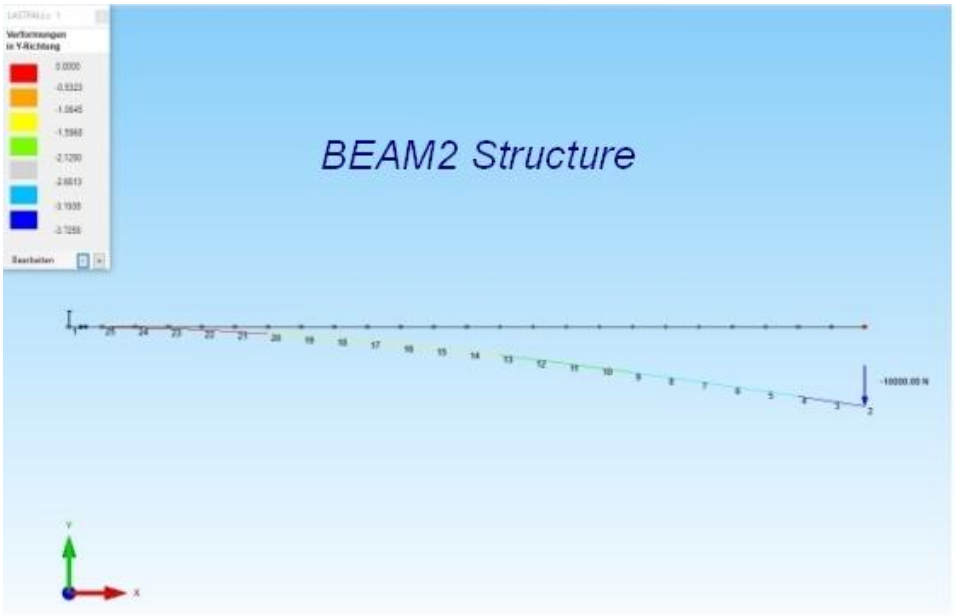
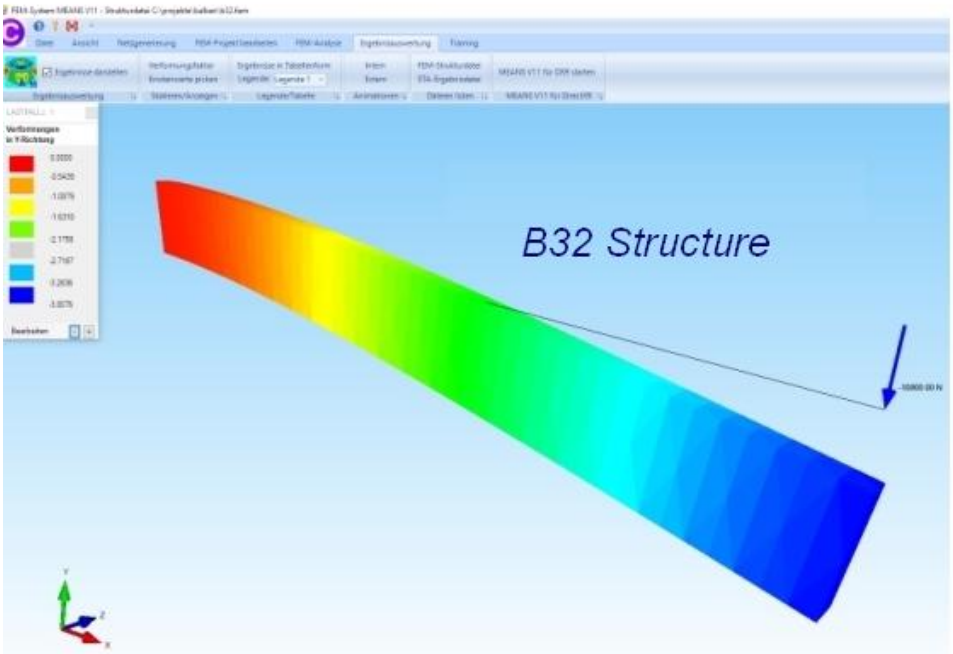
FEM-System

MEANS V12

BEAM-Analysis for
Calculation of Displacements and Stresses





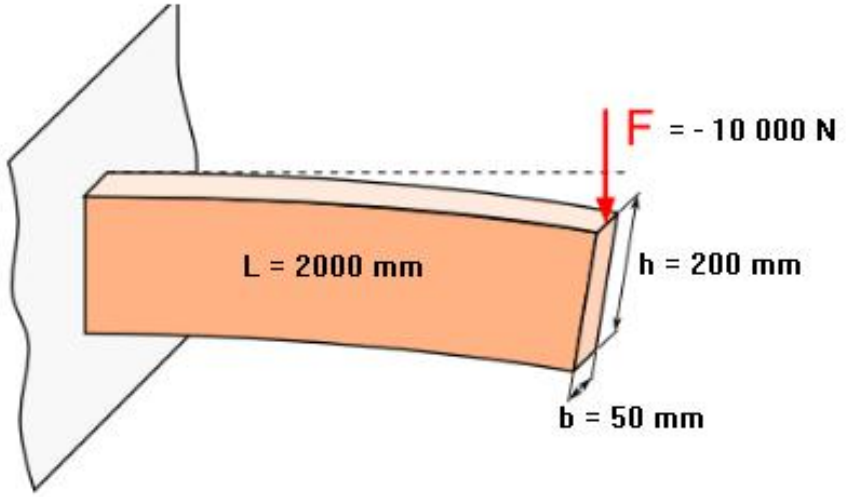
Ing.Büro HTA-Software

www.femcad.de

www.fem-infos.com

Part 6: BEAM-Analysis with MEANS V12

Example 1: Cantilever Beam



$$f = \frac{F \cdot l^3}{3 \cdot E \cdot I} = \frac{-10000 * 2000^3 * 12}{3 * 210\,000 * 50 * 200^3} = -3.809 \text{ mm}$$

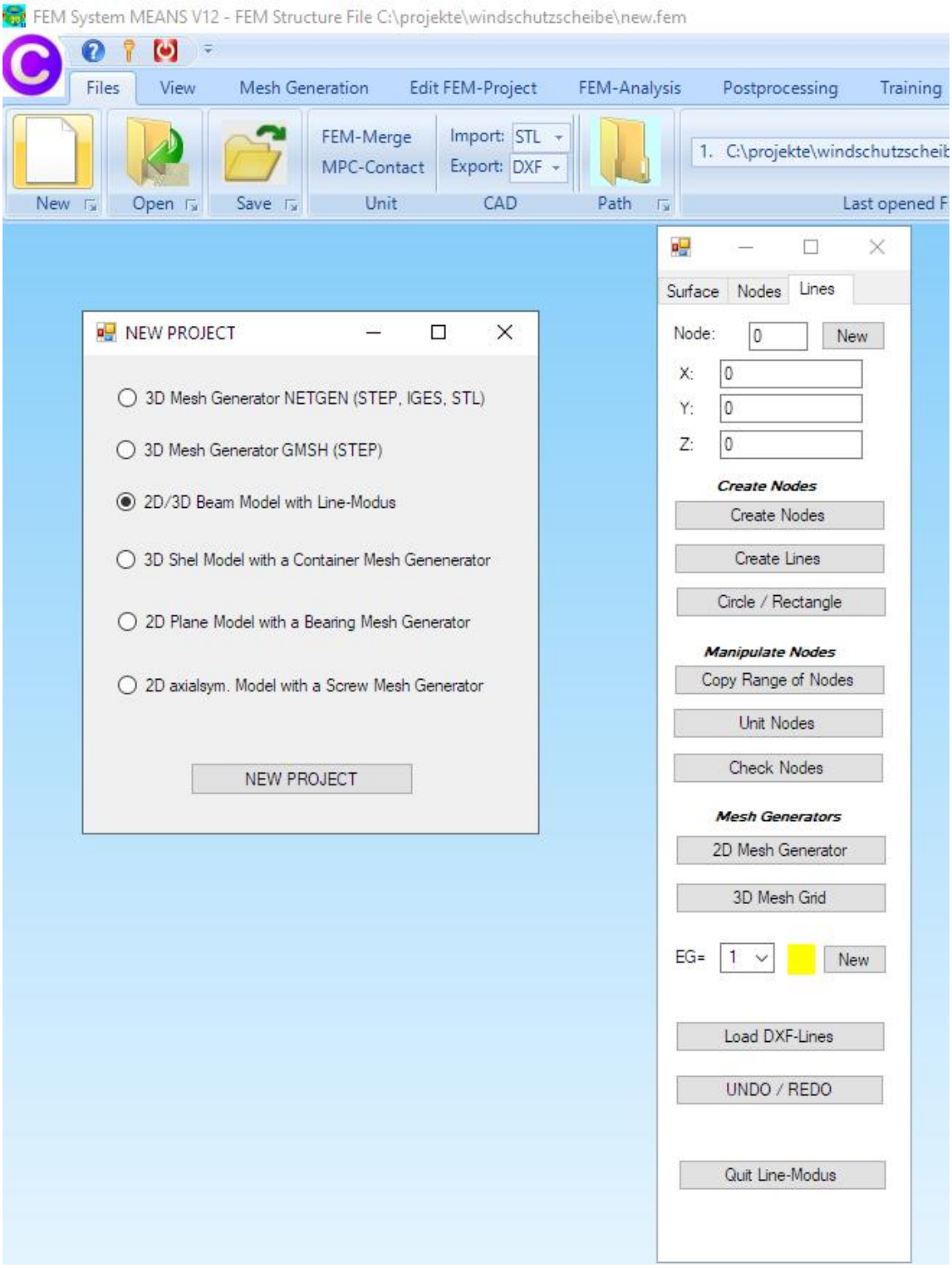
$$\sigma_b = \frac{M_b}{W} = \frac{-10000 * 2000 * 6}{50 * 200^2} = 60 \text{ N/mm}^2$$

Exact Results:
- max. Displacements (x = 2000 mm) = - 3.8 mm
- max. Bending Stress (x = 0) = 60 N/mm²

Create bars in line mode

Start MEANS V12 via the Windows desktop icon and select the tab "File" and menu "New" and click on the project menu "Create a new Model with Beam-Line-Modus" The dialog box for the line mode appears on the right side.

In the Line-Modus here you will find menus containing e.g. Nodes, Rectangles or Circles can be created and connected.



Input of the nodes

Input node 1

Click on "New" and enter X = 0, Y = 0, Z = 0

Click on "Create Nodes" to create node 1.

Input node 2

Click on "New" and enter X = 2000, Y = 0, Z = 0

Click on "Create Nodes" to create node 2.

Now both nodes should be numbered on the screen.

