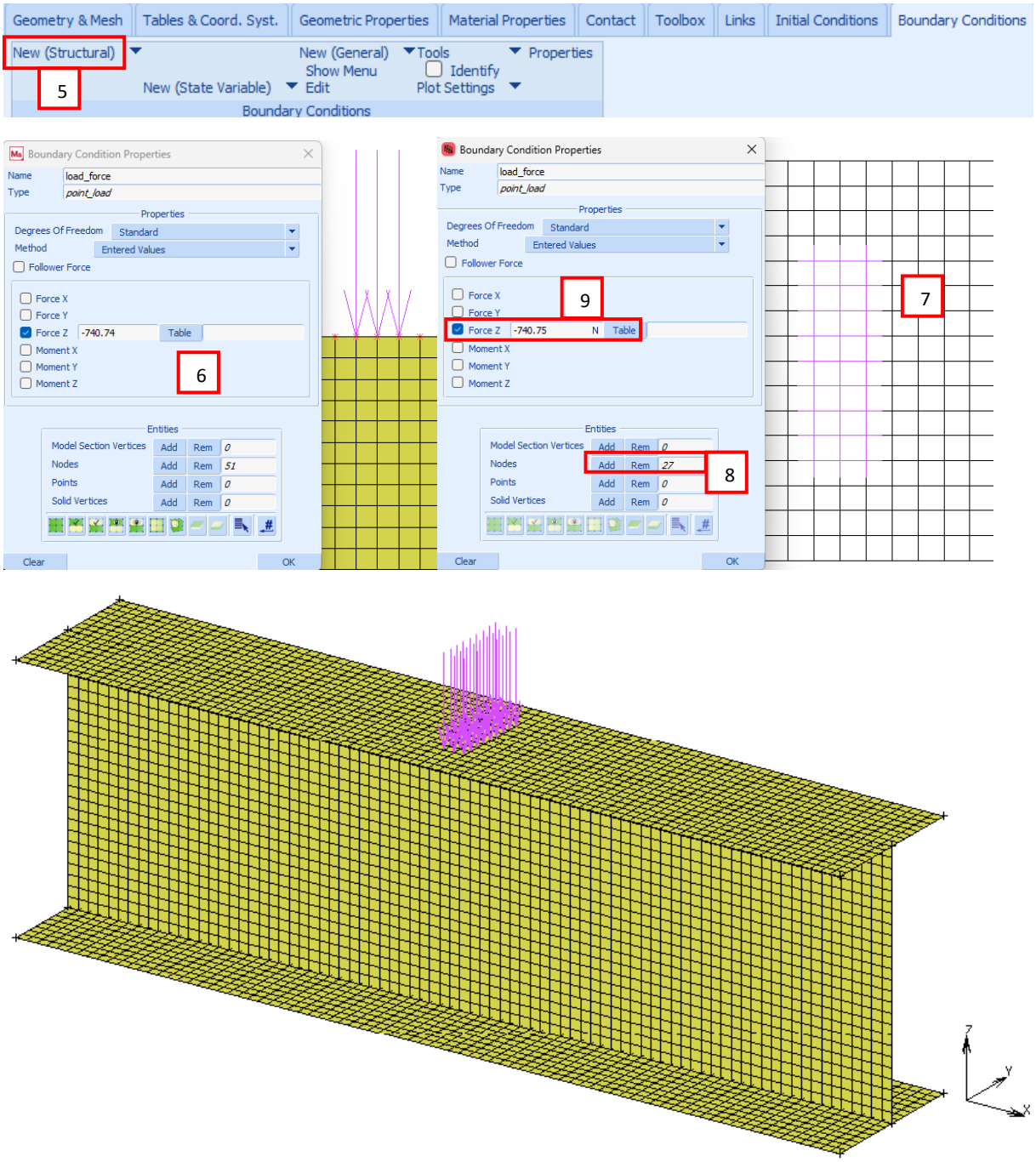
Boundary conditions

According to the conditions of the three-point bending test, two types of **Fixed Displacement** constraints must be defined. We give different names to constraints. We apply the first of the constraints to nodes offset by **3 mm (1)** from one of the lower edges of the beam and remove linear displacements in the YZ axes **(2)**. We apply the second constraint to the nodes **3 mm** from the other edge and remove linear displacements in the XYZ axes **(3)**. As with applying geometric properties, you can position the camera perpendicular to the XZ plane and drag the mouse around one of the visible nodes and click RMB. Correctly given constraints should look like the one shown below **(4)**.



