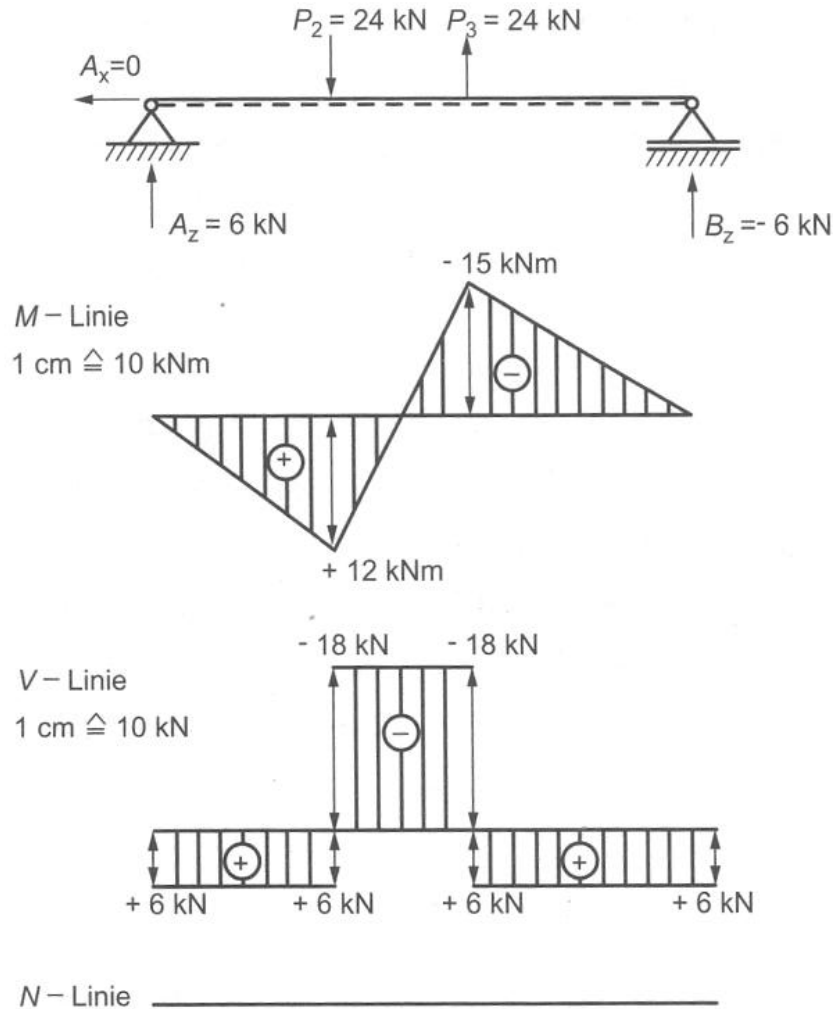


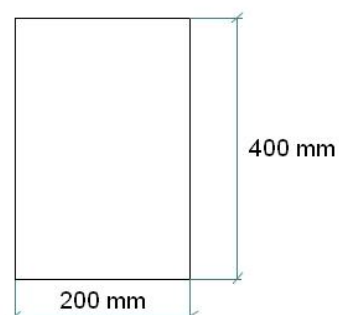
## Part 31 - Calculation of the Internal Forces from the Bending and Shear Stresses of a 3D beam

A 400 mm x 200 mm rectangular 3D steel beam with a length of  $L = 6000$  mm is loaded to two opposing nodal loads of 24 kN at  $l_1 = 2000$  mm and  $l_2 = 3500$  mm. How large are the bending and shear stresses and for the new reinforced concrete design (see Part 32) the internal forces with the maximum bending moment and the maximum shear force.

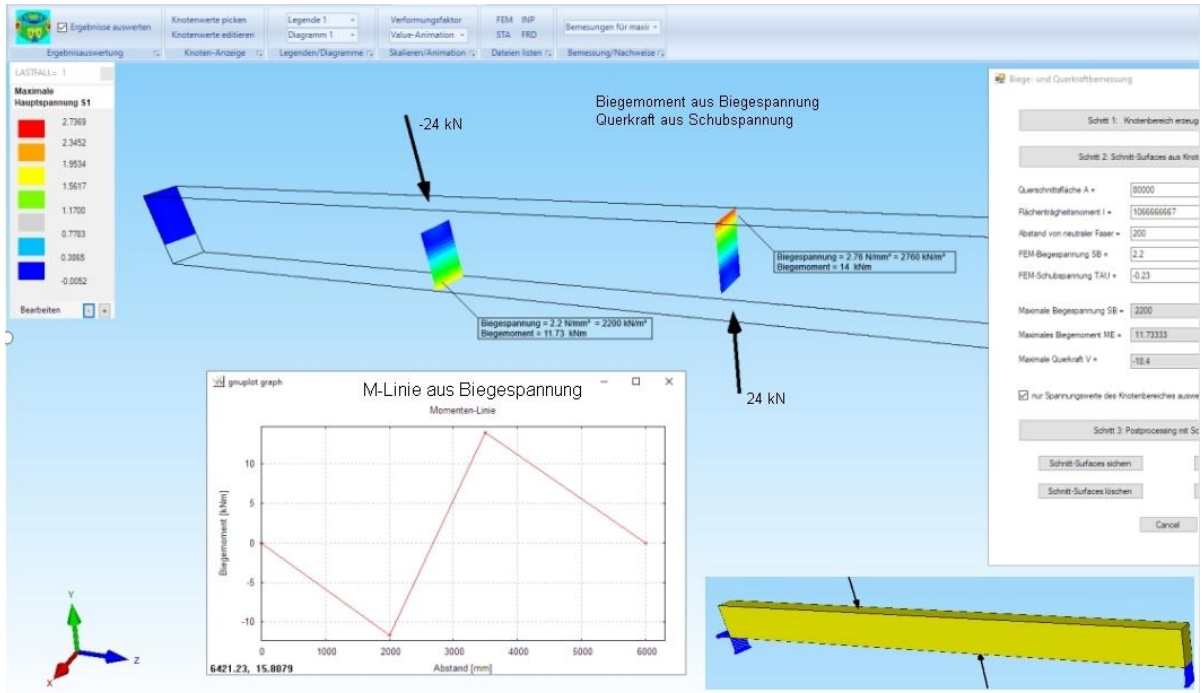
### Internal forces of a 2D beam:



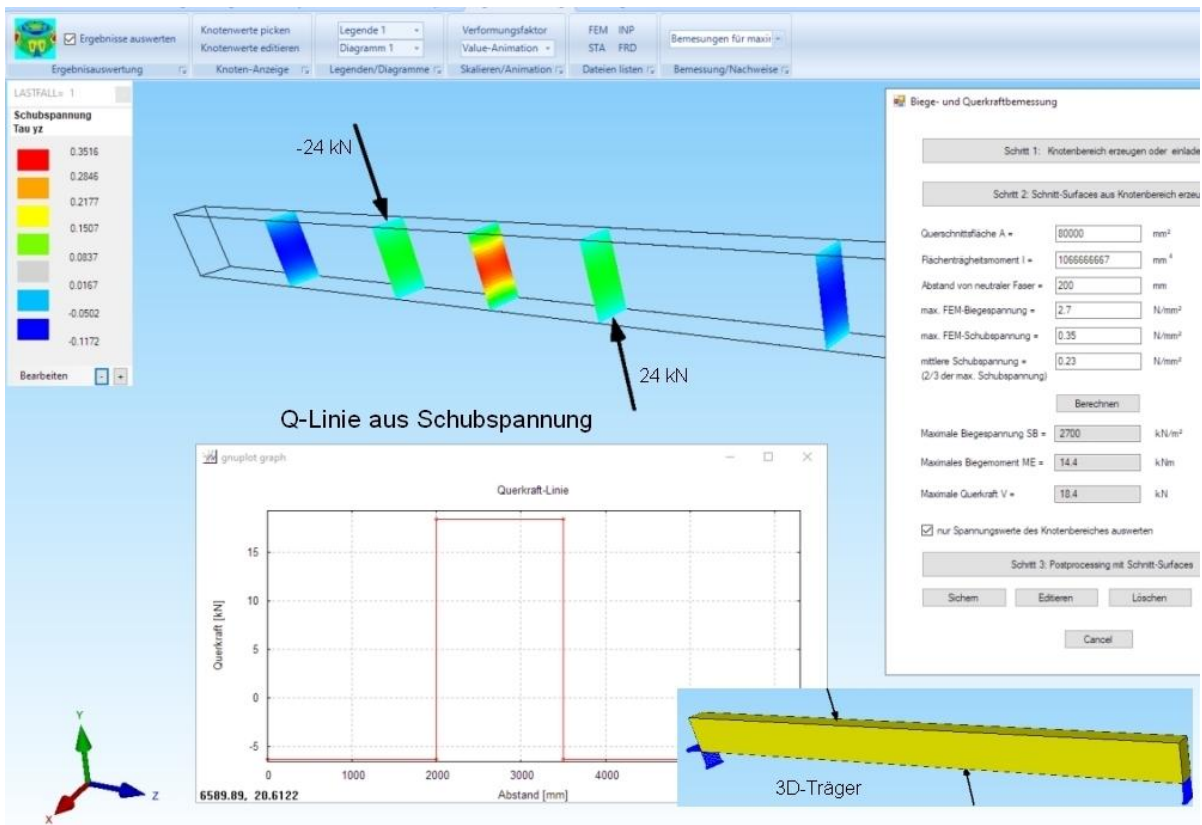
### Rectangle Profile:



### Calculation of the Moment Line from the FEM Principal Stresses

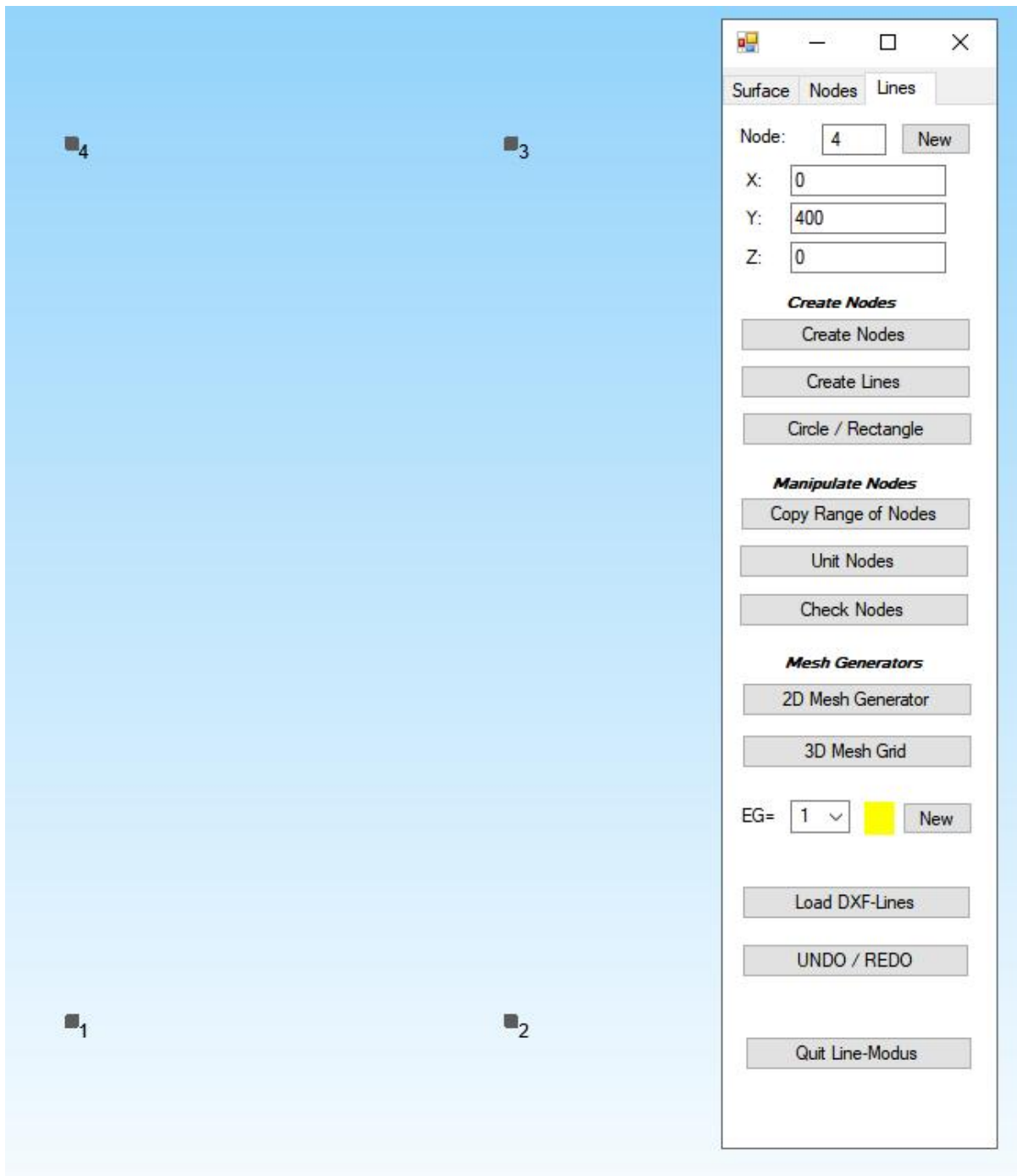


### Calculation of the Shear Force Line from the FEM Shear Stresses

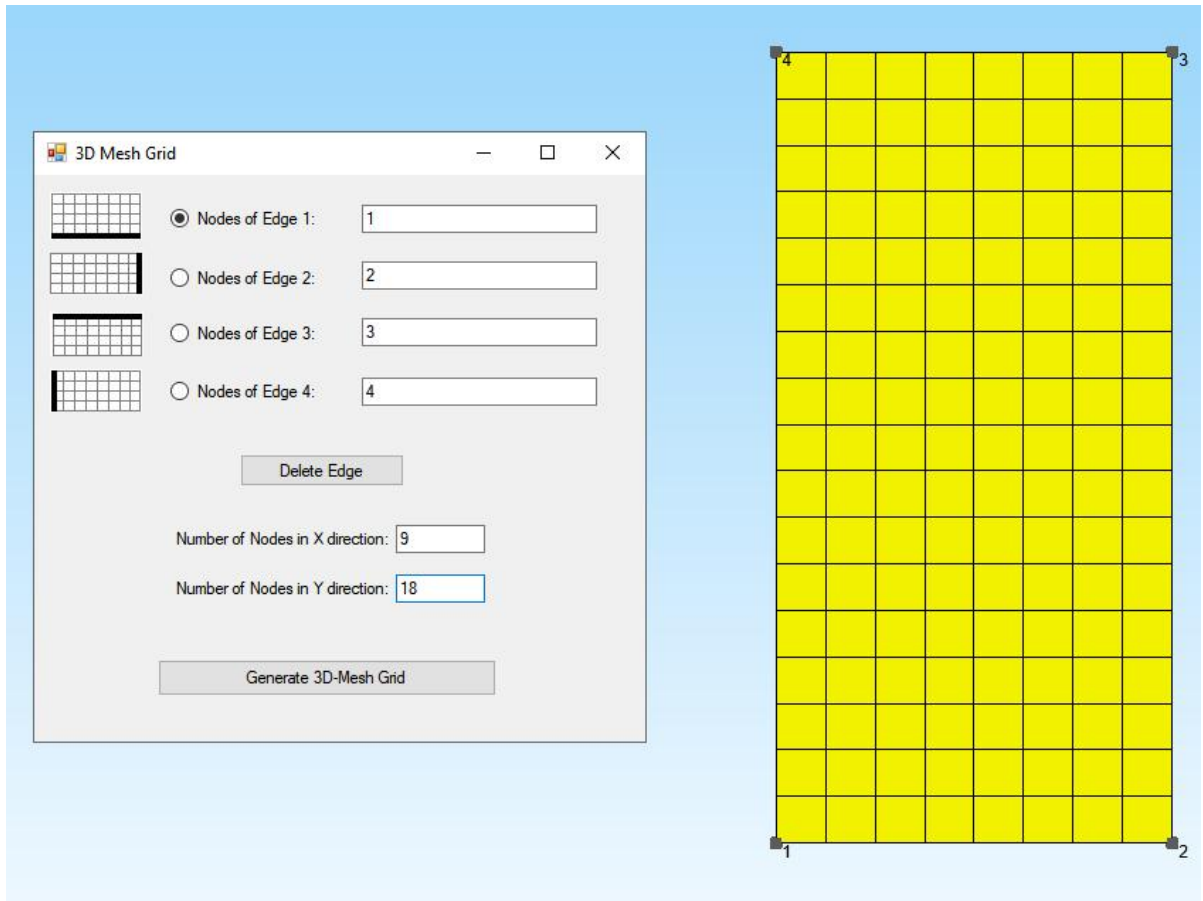


## Generate 3D beam

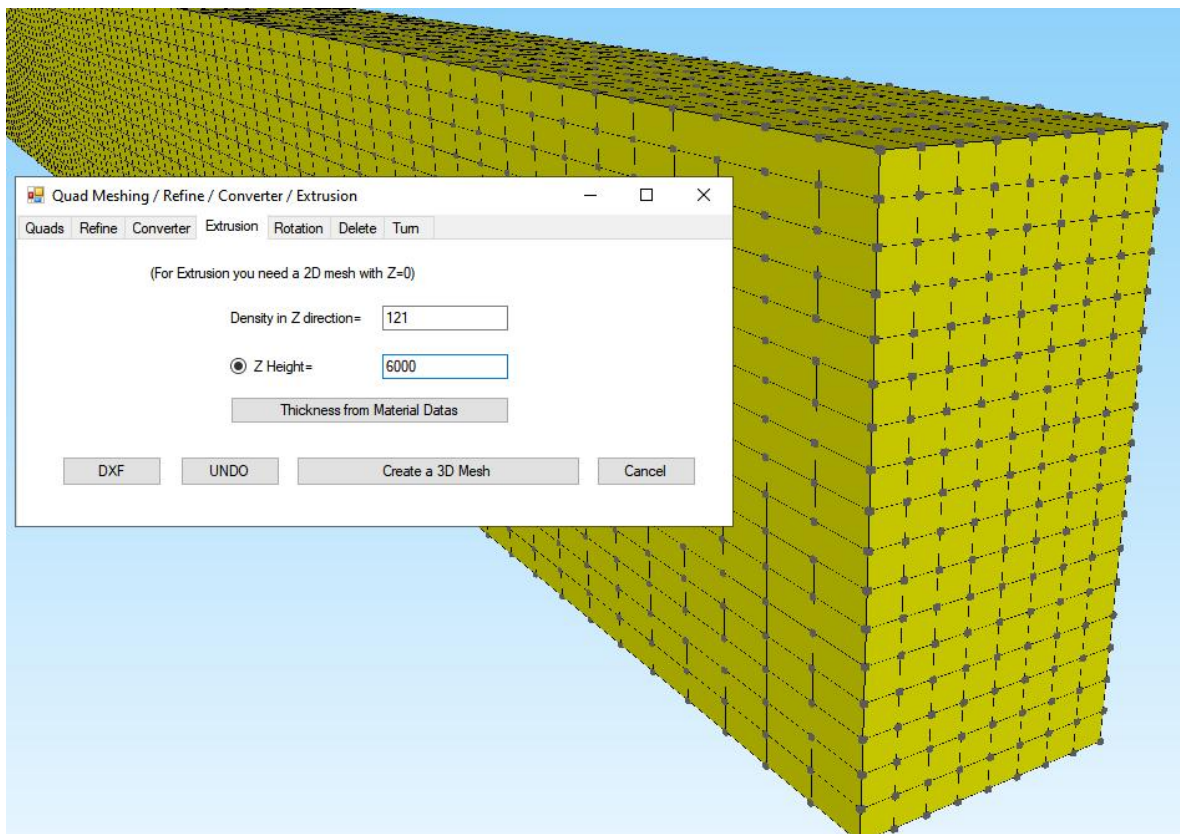
First create a 200x400 rectangle with the following 4 nodes with "New" and with „2D/3D Beam Model with Line-Modus“ the 4 nodes 0/0, 200/0, 200/400, 0/400.



Then generate a 2D mesh with the "3D Mesh Grid" menu with a mesh density in X direction = 9 and in Y direction = 18.