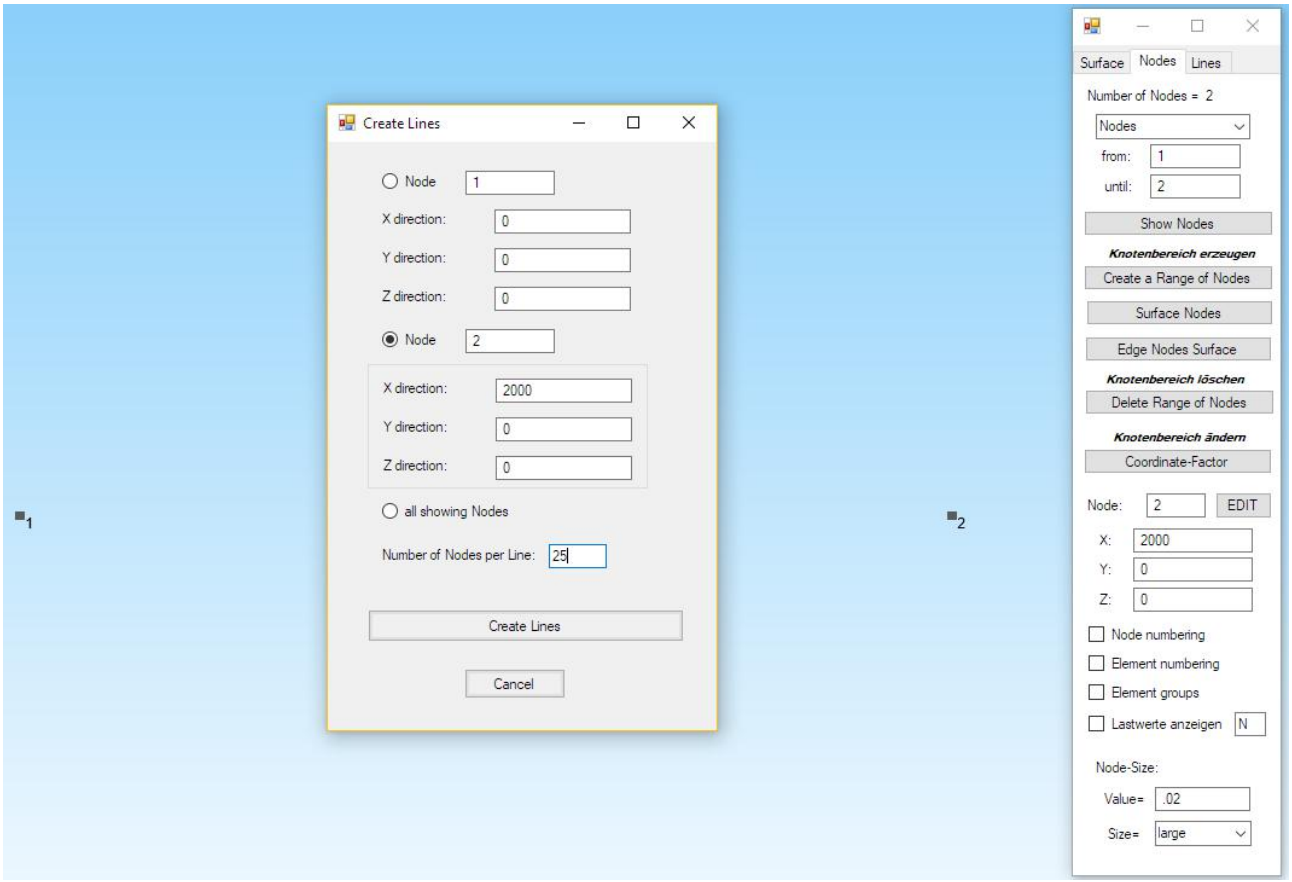
Create beam elements

Select the menu "Create Line" and create 24 beam elements.

With this dialog box, two nodes clicked on the screen can connected to a beam element.

First activate the upper point option and click on the left node so that the node number 1 with the coordinates is displayed.

Then activate the lower point option and click on the right node so that the node number 2 with the coordinates is also displayed.

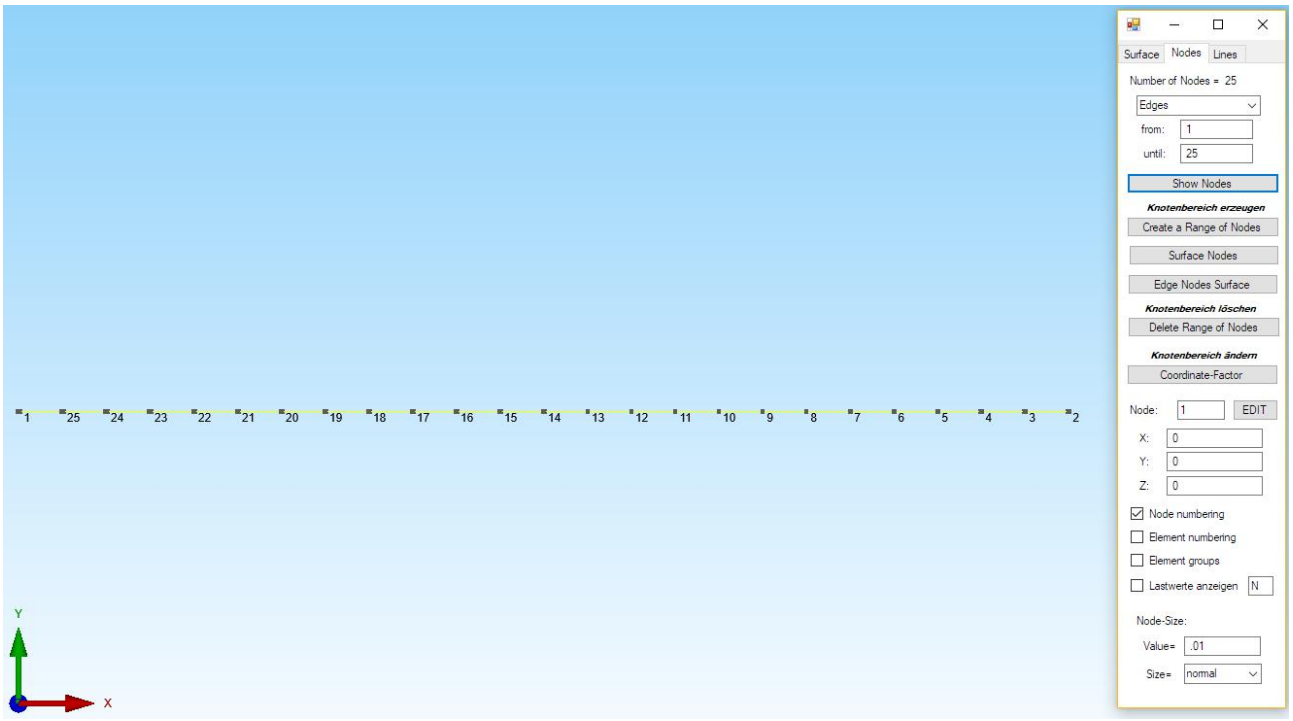


Refinement

Enter the Number of Nodes per Line = 25 so that 24 beam elements are generated.

Create beam structure

Select menu "Create Lines" to generate a beam structure of 25 nodes and 24 BEAM2 elements.

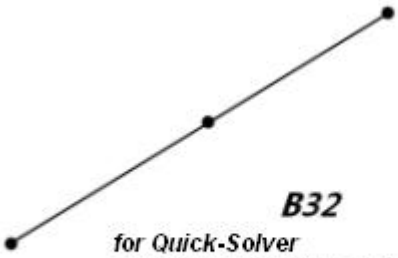


Input of the Material and Profile Datas

Select the "Edit FEM Project" tab and menu "Material-Datas" menu. There, select the menu "Create Profile Sections" to enter the Material and Profile Datas for BEAM2 or B32 elements.