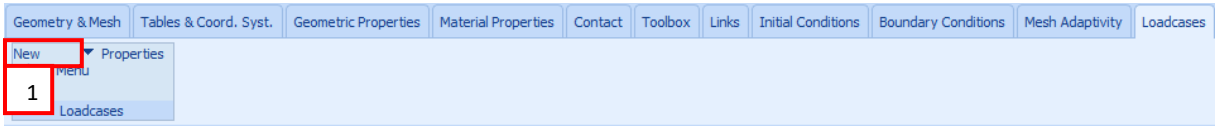


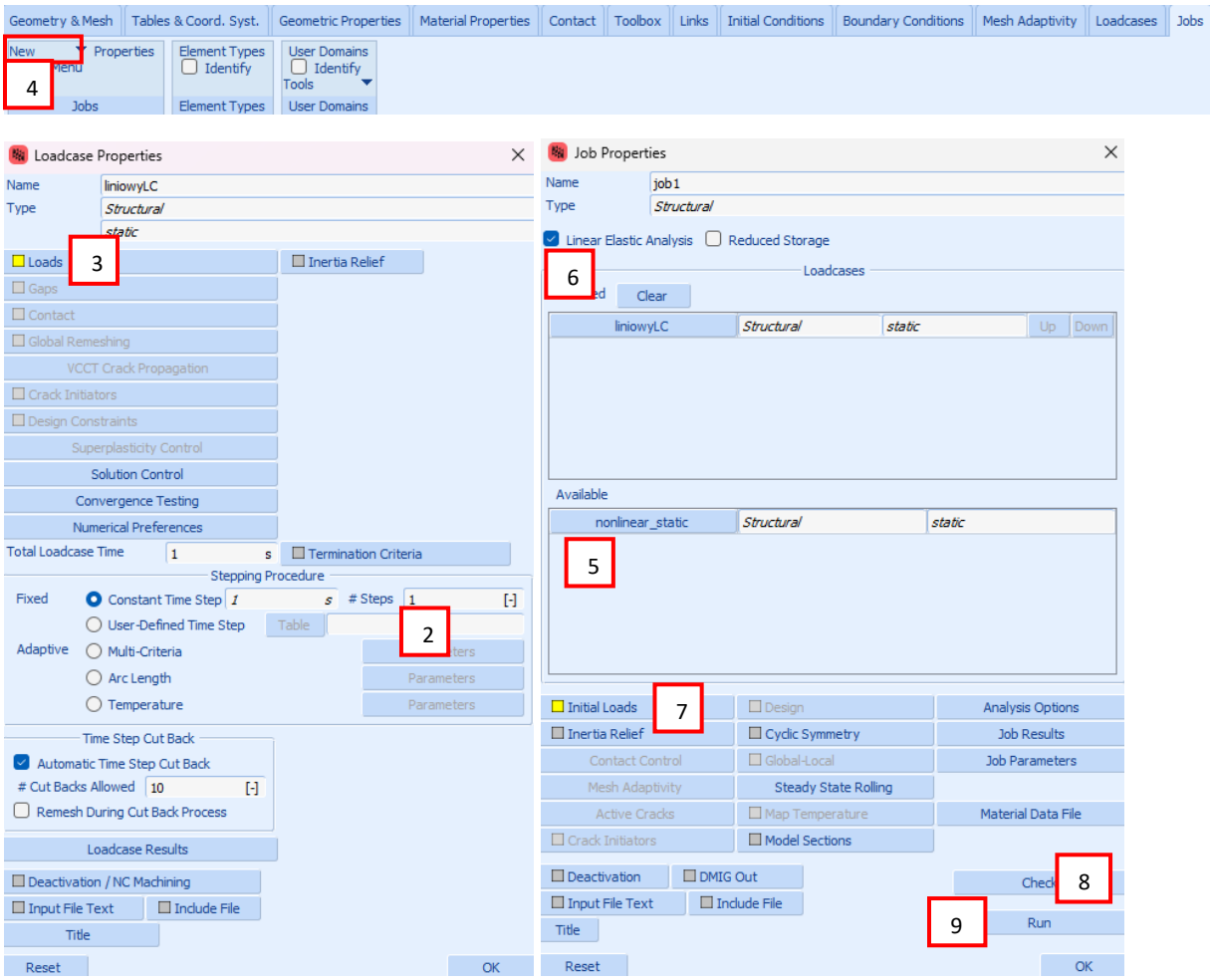
Next, we define the load on the top of the beam, in the middle of the longer axis. The force should be distributed over a 6x24 mm area. We choose **Boundary Conditions - New - Point Load (5).** To make the selection easier, we can create a line of geometry passing through the center of the slab. Next, we position the camera perpendicular to the XZ plane **(6),** and then select three adjacent nodes in the center of the top surface. Next, we select perpendicular projection for the XY plane **(7),** select **Nodes - Rem (8)** and remove unnecessary nodes. We enter the value of the force corresponding to 20kN divided by the total number of selected nodes **(9).**



Loadcase and Job

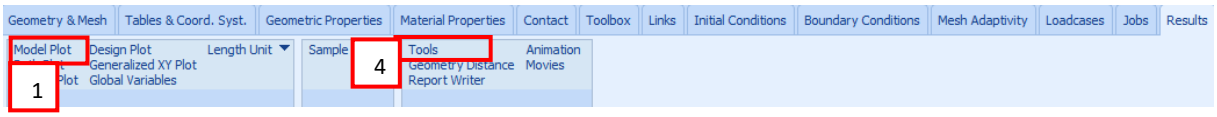
We create a new **Loadcase** from the **Loadcases - New - Static** tab **(1).** We enter 1 step in the **Steps** field **(2)** and add all the boundary conditions defined earlier in **Loads (3)** and click OK. Next, we create a new **Job** by selecting **Jobs - New - Structural (4).** In the settings, we select the **Loadcase** we created earlier from the **Available** field **(5)**, check **Linear Elastic Analysis (6),** and make sure all the load is added in **Initial Loads (7).** Then we click **Check (8)** to verify the model, click **Run (9)** and **Submit** to run the simulation. When the calculation is complete, we click **Open Binary Post File.**

