Part 24: IPE Beams with a trapezoidal line and surface load

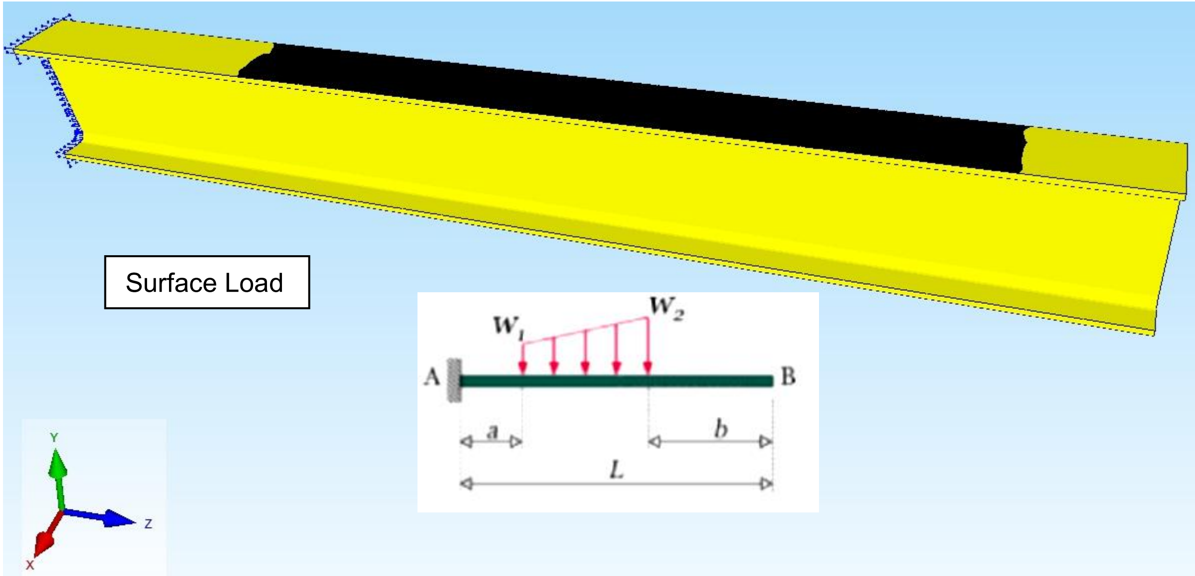
This new part of the handbook of FEM-System MEANS V12 from the website www.fem-infos.com shows how a