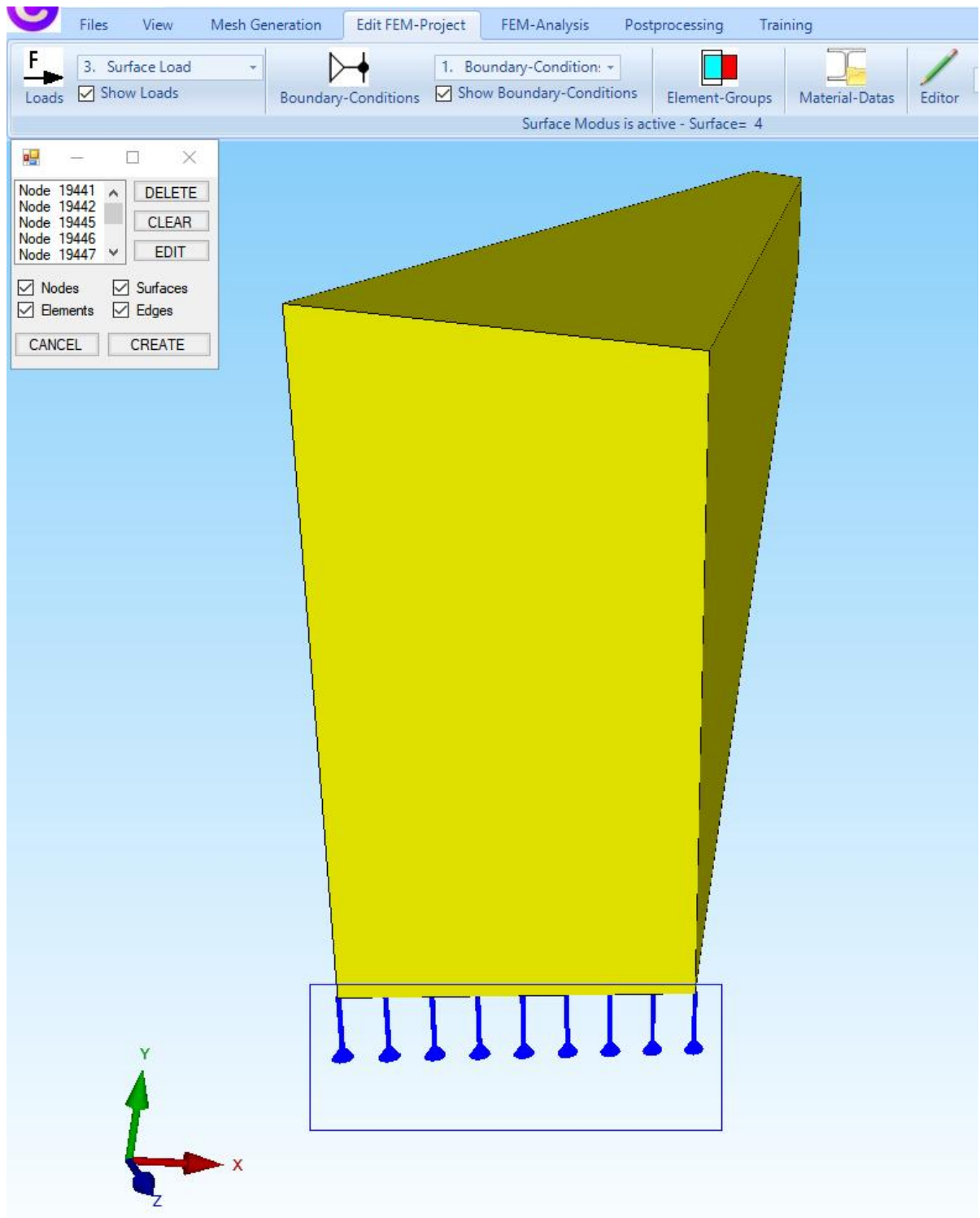


Then extrude using the "Mesh Generation" tab, menu "Quad Meshing/Refine ..." and „Extrusion“ with the mesh density in Z direction = 121 and a Z-Height = 6000 a 3D mesh with 16 320 HEX8 Solid-Elements and 16 320 nodes.



## Create a floating bearing edge with Surface Modus

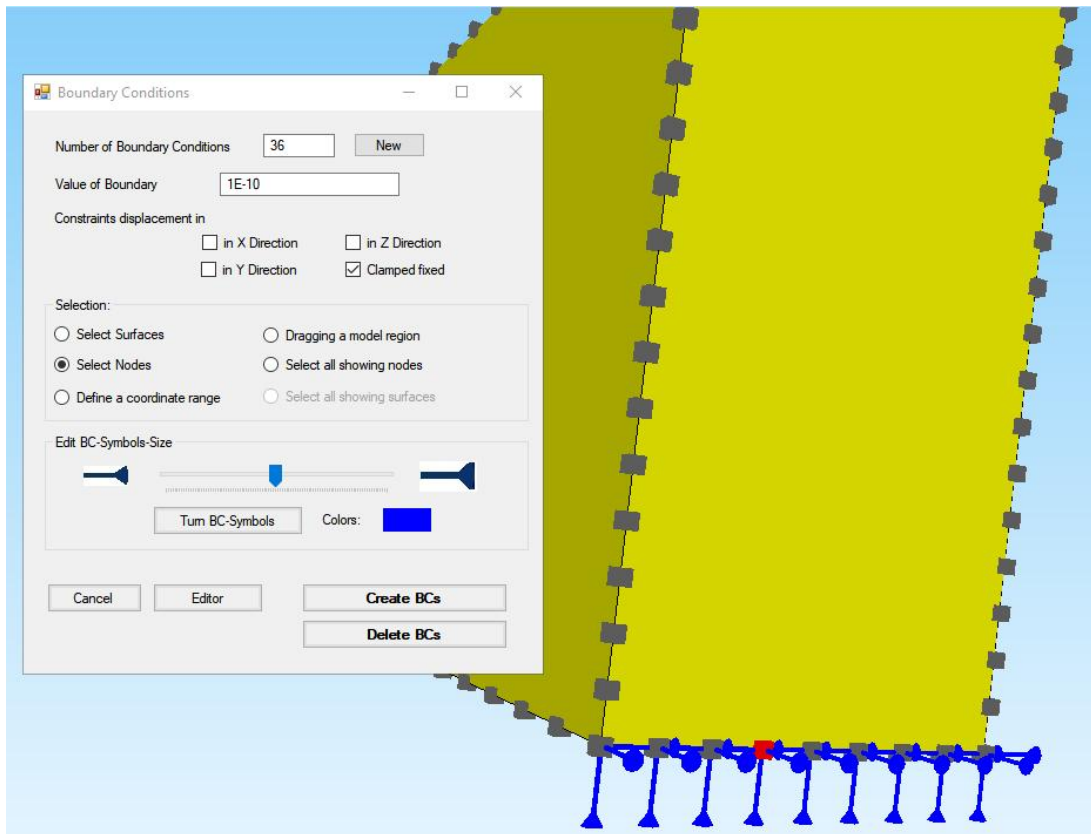
Turn the model to the right and diagonally downwards and use the "Edit FEM Project" tab and the "Boundary Conditions" menu to clamp the lower right edge with "in Y-direction" and "Dragging a model region".





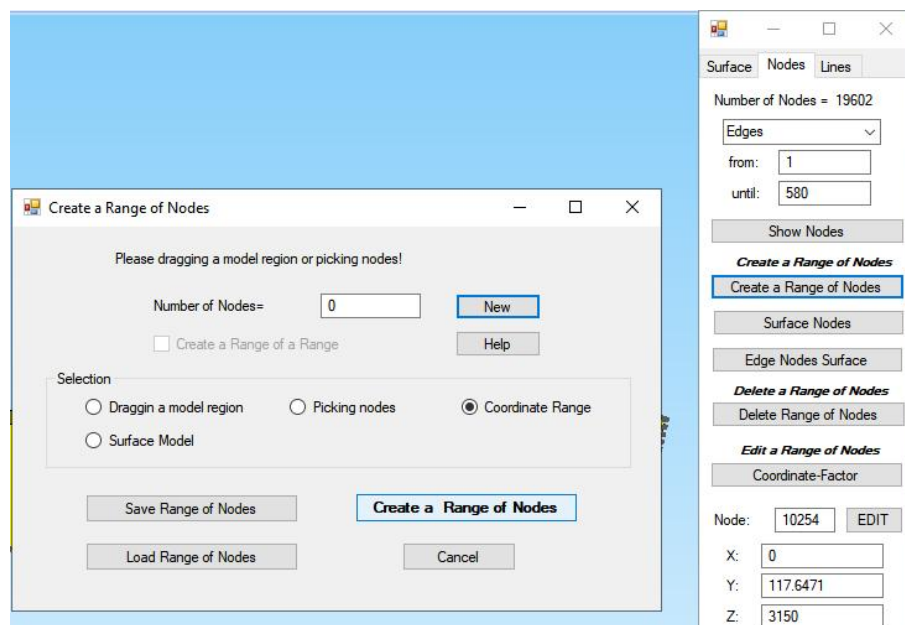
## Create clamped fixed edge with Node-Modus

First switch on the Node-Modus with the "View" tab to display all edge nodes. Select the "Edit FEM Project" tab again and "Boundary Conditions" in order to use "Clamped fixed" and "Select Nodes" to clamp fixed the left edge in all directions.