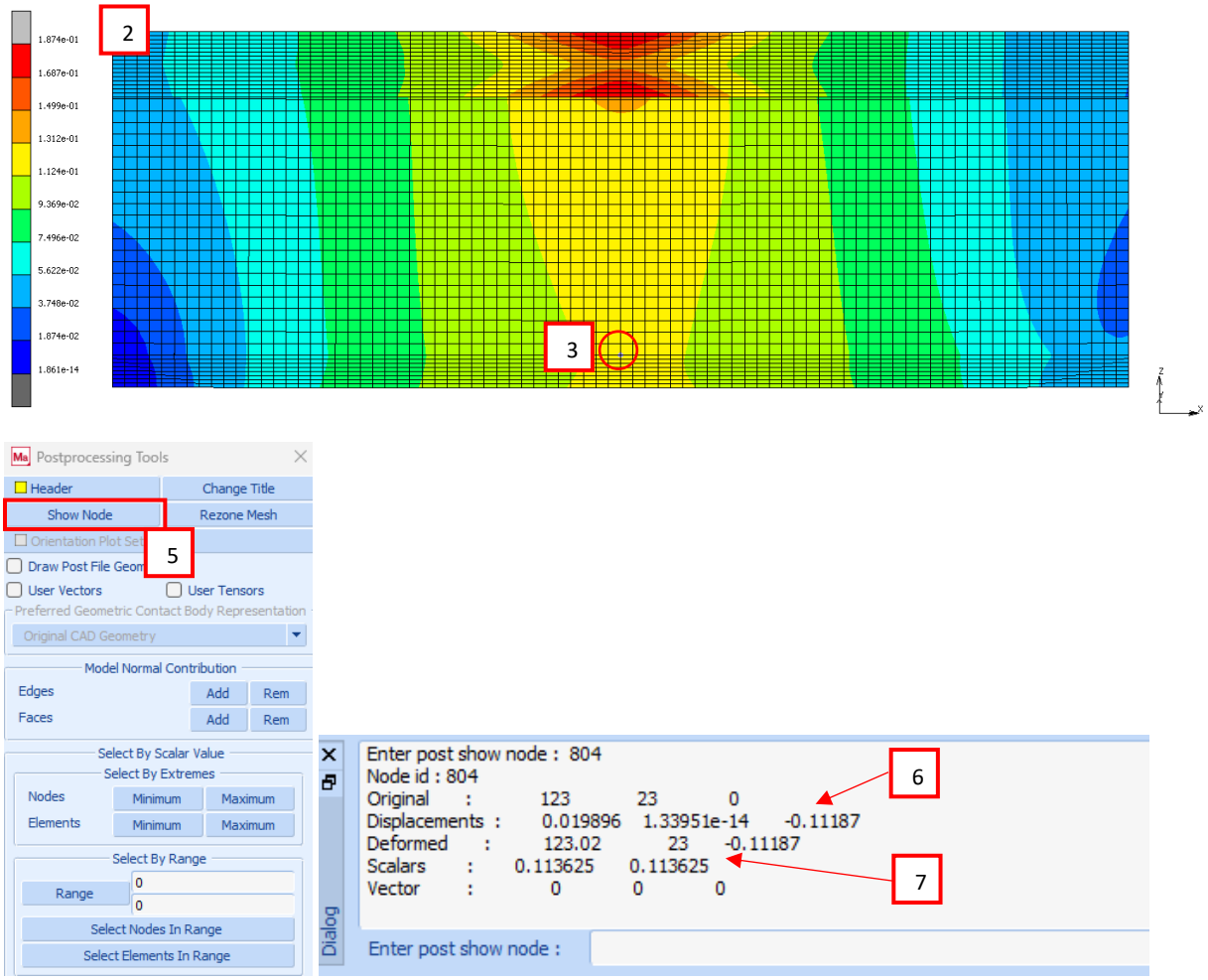




Simulation and results

After loading the results file in the **Model Plot Results** window **(1)** and in the **Scalar** field, select **Displacement.** We should get results similar to the following **(2)**. Then we read the displacement from the node in the middle of the bottom plane of the beam, on the axis of the applied force **(3)**. Select **Tools (4), Show Node (5),** select the appropriate node **(3)**. In the command window, the values of displacements (Displacements) in the XYZ axes **(6)** should appear, and the value of **Scalars (7)** should correspond to the value selected in **Model Plot Results** - in this case, the absolute value of displacement vector. The read displacement should be compared with the displacement obtained in the article (according to the graph below) **(8)**.





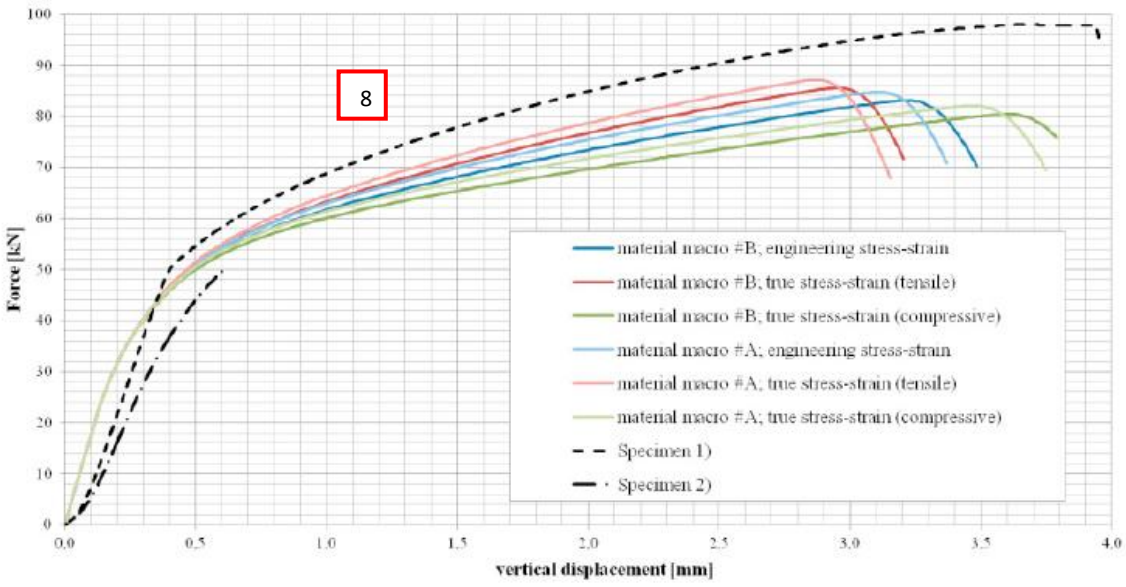


Fig. 5. Force-displacement dependence.