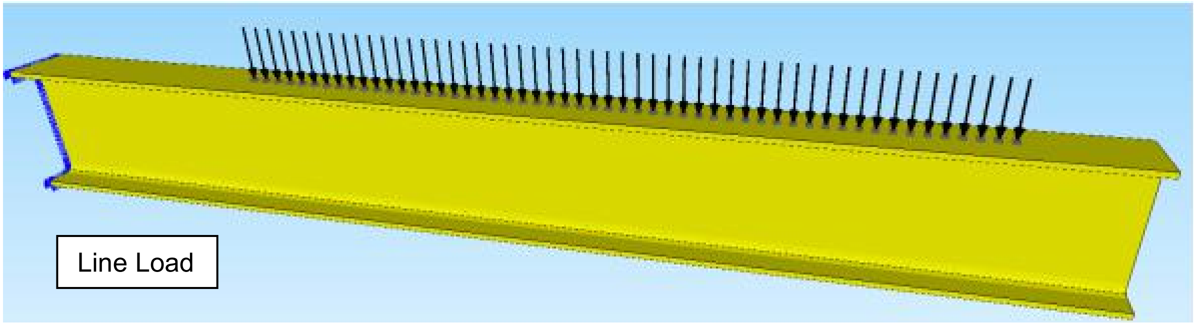
beam structure (see Part 06) with a

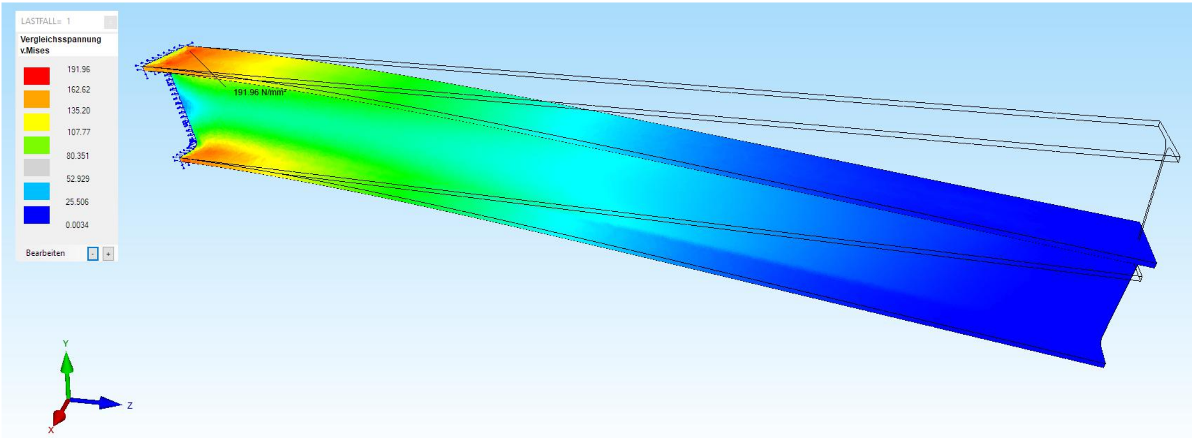
- uniform, triangular and trapezoidal line load

and having a tetrahedral, hexahedral and pentahedral structure with

- uniform, triangular and trapezoidal line load
- uniform, triangular and trapezoidal surface load can calculated.





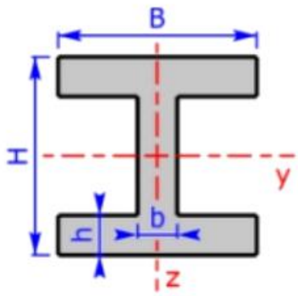


Exact results according to the beam theory

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

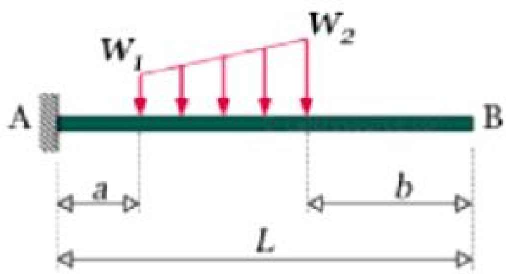


Partial trapezoidal line load:

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:

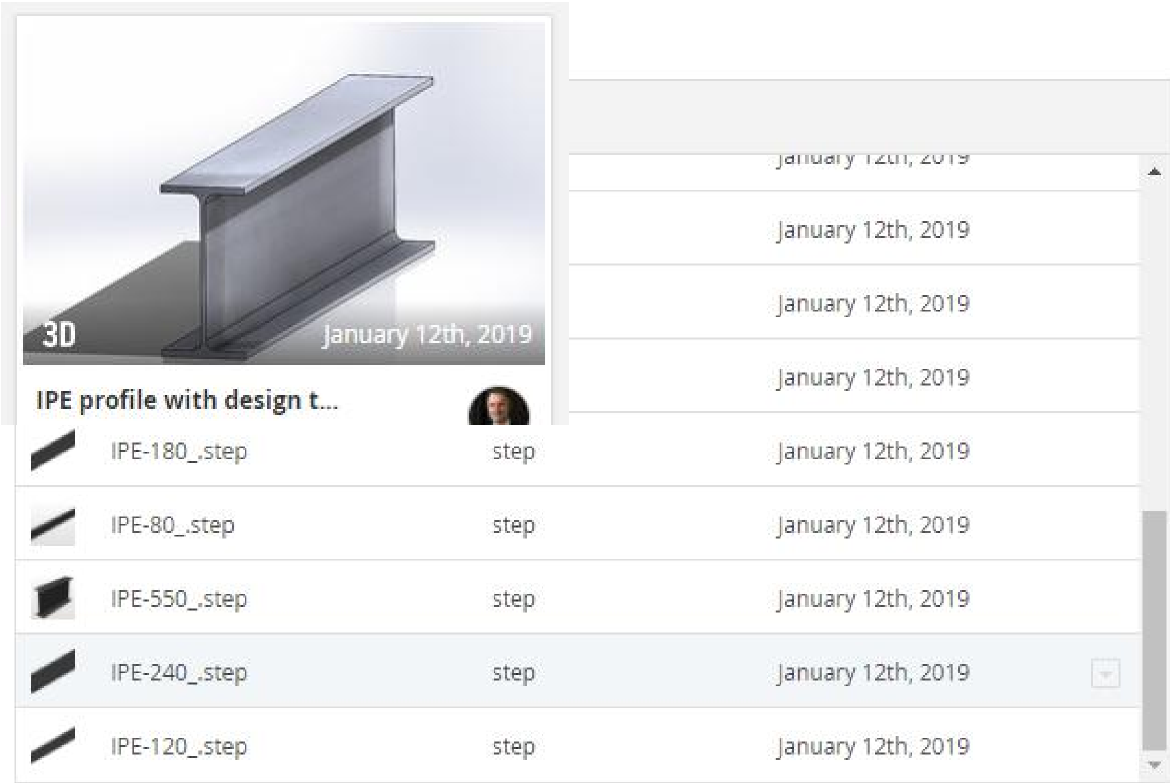
d_u = -8.19797 mm

Bending Stress

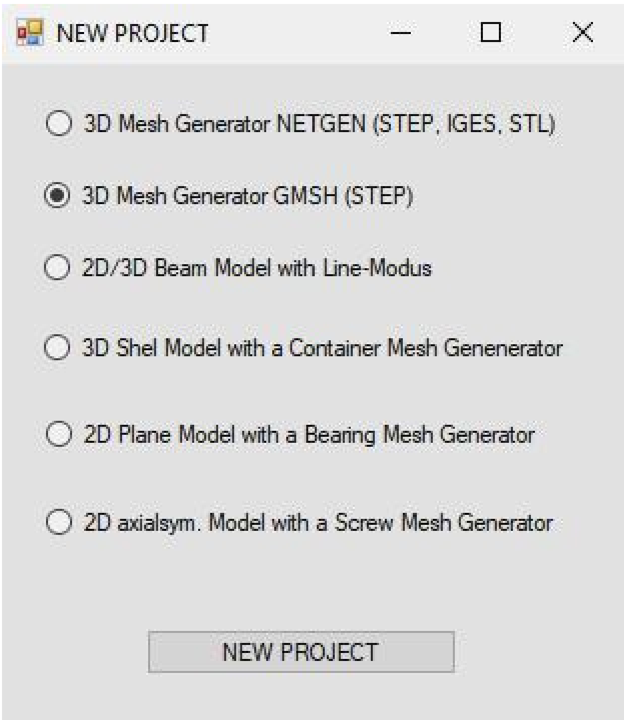
$\sigma_B = M_u / W_y$ = **204.68 N/mm²**