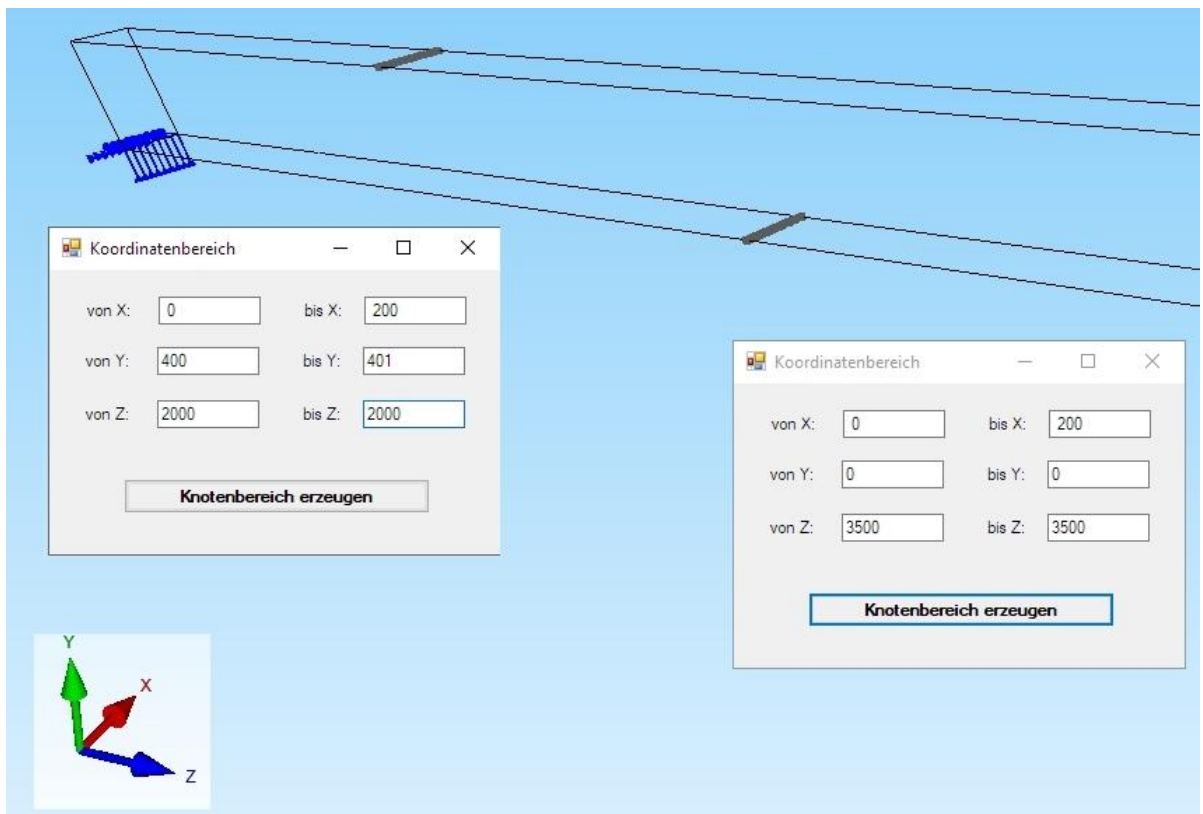


## Create loads with a Coordinate Range

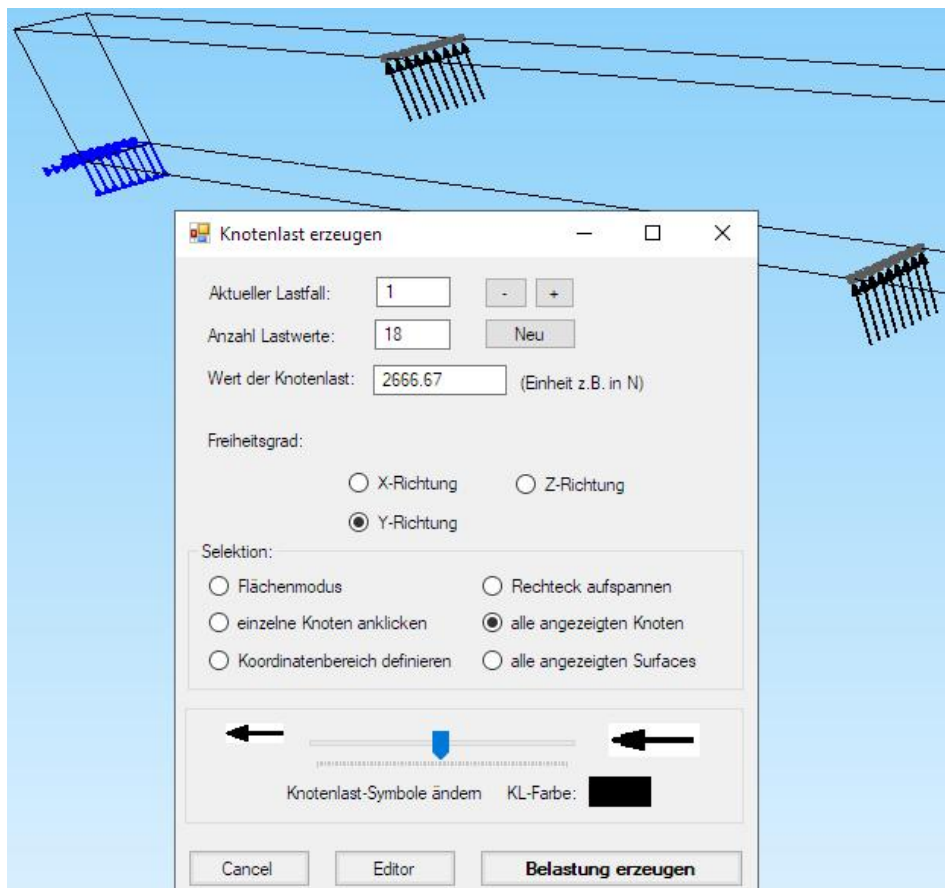
Activate "View" and "Node-Modus" tabs and menu „Create a Range of of Nodes“



to create two Coordinate Ranges at  $Z=2000$  and  $Z=3500$ .