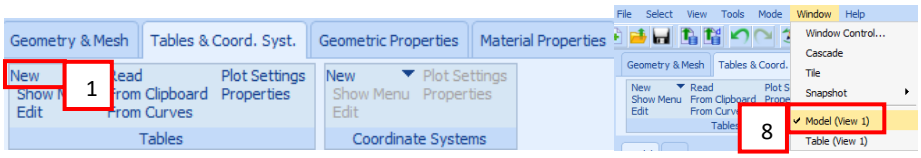
Task II

Repeat the simulation by changing the load to 70 kN. Again, read the displacement from the model and compare with the results from the article.

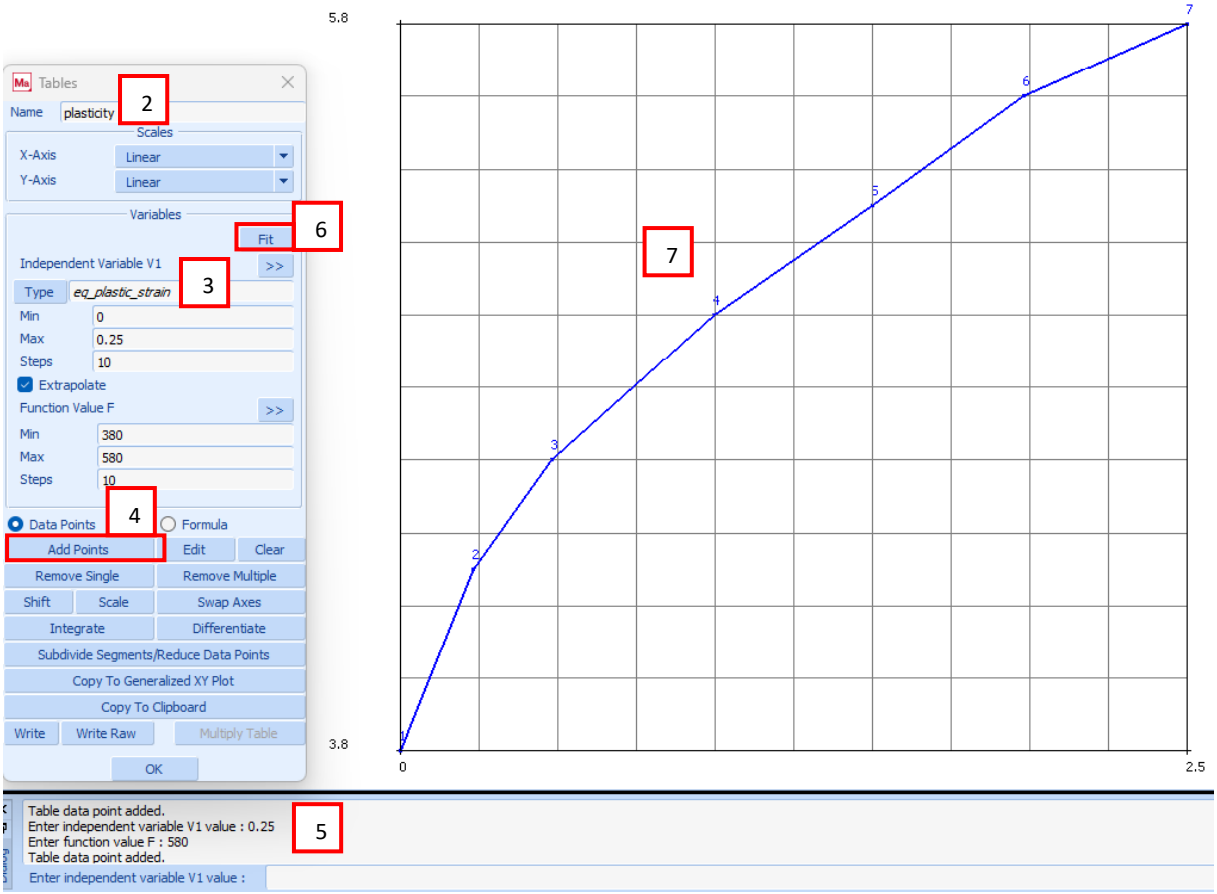
Task III

Perform a nonlinear simulation, assuming plastic properties of the material and loading by incremental displacement.