Tetrahedron model with a trapezoidal surface load

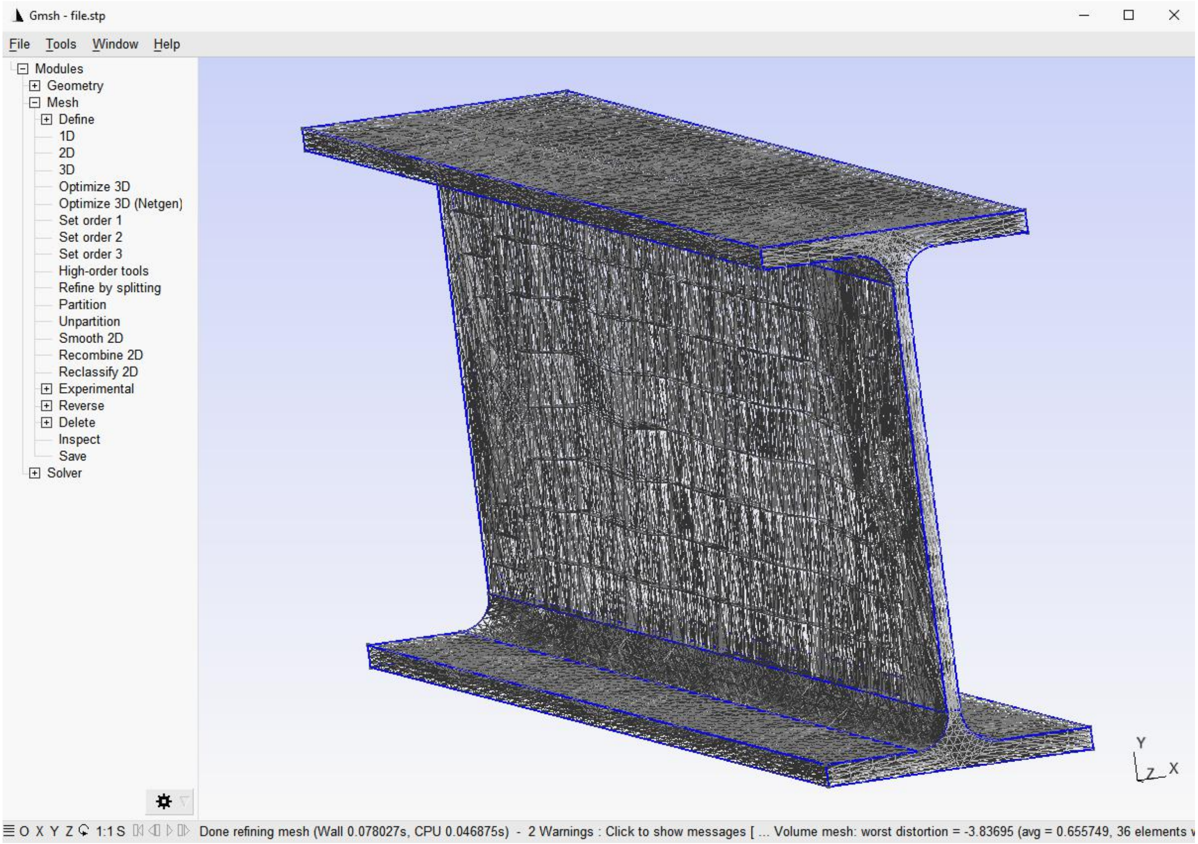
A suitable IPE-240 carrier for the FEM calculation can be downloaded in STEP format from the free 3D library www.grabcad.com:



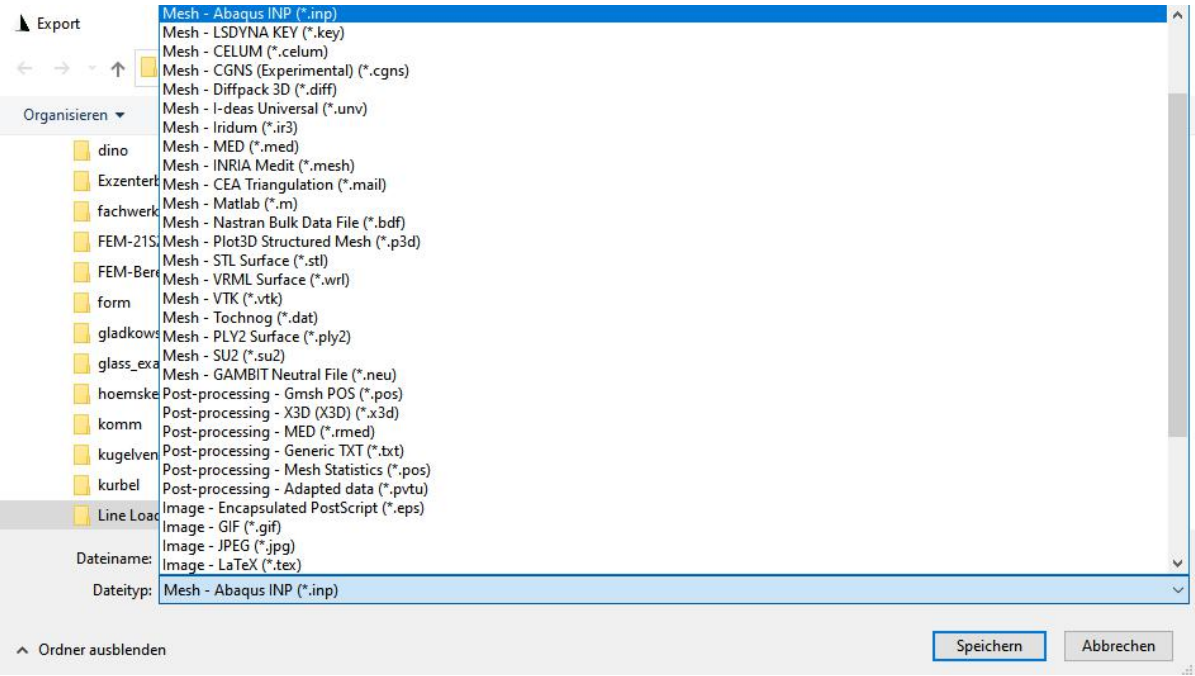
To do this, select “New” and load the “IPE-240.step” STEP file to create a tetrahedral mesh with the GMSH 3D Mesh Generator.



In GMSH, select the "3D" menu and the menu 2x "Refine by splitting" to generate a FEM Mesh with 46 272 TET4 elements and 9 975 nodes.

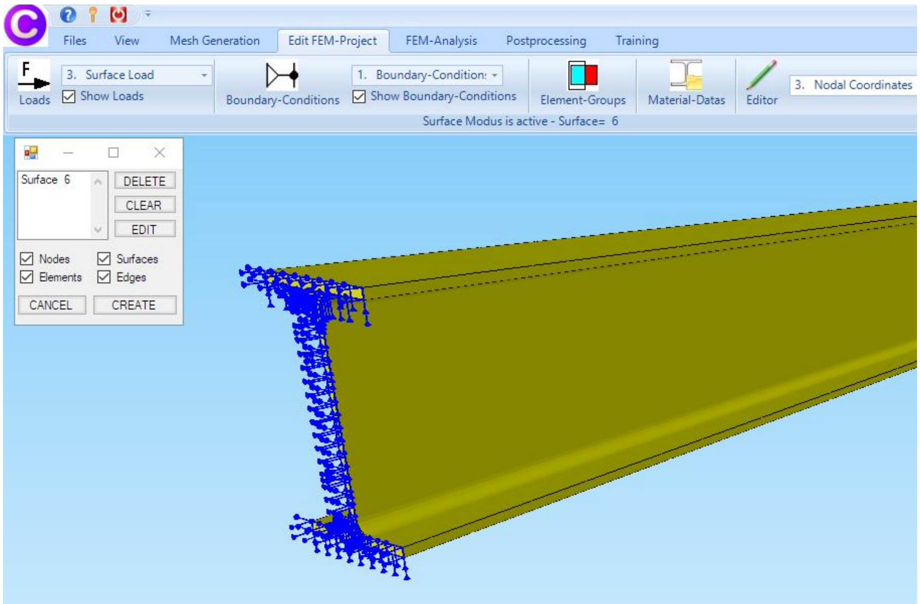


Select the menu "File" and "Export" and export the Mesh in Abaqus INP format back into the same directory of the STEP file so that it can be automatically imported and displayed in MEANS V12.