BEAM2

for Dr.Kühn-Solver
for all profiles
for linear Statics and Dynamics

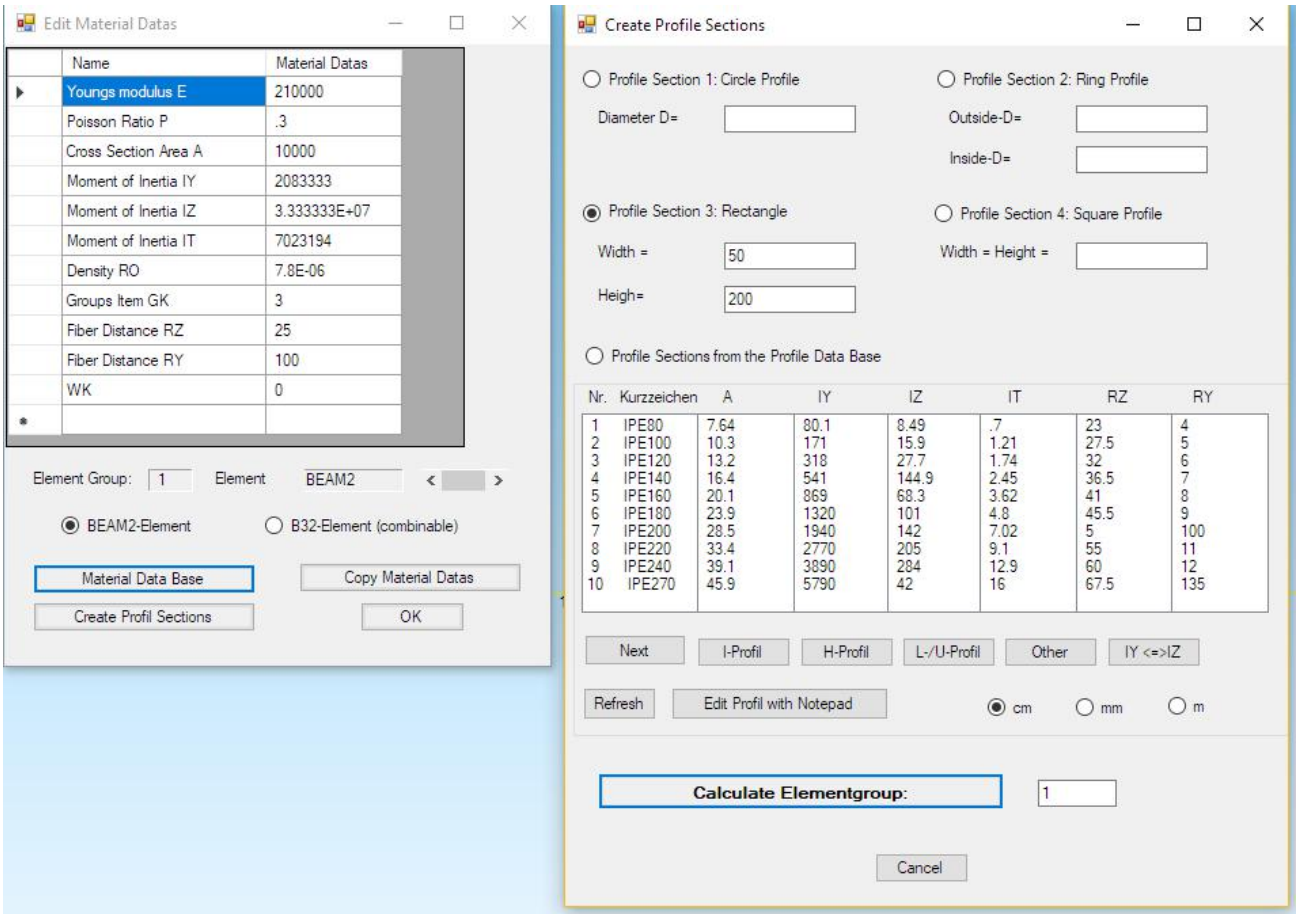


B32

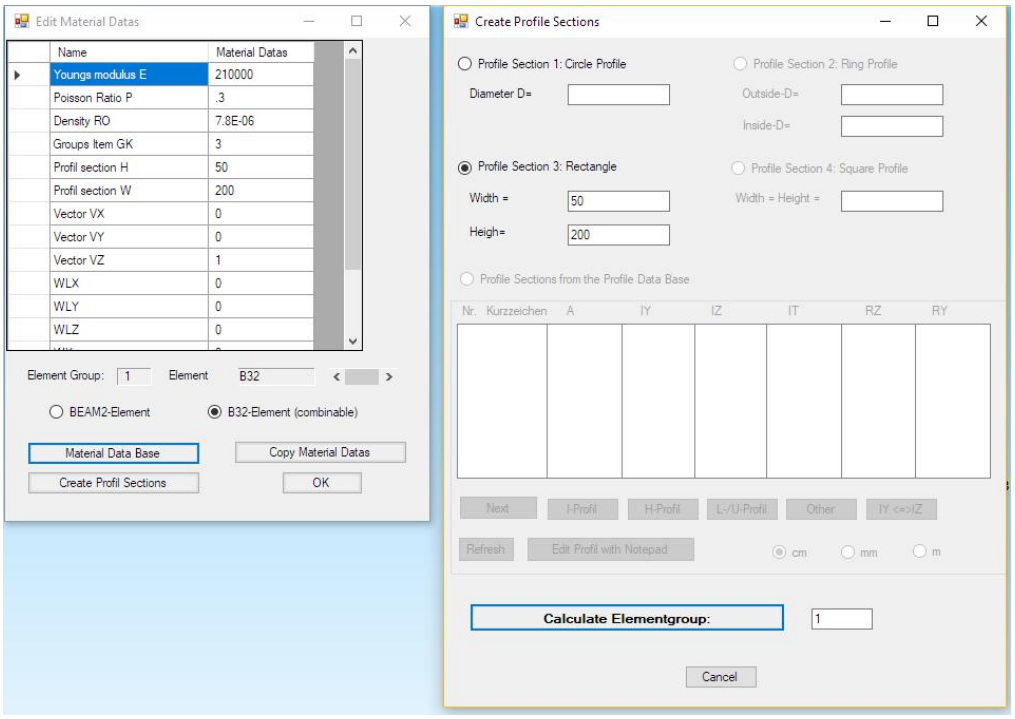
for Quick-Solver
only for rectangular and circle profiles
for Statics, Dynamics, Buckling,
Nonlinear, Temperature

In the dialog box for BEAM2 Profiles, select "Profile Section 3: Rectangle" and enter a Width of 50 mm and a Height of 200 mm, then select the bottom "Create Element group 1" to generate the material and profile datas.

Material Datas for **BEAM2** Elements with Circle-, Ring-, Rectangle- or Square Profile and a own Profil Data Base in the ASCII-Format for special profile as IPE or HEB which can be edited or extended even with Notepad:



Material Datas for **B32** Elements only with a Circle- and a Rectangle-Profil



Consider Units of Beam elements

Especially with the beam structures, you have to consider the correct units:

Coordinates are in millimeters -> then profile and material data also in millimeters

X = 2000 mm, Y = 0 mm -> E-Modulus = 210 000 N / mm², H = 50 mm, B = 200 mm

Coordinates are in meters -> then profile and material data also in meters

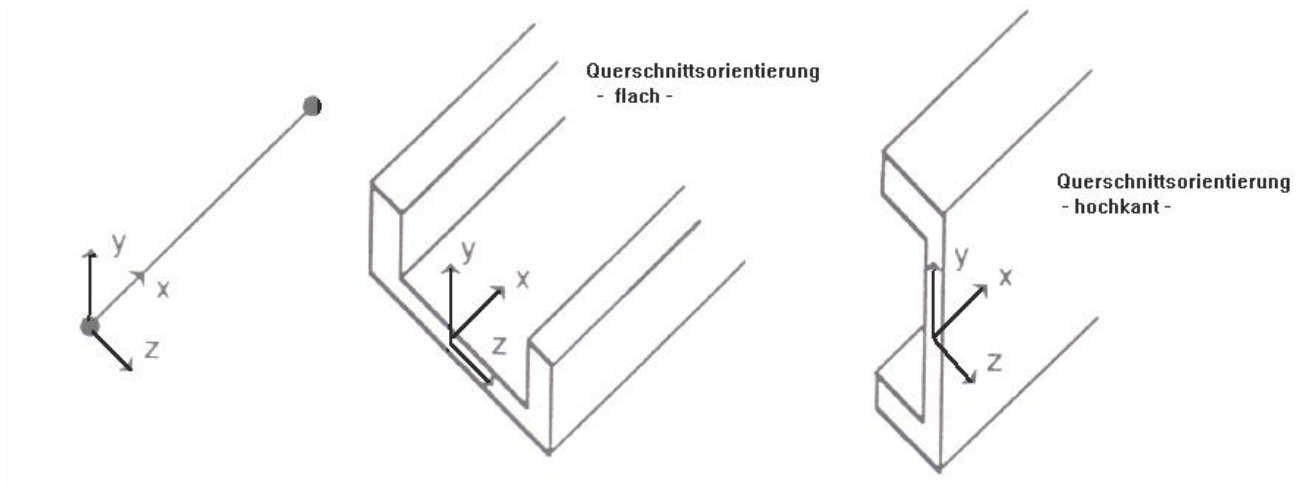
X = 2 m, Y = 0 m -> E-Modulus = 210 000 000 000 N / m², H = 0.05 m, B = 0.20 m

Bending stress

With beam elements, almost all structures can be calculated with sufficient accuracy with relatively little mesh generation effort, provided that the cross-sectional area and the moments of inertia can be easily determined.

However, if the profile has an irregular cross-sectional shape, it is very difficult to determine, in particular, the torsional moment of inertia, and then the structure can only be calculated accurately with either area or volume elements.

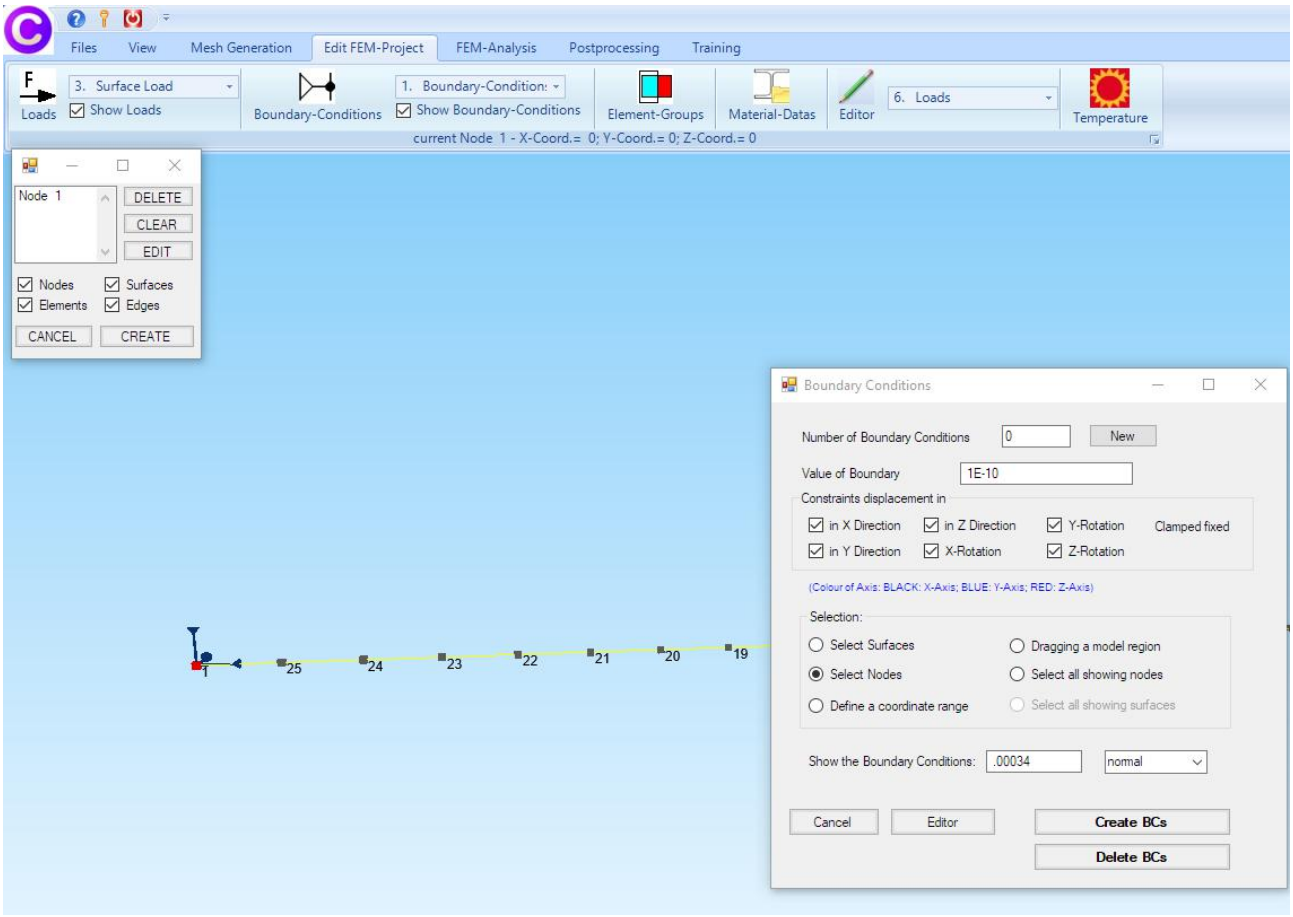
Since the beam elements on the screen can only be represented as a line, the orientation of the cross section - very important in bending stress - can not always be optically controlled, so that the transverse and vertical axes can be easily reversed.