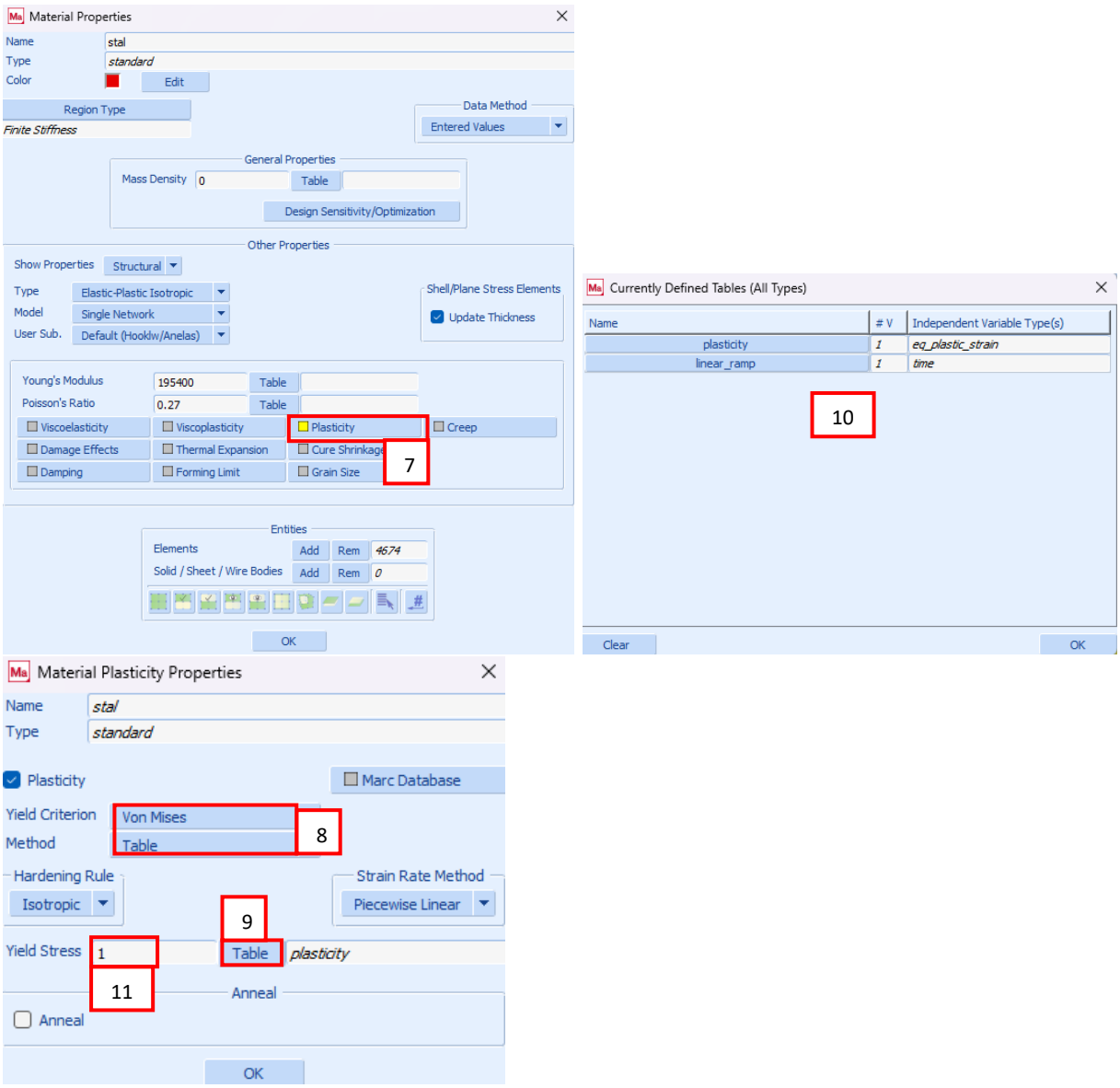
In the first step, we define a table with strain and stress values. We select **Tables & Coord. Syst. - New - 1 Independent variable (1).** We give a new name **(2)**, and in the table settings as **Independent Variable V1 (3)** we choose **Equivalent Plastic Strain.** We click **Add Points (4)** and enter the values of the **F** and **V1** variables **(5)** for all the points given in the table below, clicking ENTER after entering each one. The specified strains are for the plastic range. The stress value for strain 0 indicates the yield stress (in this case 380 MPa). After adding the points, set the display range on the graph by either clicking the **Fit** button **(6)** or by entering the appropriate values in the **Min/Max** fields. The resulting curve should look like the image below. **(7)** To return to the model view, choose **Window - Model (8).**



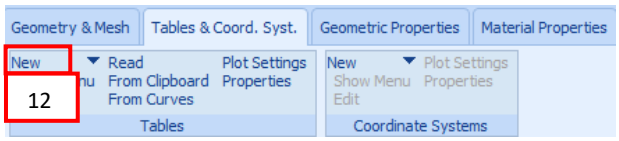
Strain (~) **V1**	0	0.0232	0.048	0.0998	0.15	0.198	0.25
Stress (MPa) **F**	380	430	460	500	530	560	580