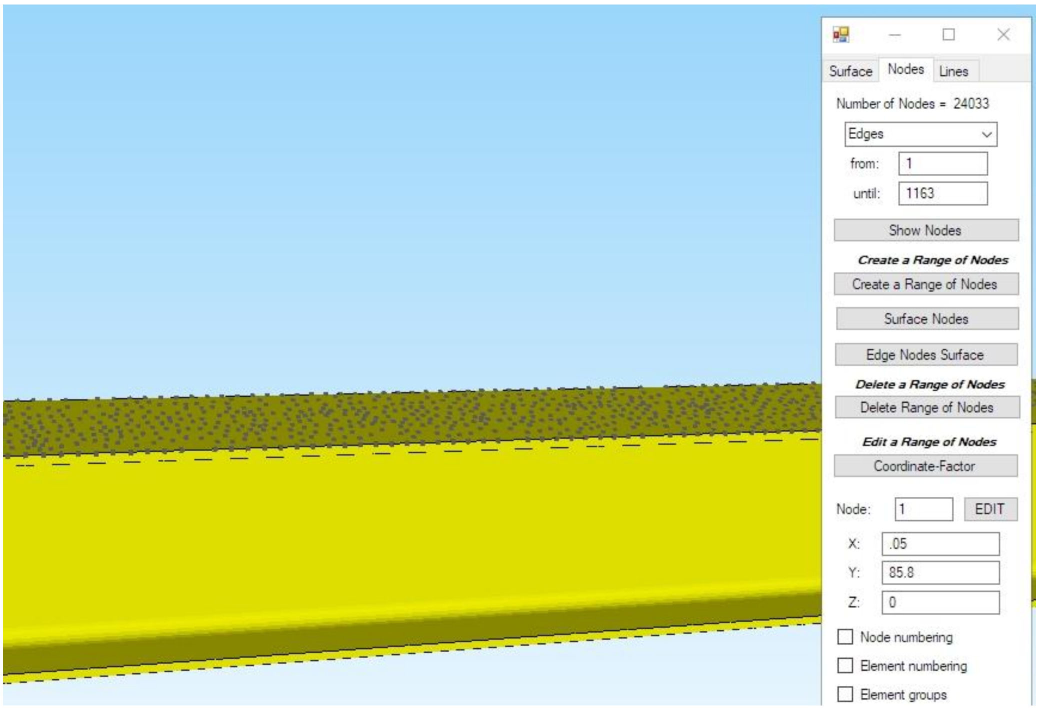
Boundary Conditions

The IPE is clamped fixed on the left side. Create it with the “Edit FEM project” tab and “Boundary-Conditions” by selecting the left Surface 6 and confirming with “Create” in the select box.