Select „Wireframe-Modus“ to displayed 18 nodes and select the "Edit FEM Project"



and "Point Load" tabs and generate a Point Load with the value = 2666.7 in "Y-direction" with "all displayed nodes", whereby the value is calculated from the total load  $2 \times 24,000 \text{ N}$  divided by 18 nodes. Finally, the sign of the last 10 - 18 loads must be rotated against the Y-Axis using "Editor" and the "Change FHG" menu.

The screenshot displays the 'Edit Loads' dialog box in a software application. The dialog box is divided into several sections:

- Table:** A table with columns 'Nr.', 'Node', 'FHG', and 'Value'. It lists 18 nodes with their respective values and FHG settings.
- Configuration:** Fields for 'Load Case' (1), 'Number of Loads/Load Case' (18), and 'Load Type' (1 Point Load).
- Buttons:** 'New Load Case', 'Delete Load Case', 'Load Factor', 'Pressure->Point Load', 'Point Load->Line Load', 'Combine Load Cases', 'Copy Load Case', 'Convert Temperature to a Load Case', and 'Change FHG' (highlighted in blue).
- Advanced Options:** 'Edit Degrees of Freedom' (Freedom old: 0, Freedom new: 1) and 'Edit Value' (Value: 2666.67, Value new: -2666.67).
- Final Options:** 'Delete all Nodes define in the Range above:' and 'Check for duplicate Nodes in the Point Load'.

The background shows a 3D beam model with a load distribution. The 'Edit Loads' dialog box is open, and the 'Change FHG' button is highlighted in blue. A 3D coordinate system (X, Y, Z) is visible at the bottom left.

Nr.	Node	FHG	Value
8	11350	2	2666.67
9	11351	2	2666.67
10	6483	2	-2666.67
11	6484	2	-2666.67
12	6636	2	-2666.67
13	6637	2	-2666.67
14	6638	2	-2666.67
15	6639	2	-2666.67
16	6640	2	-2666.67
17	6641	2	-2666.67
18	6642	2	-2666.67

## FEM-Analysis

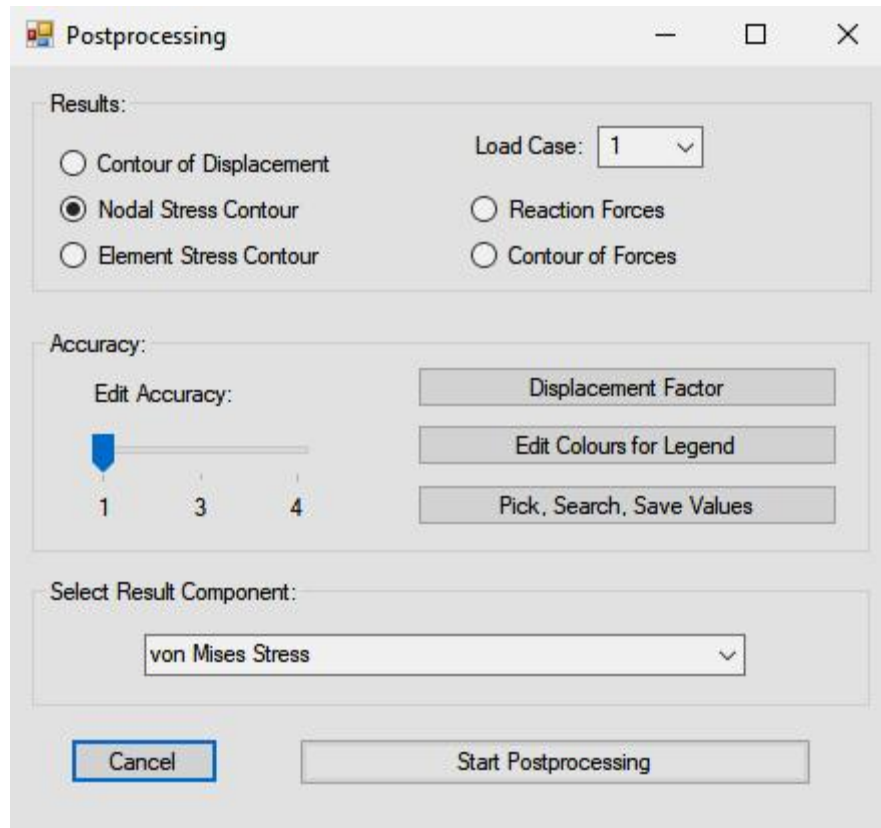
The material data for steel are preset and therefore do not have to be entered. Now save the 3D beam with a name on the hard disk and use the "FEM Analysis" tab and the Quick Solver to calculate the results such as displacements, stresses and reaction forces.