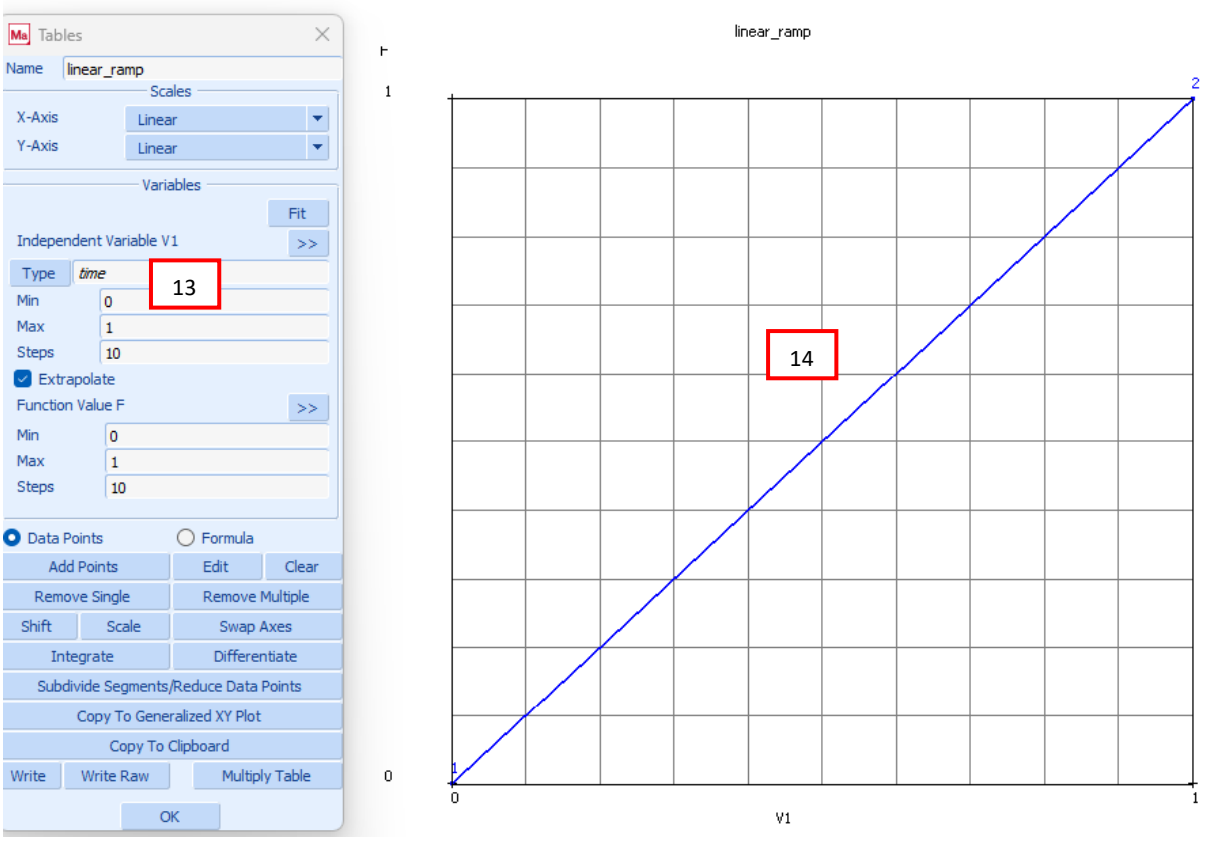


We open the previously defined material properties on the model tree (**Materials - Standard - <name>**). We click **Plasticity (7)** and in **Yield Criterion (8)** we select **von Mises** and in **Method** we select **Table (8).** We click the Table button **(9)** and click the name of the table defined earlier **(10)** and click **OK**. We enter 1 in the multiplier field **(11).** We close the **Material Plasticity Properties** window. If the window is highlighted in yellow in **Material Properties (7)**, it means that the material uses plasticity properties.

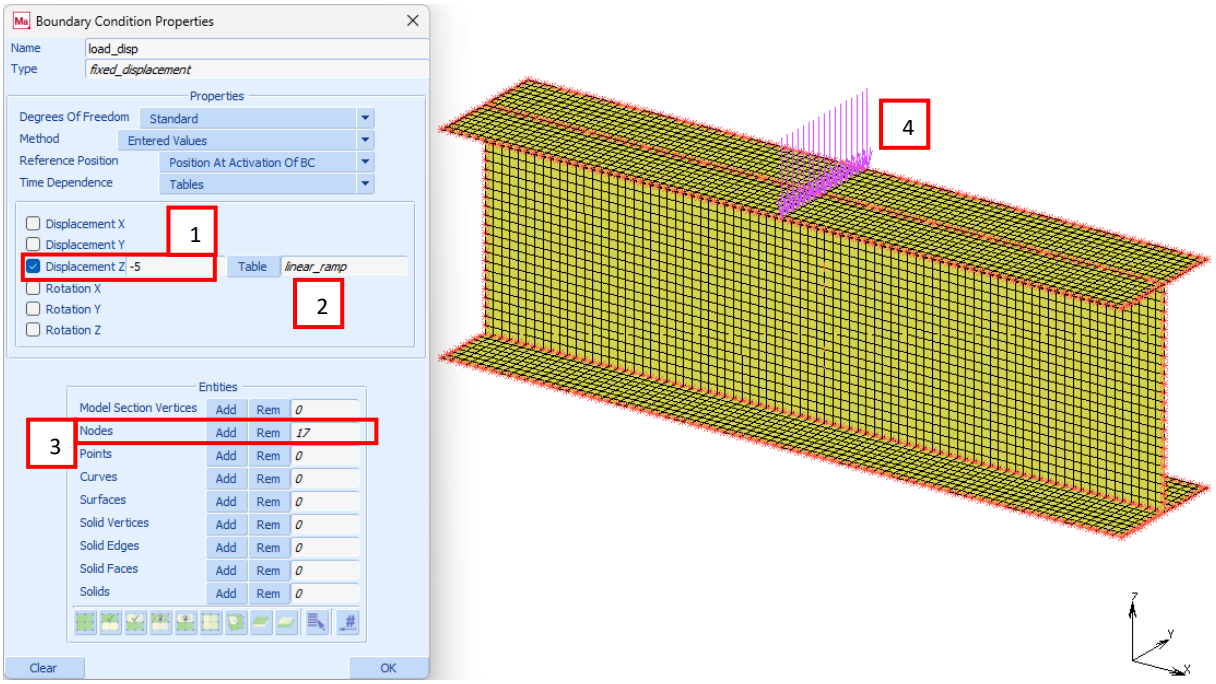


Next, to obtain conditions more similar to the experiment, we define a load in the form of a displacement at the same nodes as point force. To do this, we define another table (**Tables & Coord. Syst. - New - 1 Independent variable (12)).** We give it a name and for **Independent Variable** type **time (13).** Then we add two points with coordinates (0,0) and (1,1). We get a function as shown in the following image **(14)**.