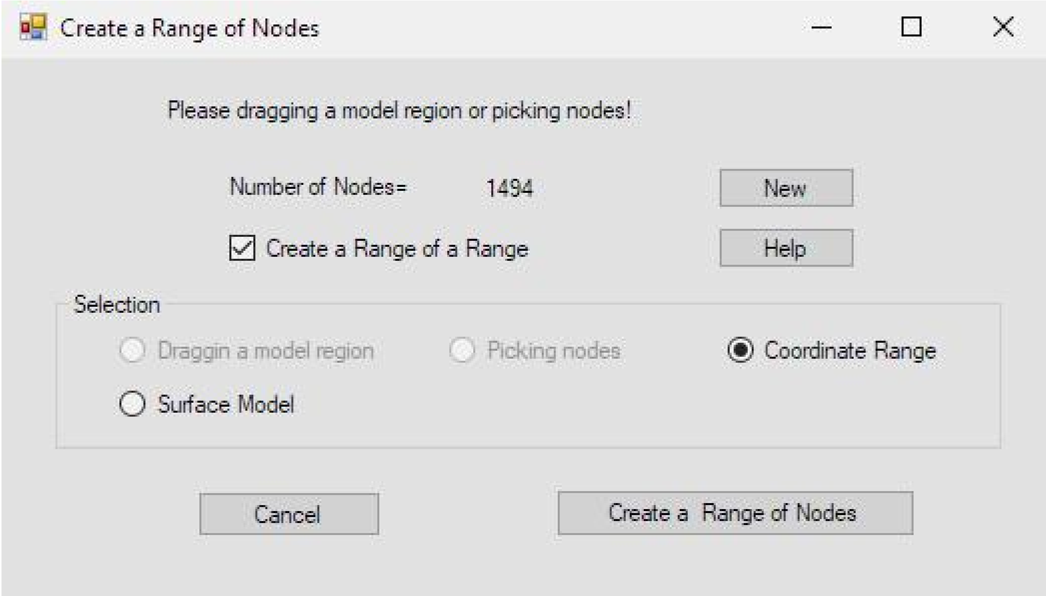


Create a Range of Nodes for Surface Load

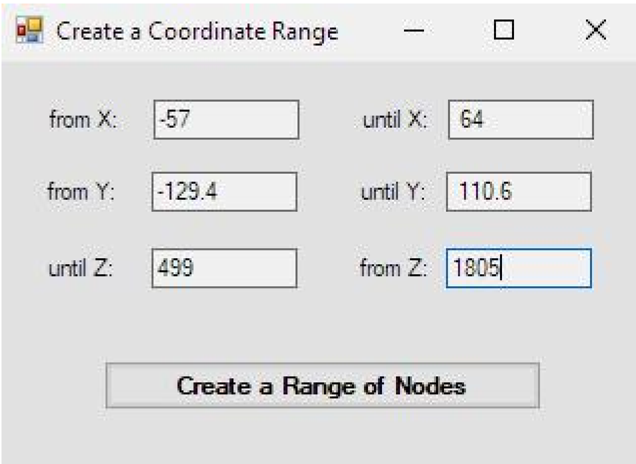
Select the “View” tab and the “Node-Modus” menu as well as the “Surface Nodes” side menu and click on the upper Surface 3 to display all nodes.



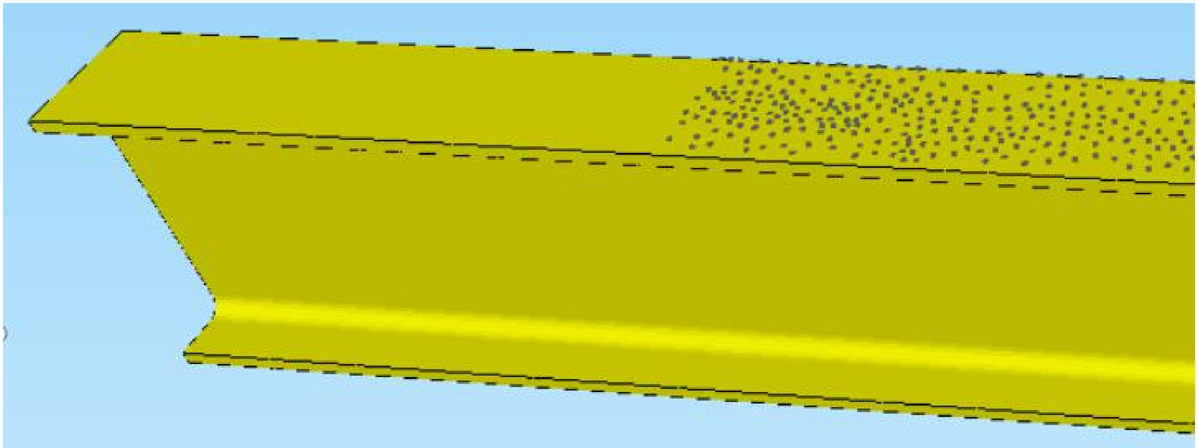
Then create with menu "Create a Range of Nodes" with "Coordinates Range" and additional with the option „Create a Range of Range“



new range of nodes from Z = 499 mm to Z = 1805 mm inclusive a small tolerance due to the unstructured nodes.