## Postprocessing

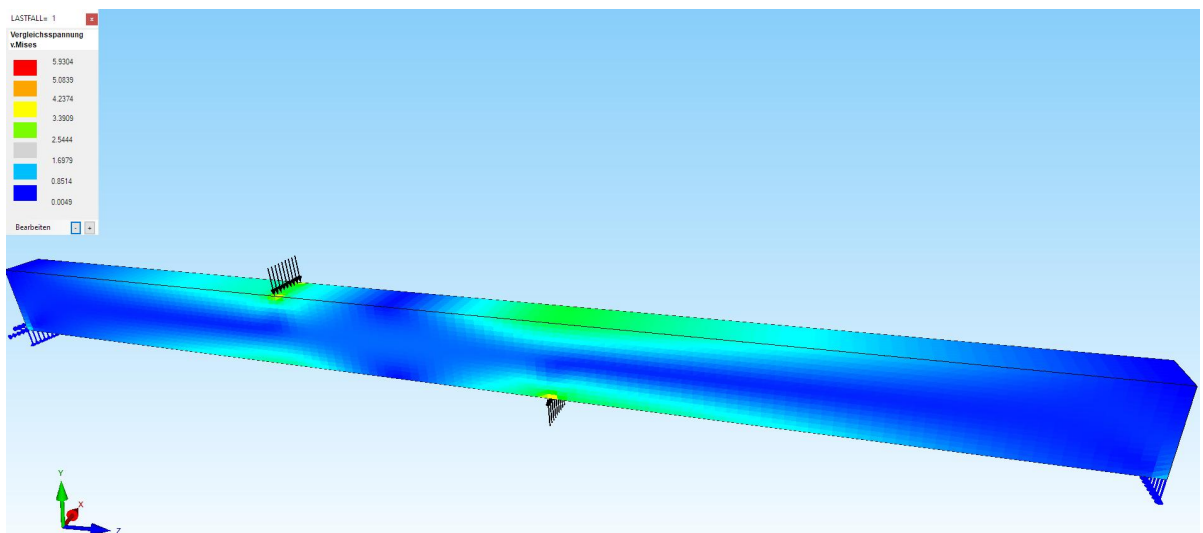


Select the "Postprocessing" tab and the icon to evaluate the stresses.



### v.Mises Stress

A max. v.Mises Stress of 5.9 N/mm<sup>2</sup> is displayed, but this value is too high due to the punctiform node load.

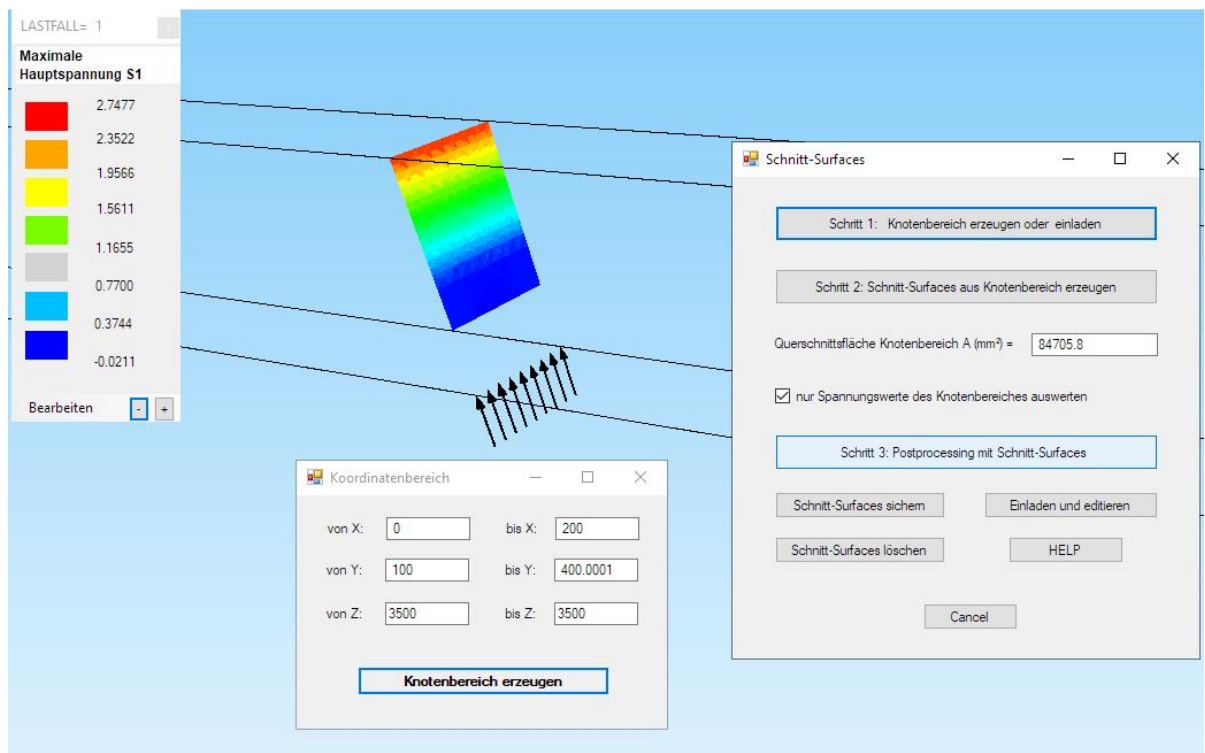


## Create Cut Surfaces

These secondary stresses can be hidden with a Range of Nodes. To do this, select the "Cut-Surfaces" menu in the Surface-Modus and define a Coordinate Range at the point X= 0 - 200 / Y= 100 - 401 / Z = 3500 - 3500 above the nodal load. With Step 2 you create the section surfaces, which can then be displayed with Step 3 as local stress distribution.

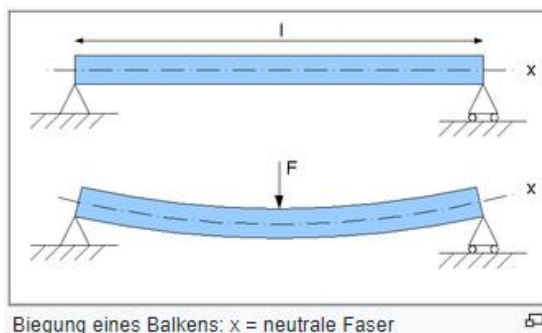
## Maximum Bending Stress S1

The maximum Bending Stress S1 is **2.75 N/mm<sup>2</sup> bzw. 2750 kN/m<sup>2</sup>**



## Maximum Bending Moment

If the maximum Bending Stress is multiplied by the Area Moment of Inertia and divided by the distance of 200 mm between the Neutral Faser, the maximum Bending Moment is **14.7 kNm**.



$$\sigma = \frac{M}{I} \cdot z$$

$$M = \frac{\sigma}{z} \cdot I$$