Create Boundary Conditions

The cantilever beam is fixed clamped at the left end. Create the BCs in the following steps:

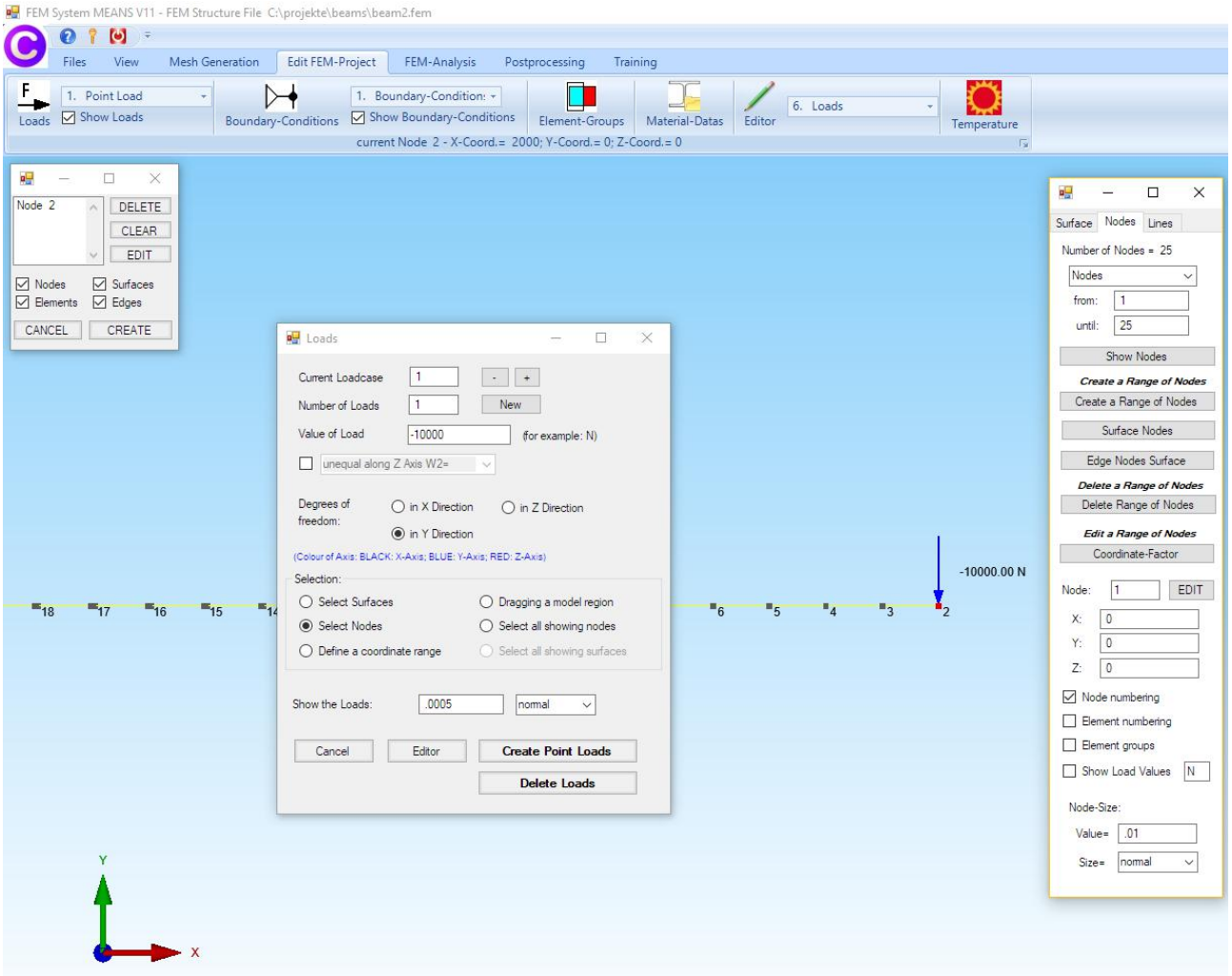
- Select the "Edit FEM Project" tab and click on the BC icon to enter the boundary conditions.
- In the BC dialog box, select "Clamped fixed" and "Create BCs" and click on node 1.
- Node 1 is displayed in the Selectbox, there you select "Create" to create the RBs.



Create a Point Load

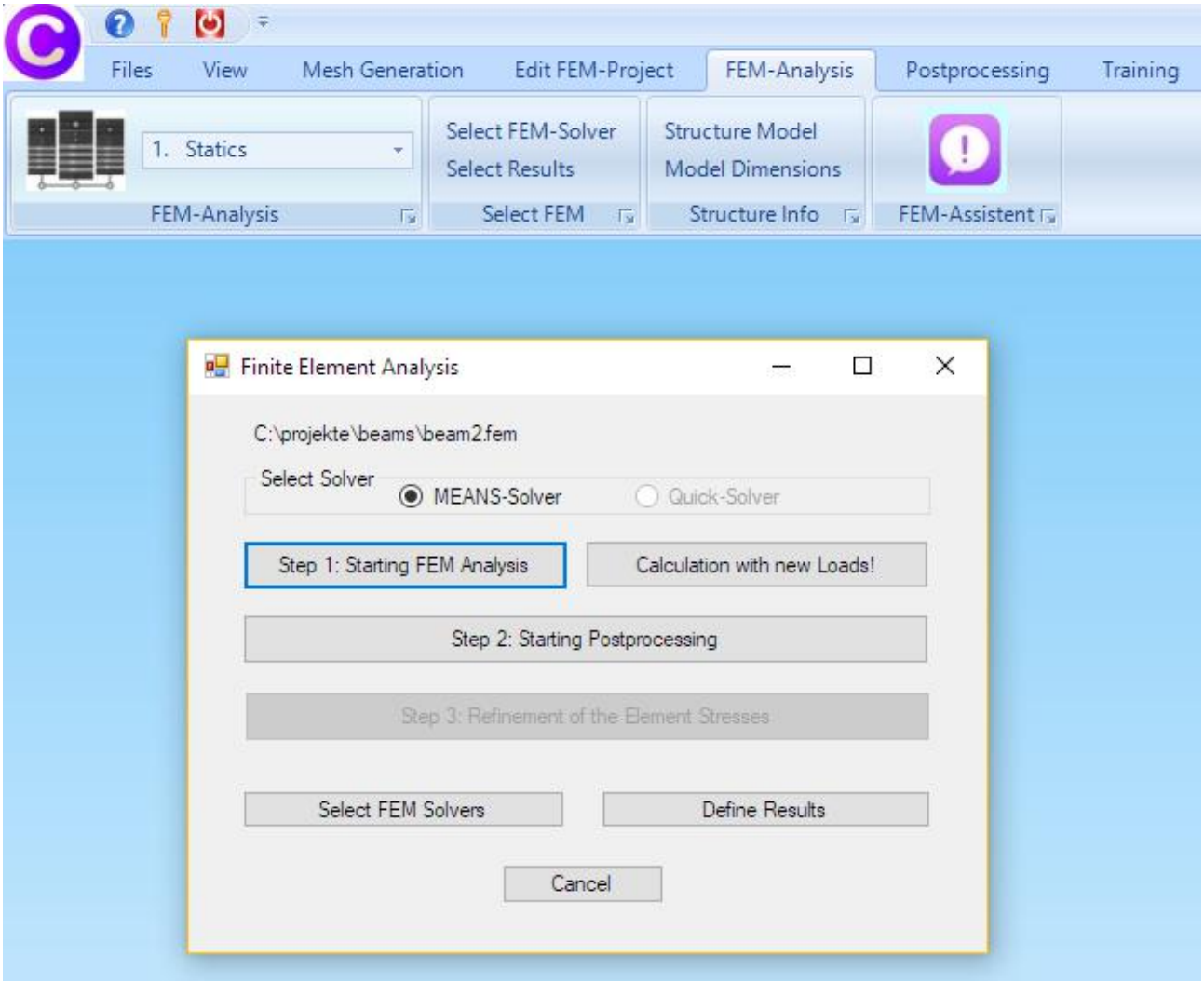
The cantilever beam is loaded at the right end with a single load of -10 000 N in Y direction. Create the Point Load in the following steps:

- Select the "Edit FEM Project" tab and menu "1. Point Load" to enter the node load.
- Enter Loadcase = 1, Number of loads = 0 and Value of Load = -10000
- Select the load direction "Y direction" and "Create Point Load" and click on node 2
- Node 2 is displayed in the Selectbox, there you choose "Create" to generate the load
- Select the "View" tab and menu "Node-Modus" and switch on the node numbering and load values.