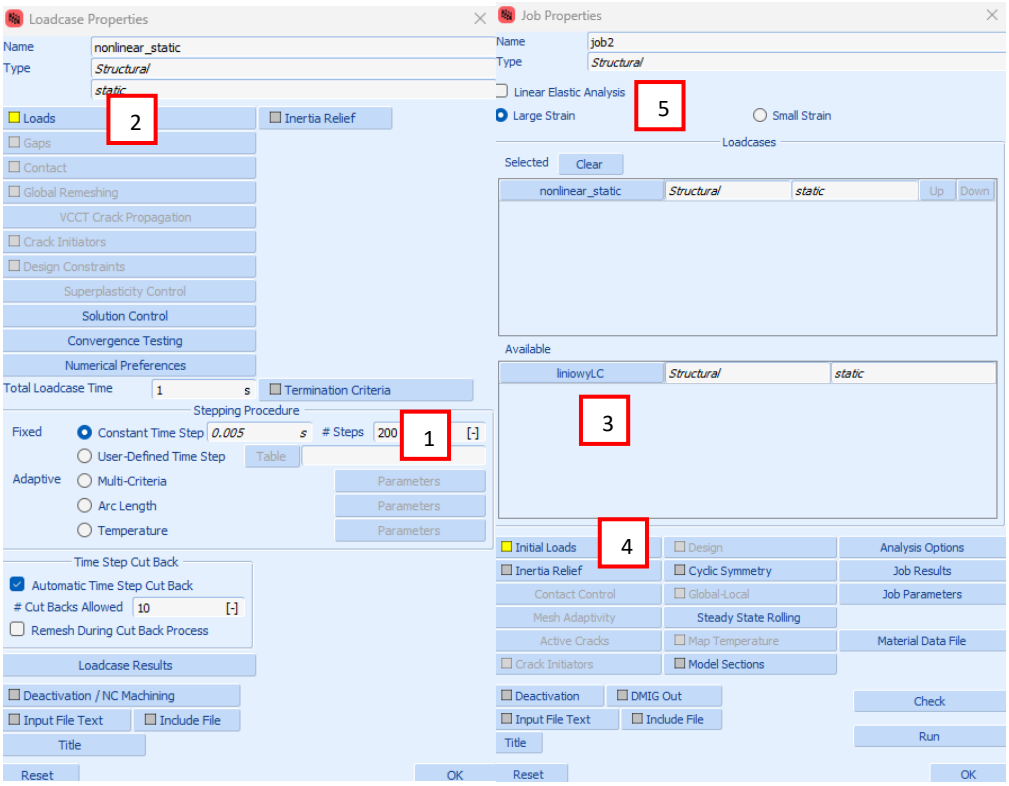




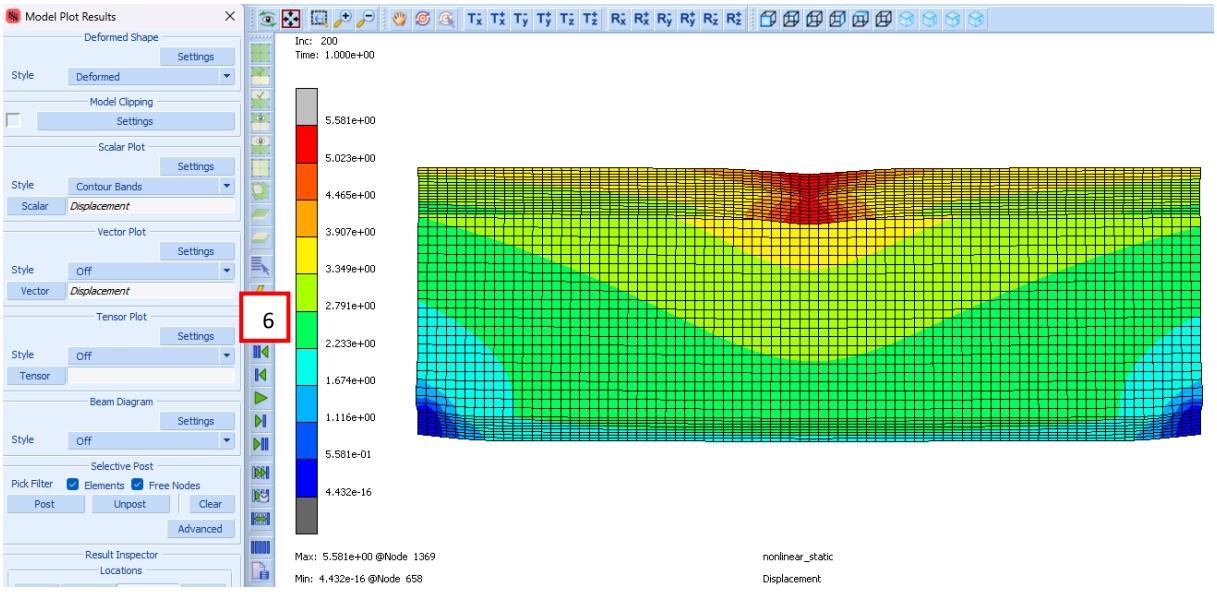
We choose **Boundary Conditions - New - Fixed Displacement.** We select **Displacement Z (1),** enter a value of -5 (mm), click on **Table (2)** and select the previously created linear function. This will make the displacement value grow from 0 to 5 mm during the simulation. We apply the displacement by selecting **Entities - Nodes (3)** and point to all nodes in the middle of the length of the top surface of the beam, as shown in the figure below **(4)**.