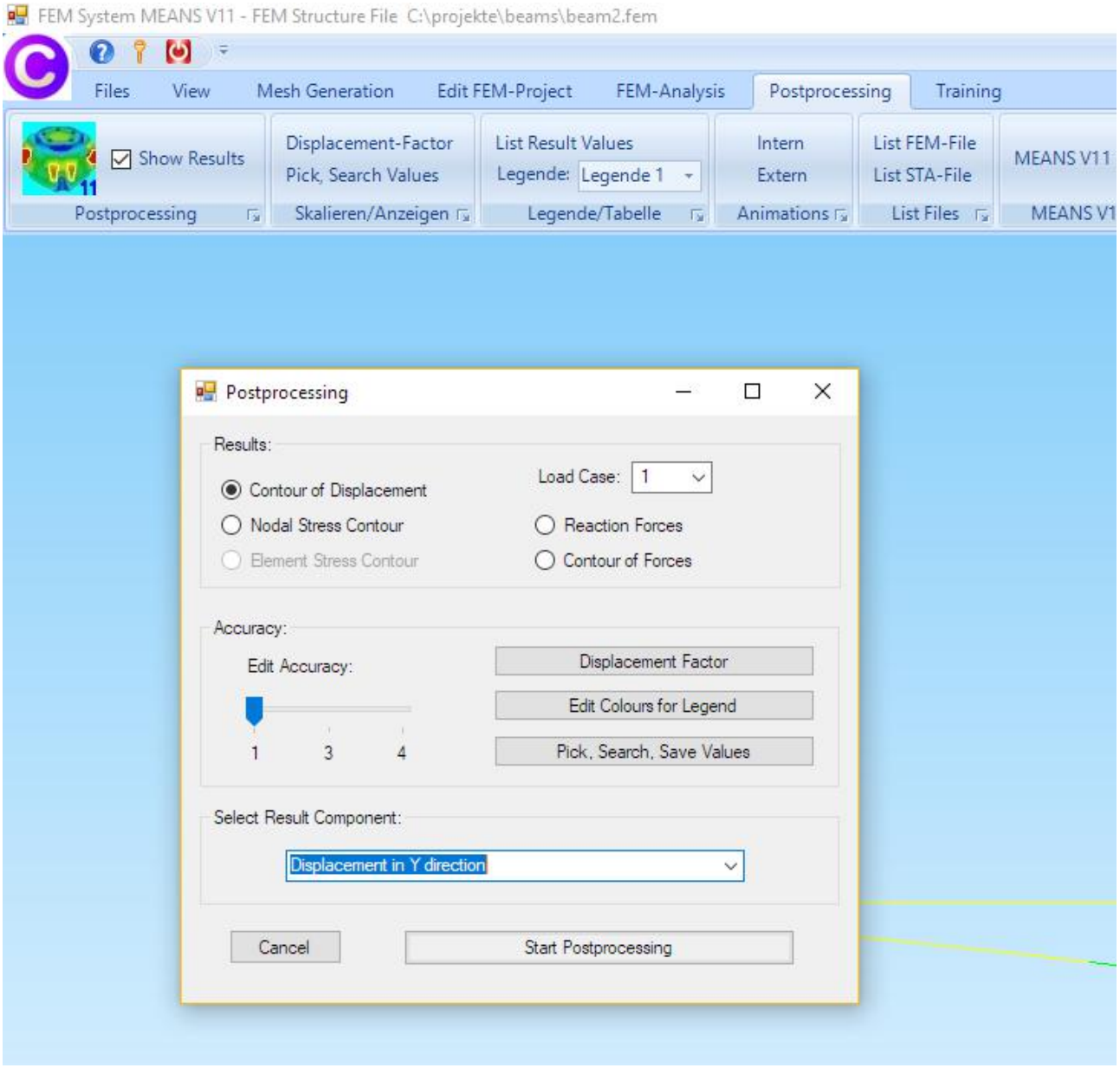
FEM Analysis

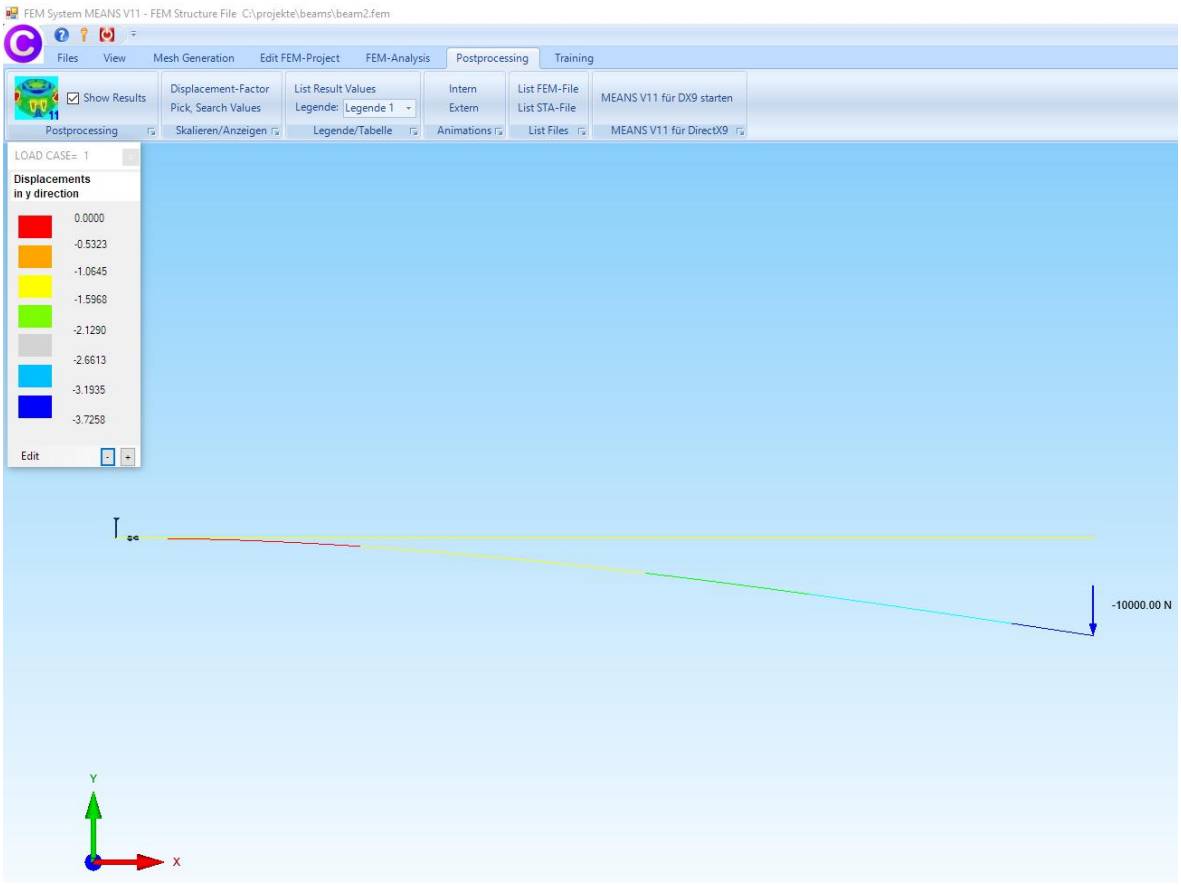
Select the "FEM-Analysis" tab and "1. Statics" to start the FEM solver and to calculate the displacements and stresses.



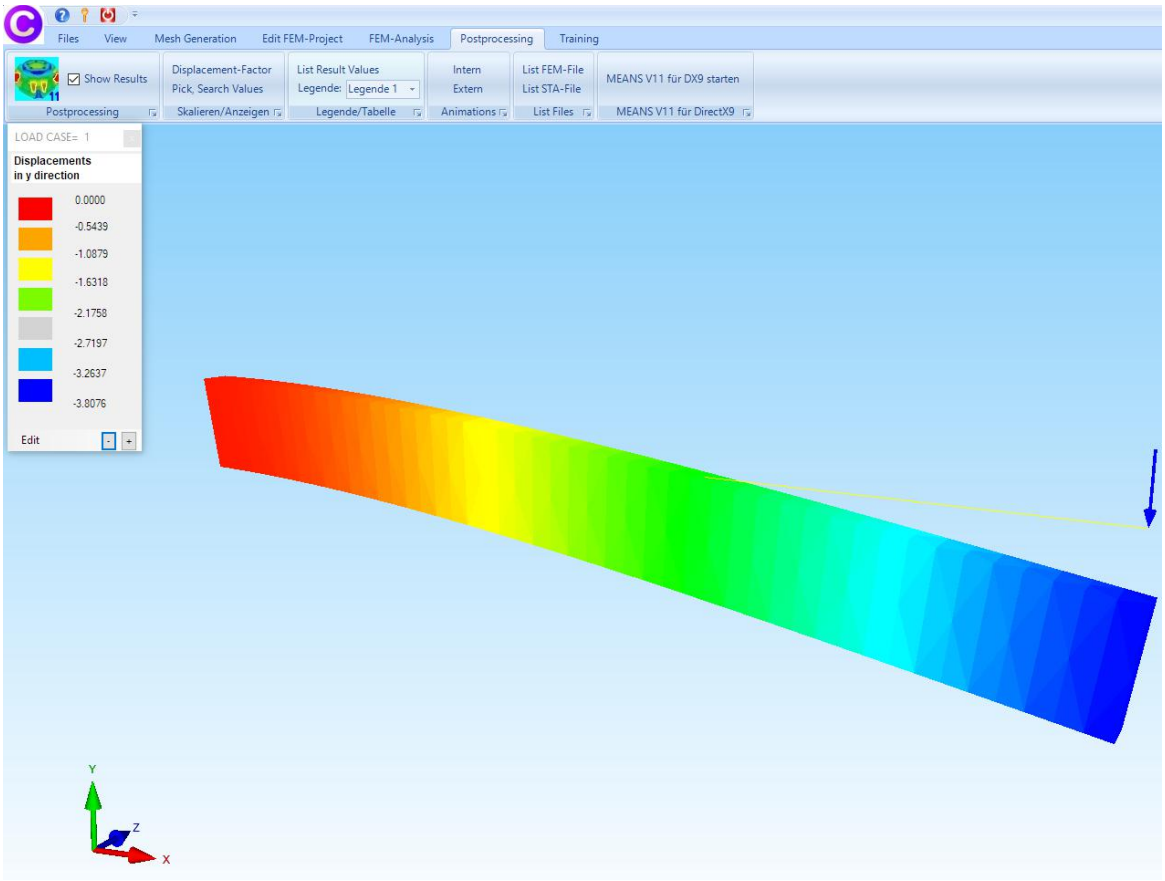
Postprocessing

Select the "Postprocessing" tab and click on the icon  to show the Contours of Displacments, Stresses and Forces.

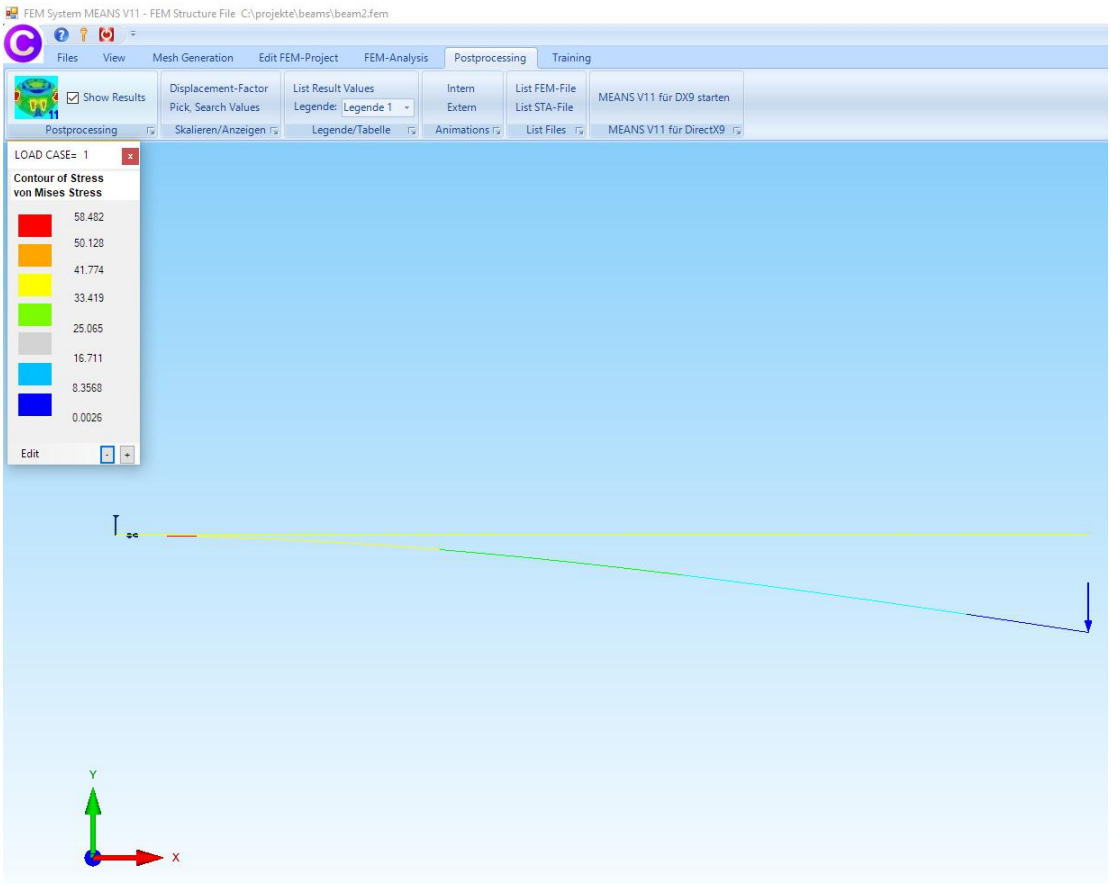




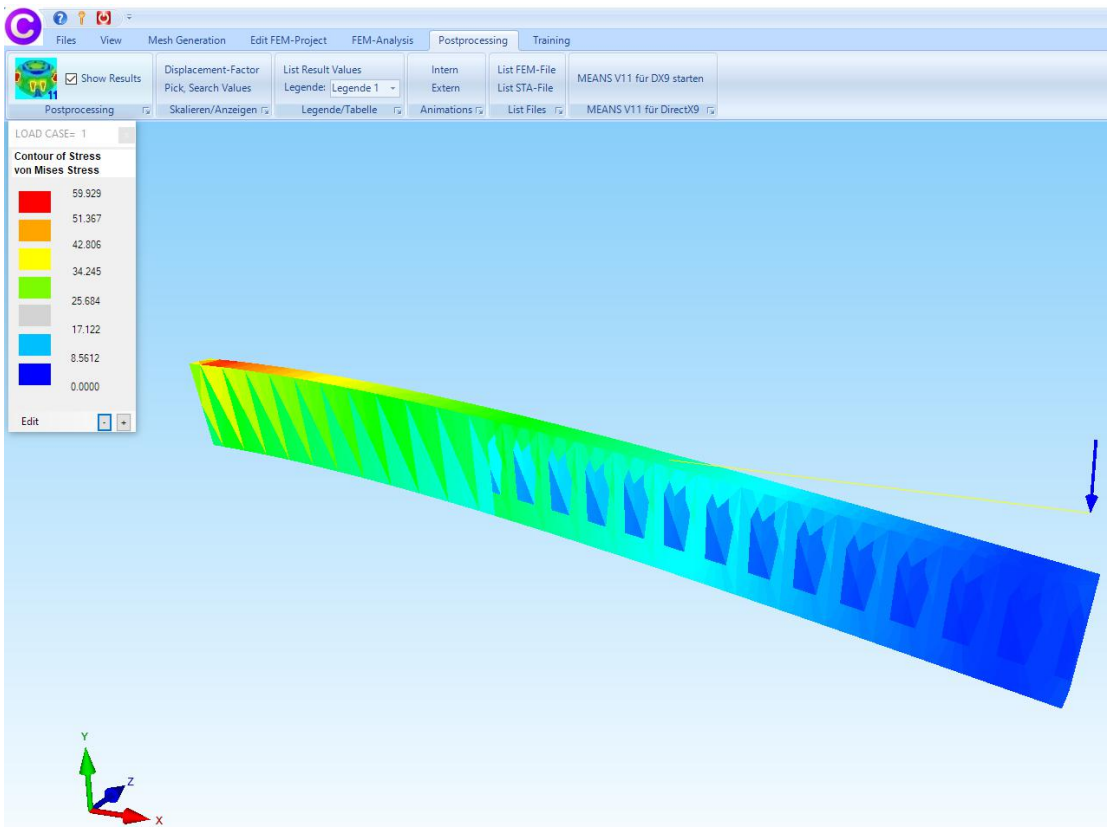
Max. Displacements in Y Direction with BEAM2 Elements = - 0.3725 mm



Max. Displacements in Y Direction with B32 Elements = - 3.8076 mm



Max. v.Mises-Stress with BEAM2 Elements = 58.42 N/mm²



Max. v.Mises-Stress with B32 Elements = 58.42 N/mm²

Example 2: IPE beam with trapezoidal line load

This is followed by a beam calculation with a steel beam IPE-240 with a length of 2000 mm that is loaded with a trapezoidal line load.

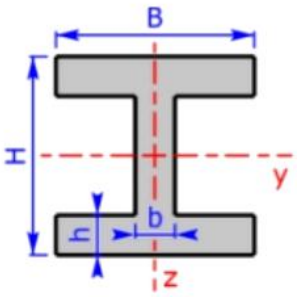
Exact results according to the beam theory

With the BEAM-Calculator https://calcresource.com/statics-cantilever-beam.html

the exact results are calculated according to the beam theory:

Material Datas:

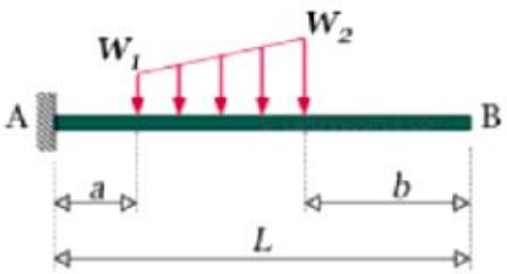
A	I/H-Profil				
H	240	mm	h	9.8	mm
B	120	mm	b	6.2	mm
Wst.	Stahl		E **	210000	N/mm²
I_y *	3670.9673	cm^4	W_y *	305.914	cm^3



Partial trapezoidal line load:

Partial trapezoidal load		
w_1 =	-30	kN/m
w_2 =	-50	kN/m
a =	500	mm
b =	200	mm

L = 2000 mm



Results:

Reactions:

R_A =	-52	kN
M_A =	62.6167	kNm

Bending Moment:

M_u =	62.6167	kNm

Deflection:

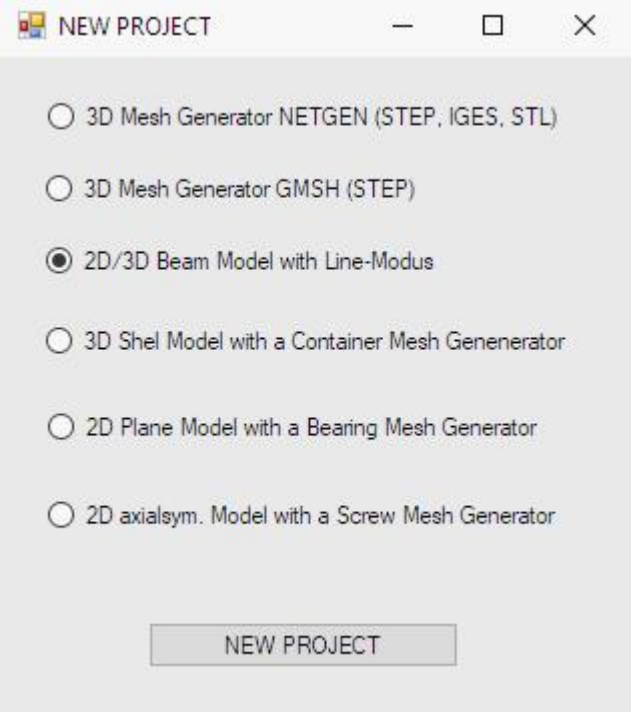
d_u =	-8.19797	mm

Bending Stress

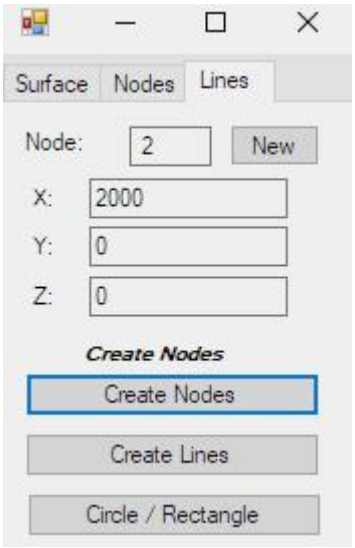
σ_B = M_u / W_y = ***204.68 N/mm²***

Create BEAM model in Line Mode

To create a beam model with 21 nodes and 20 BEAM elements, select the "File" tab and "New" as well as "2D/3D Beam Model with Line-Modus“ to display the side menu of the line mode.



Input of the nodes



Input node 1

Click on "New" and enter X = 0, Y = 0, Z = 0

Click on "Create Nodes" to create node 1.

Input node 2

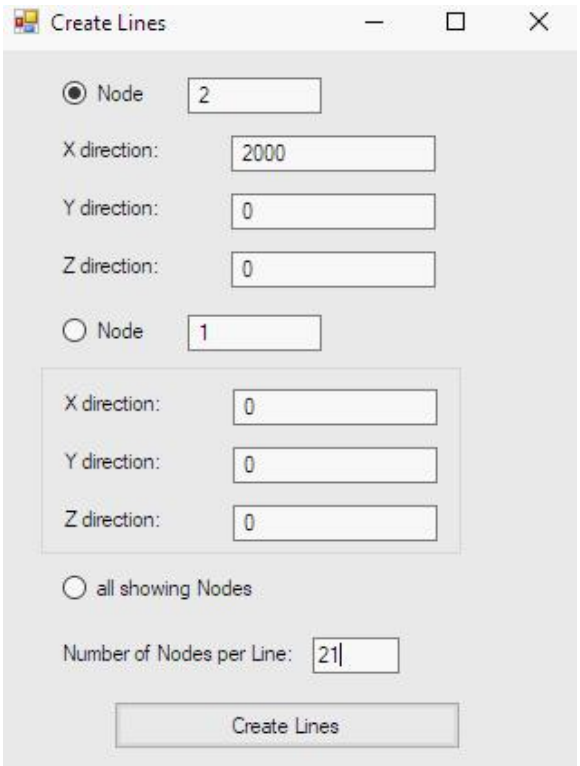
Click on "New" and enter X = 2000, Y = 0, Z = 0

Click on "Create Nodes" to create node 2.