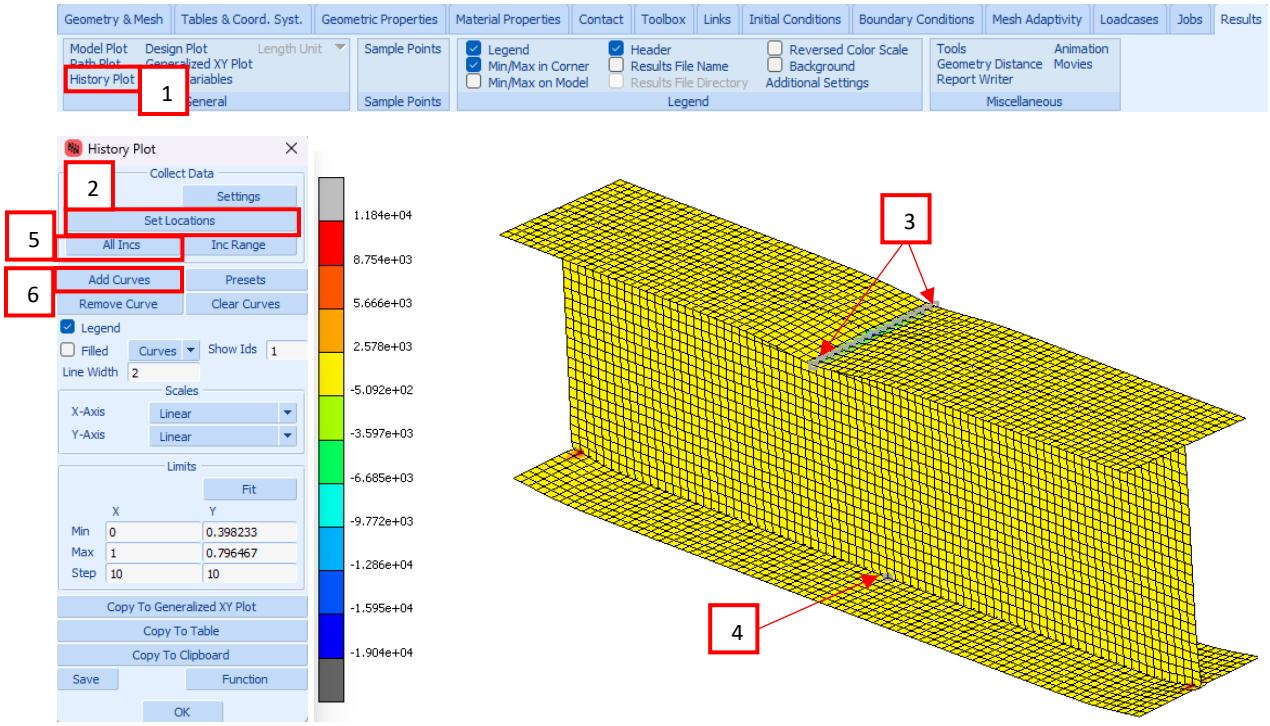
We create a new **Loadcase**. We enter 200 steps to ensure better calculation accuracy **(1)**, in **Loads (2)** we add the newly defined displacement and deselect the force. Similarly, we create or modify the **Job** making sure that the new **Loadcase** is in **Selected** field, and the old **Loadcase** is in **Available (3)** field. We set the same boundary conditions in **Initial Loads (4)** as in the **Loadcase.** We uncheck the **Linear Elastic Analysis** option **(5).** We make sure the **Large Strain** option is checked. We click **Check, Run** and **Submit.**



After loading the results, we can display the change in displacements or stresses over time using the icons on the left side of the model window **(6)**. You can also read the displacements of individual nodes using **Tools - Show Node.**



To create a load-displacement relationship graph of the lower surface of the beam, we choose **Results - History Plot (1).** In the new window, click **Set Locations (2),** select all the nodes on which the load was applied **(3)** and one node on the bottom surface **(4)** on the axis of the set load. **Make a note** of the number of the node on the bottom surface. We click RMB to confirm the selection. We click **All Incs (5)** to add all simulation steps to the display. Then we click **Add Curves (6).**



In the **Add History Plot Curves** window, for the X-axis we choose **Data Carrier Type - Location (Node)**, **Mode - Single (7)** and as **Variable** we set **Displacement (8)** for the node on the bottom surface (in this case number **804**) and for the Y-axis we set **Reaction Force Z** and another node number. We click **Add Curve (9)** to add a curve to the graph. We create curves for each node from the top plane, so that all curves are in one graph. To adjust the range of values on the graph axes, we click **Fit.** We should get a graph like the one below **(10)**. In the **History Plot** window, we click **Function (11)** and in the new window click **Add (12)** and then **All (13).** We click **Fit** to display the result of adding charts. For more clarity, we can remove the remaining graphs by clicking **Remove Curve (14)** (but not **Clear Curves!**), and then clicking on the individual curves. The resulting curve can be compared with the force-displacement relationship graph from the article **(15)**.