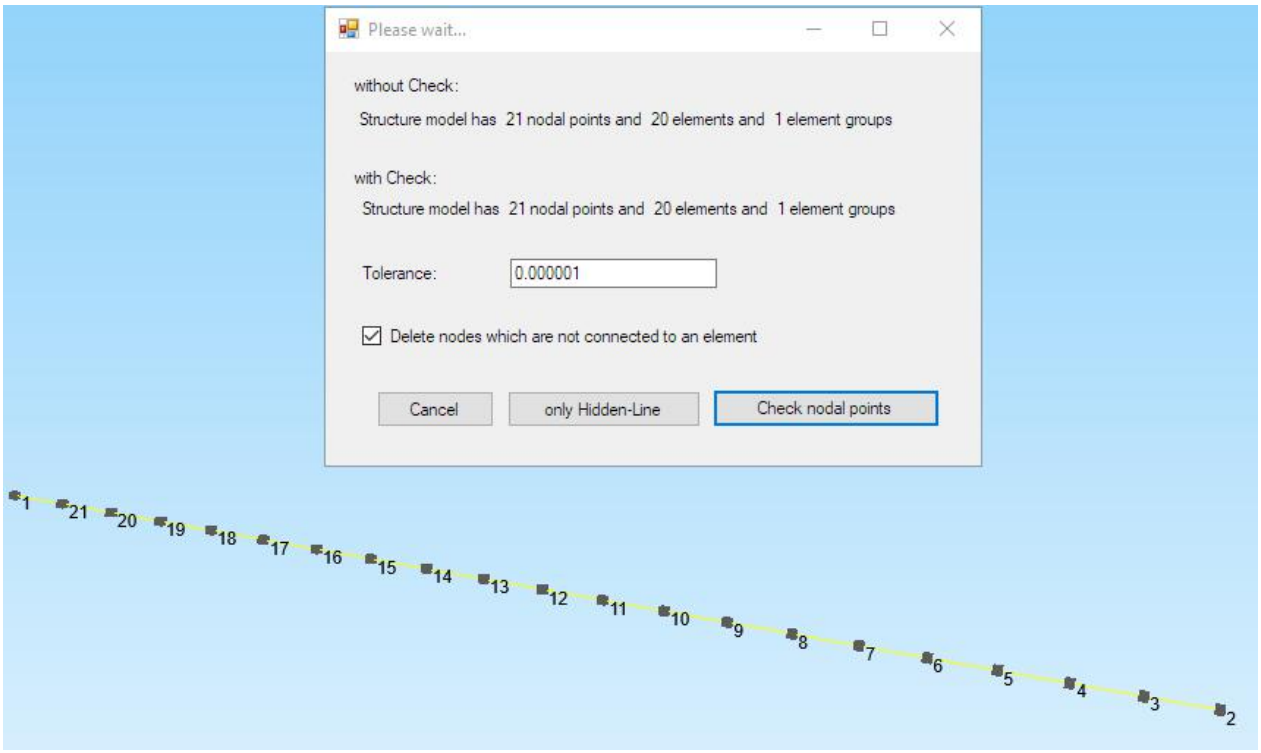
Now both nodes should be numbered on the screen.

Create beam elements

With the menu “Create lines” nodes 1 and 2 are connected with “Number of nodes = 21”, thus creating 20 BEAM2 bars, each with a node distance of 100 mm.

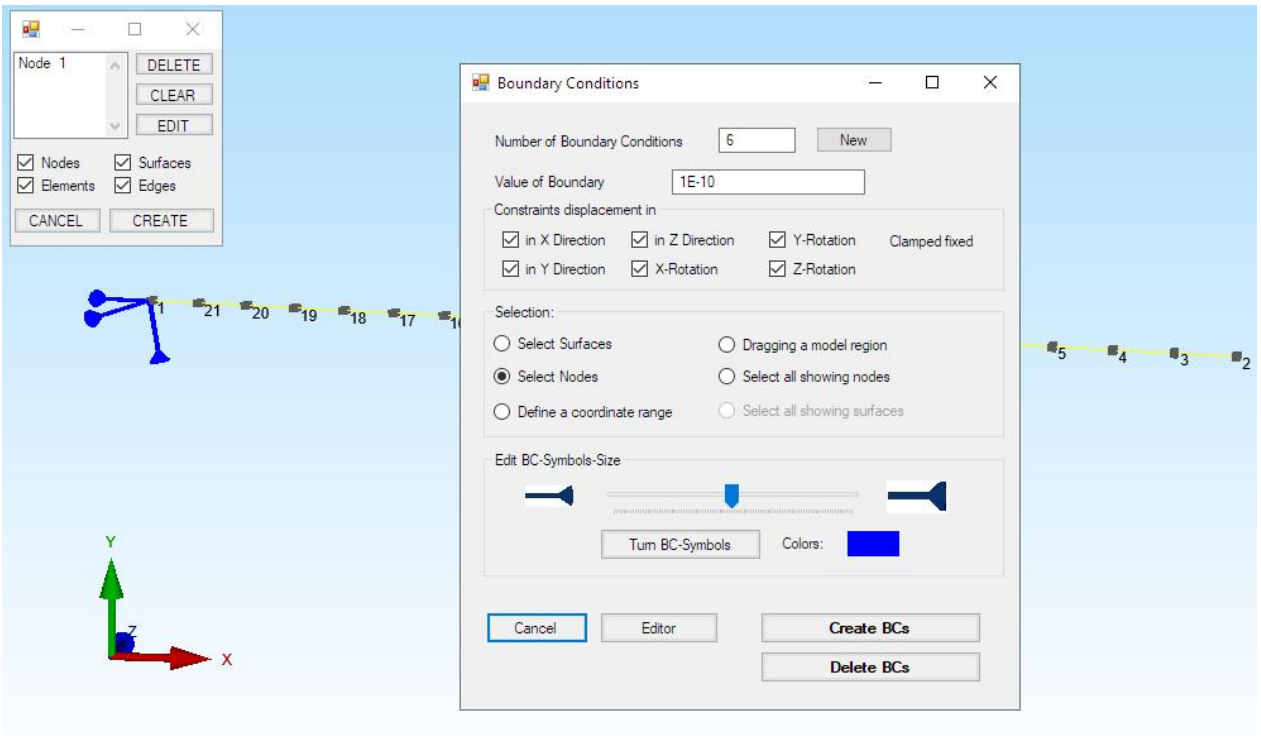


Check with menu "Check Nodes" whether there are still overlapping nodes.



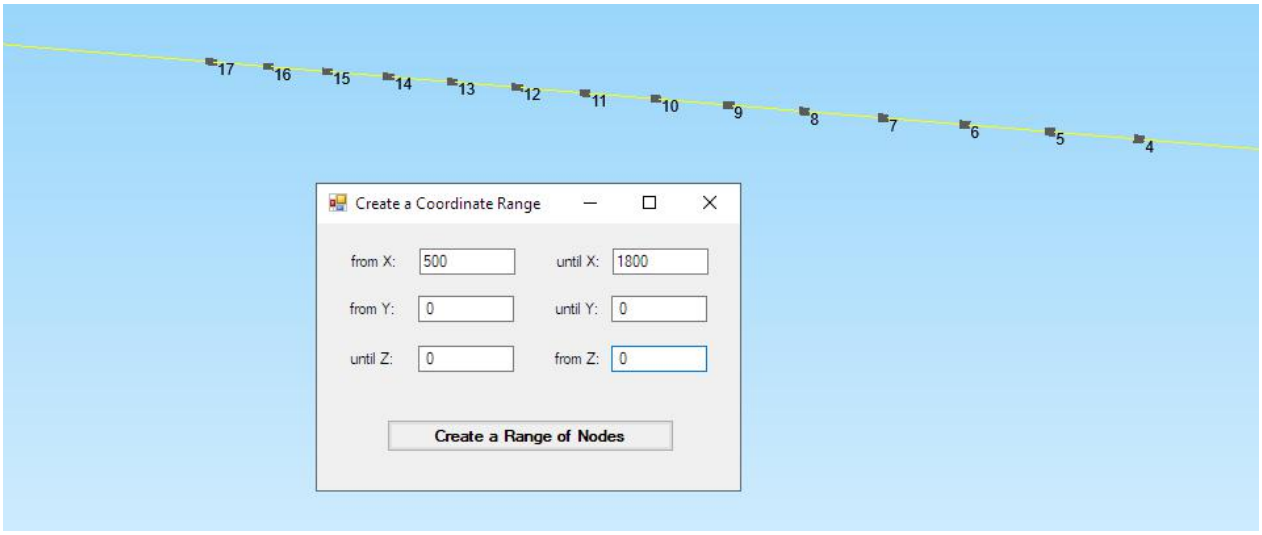
Create Boundary Conditions

Select the register “Edit FEM Project” and “Boundary Conditions” as well as the degree of freedom “Restraint” and the selection “Click individual nodes” and click on node 1 after the “Create BCs” menu to clamp this in the select box with the “Create” menu .



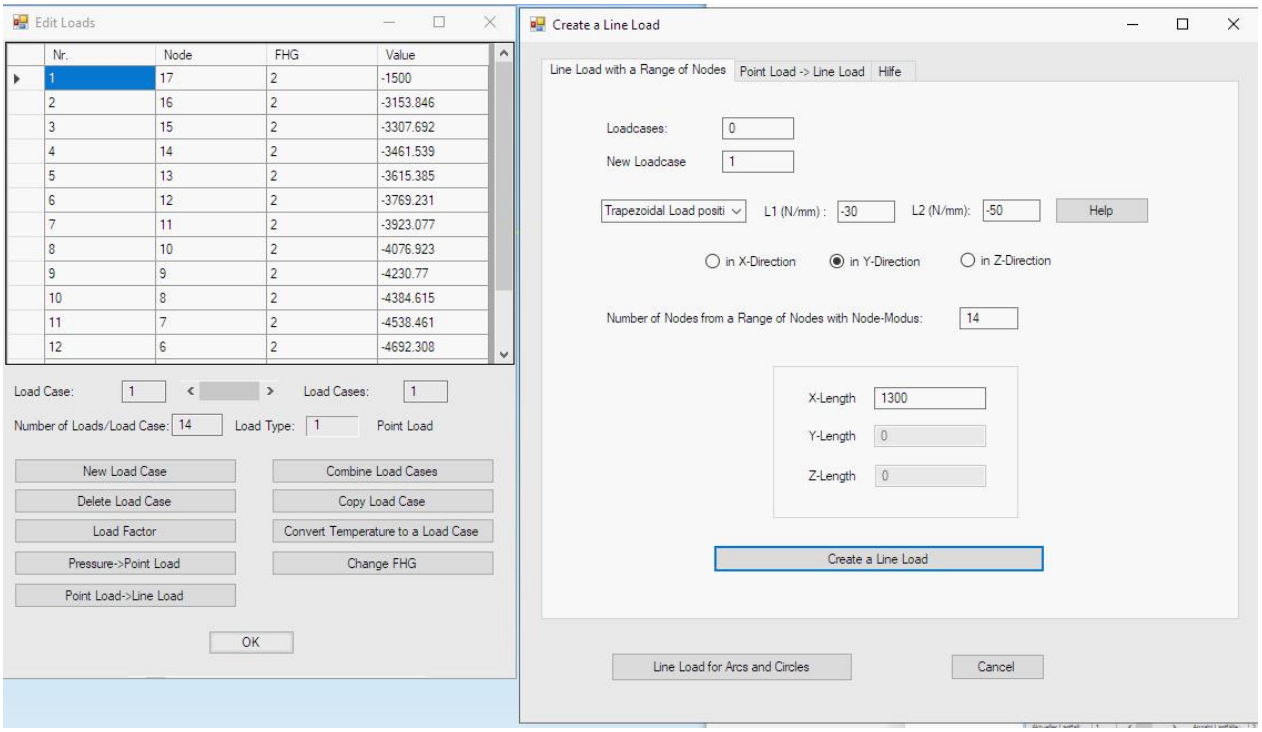
Create a Range of Nodes

In order to generate a trapezoidal line load, a Node of Range with a defined coordinate range of X = 500 mm to X = 1800 mm must first be generated in Node-Modus.

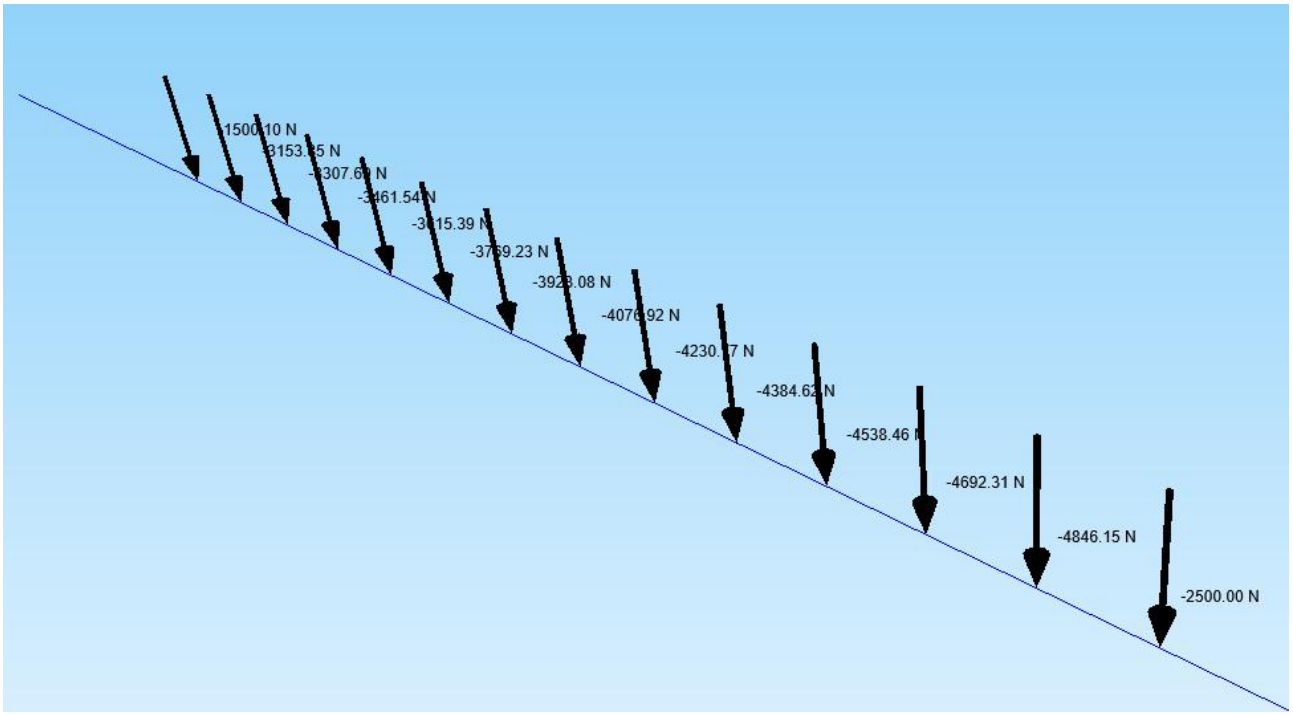


Create a trapezoidal line load

Select the "Edit FEM Project" tab and the "Line load" menu and select the drop-down menu "Trapezoidal load positive" in the dialog box to apply a nodal load in the Y direction with load case 1 and the load values L1 = -30 N / mm and L2 = -50 N / mm at the indicated node area.



A Point Load with 14 load nodes in the Y direction is now generated. In the Node Modus, select “Show load values” to display the load values in N.

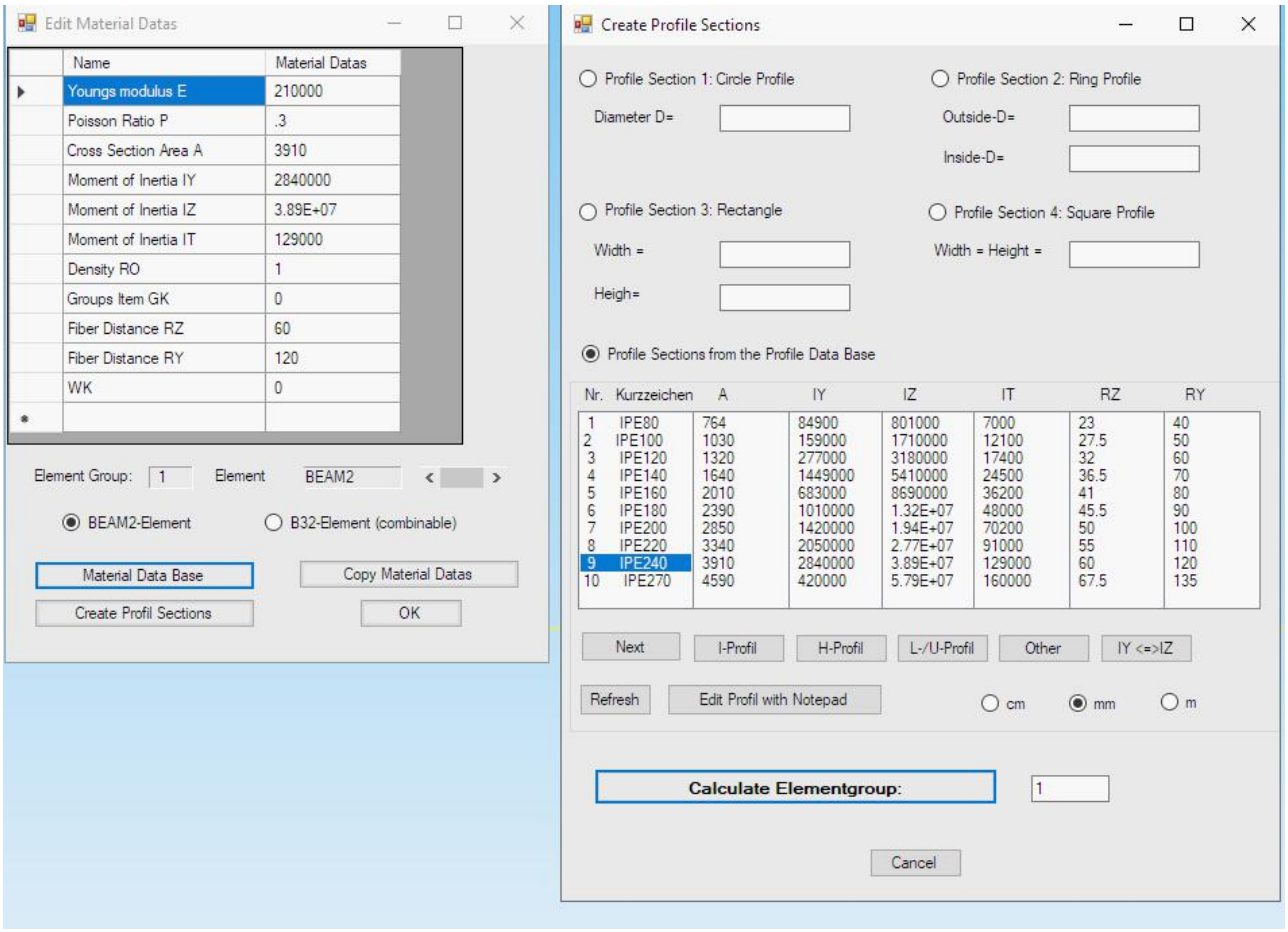


Circular line loads

For circular or round line loads, select the "Line Load for Arcs and Circles" menu, but here the node distance is always determined by the length of the element edges.

Material Datas

Select the register “Edit FEM Project”, “Material data” and “Generate bar profiles” and use the menu “Create bar profile for element group 1” to transfer the profile data from “IPE240” from the profile database with Notebad ”can be changed or expanded with any text editor.



Profile data in the XY plane

If, on the other hand, the vertical axis of the profile does not run in the Z but in the Y direction, the profile must also be reversed accordingly, select "IY <=> IZ" to rotate the profile data.

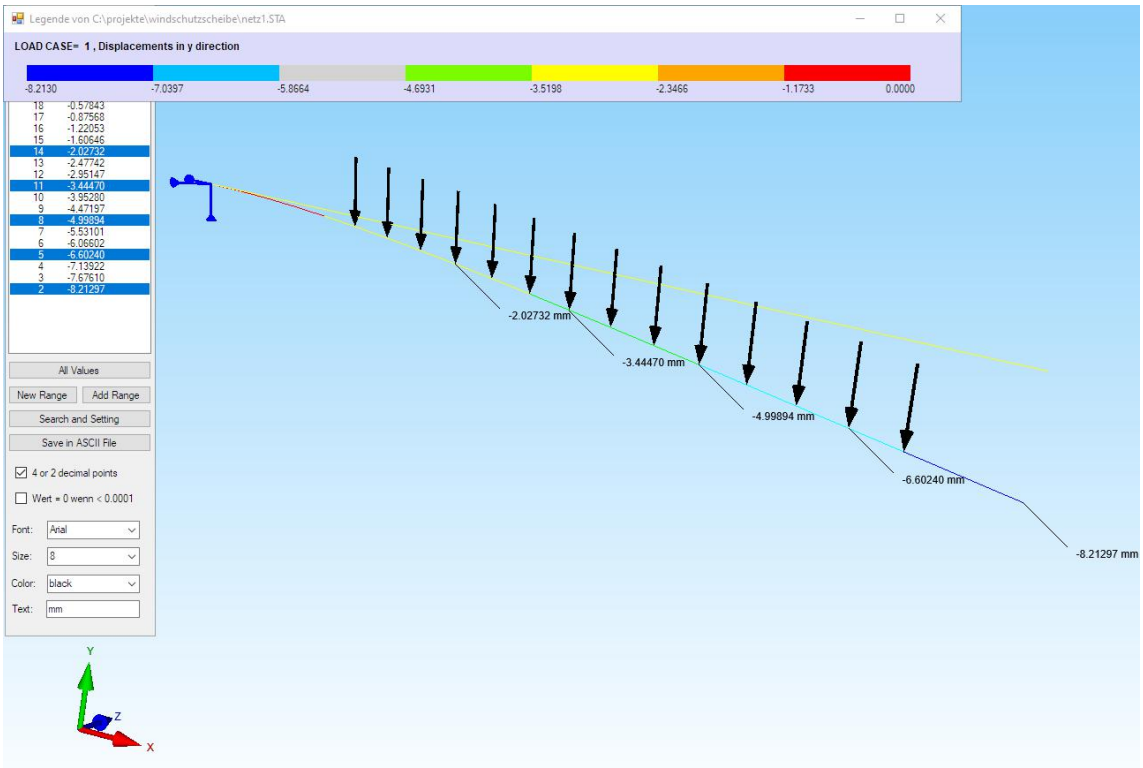
FEM analysis

Select the "FEM Analysis" tab and use the static solver to calculate the displacements, reaction forces and stresses. Before doing this, back up on the hard drive under any name.

Evaluate Displacements and Bending Stresses

Select the "Postprocessing" tab to evaluate the displacements and bending stresses.

Max. Displacements = -8.21 mm (exactly = 8.19 mm)



Max. v.Mises Stress = 204 N/mm² (exactly = 204 MPa)