New Range of Nodes for the trapezoidal Surface Load

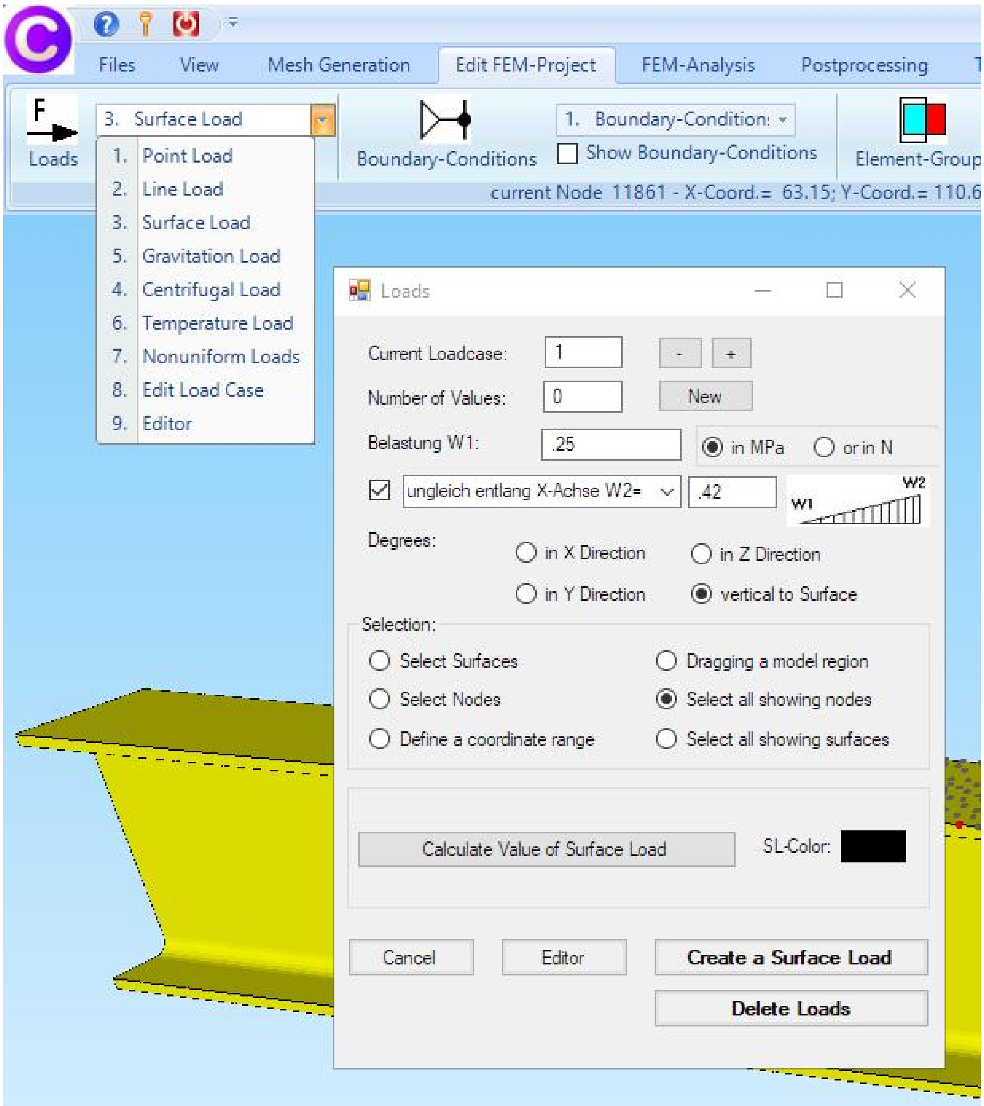


Now select the "Edit FEM Project" tab and "Surface Load" to create a trapezoidal Surface Load with the following values:

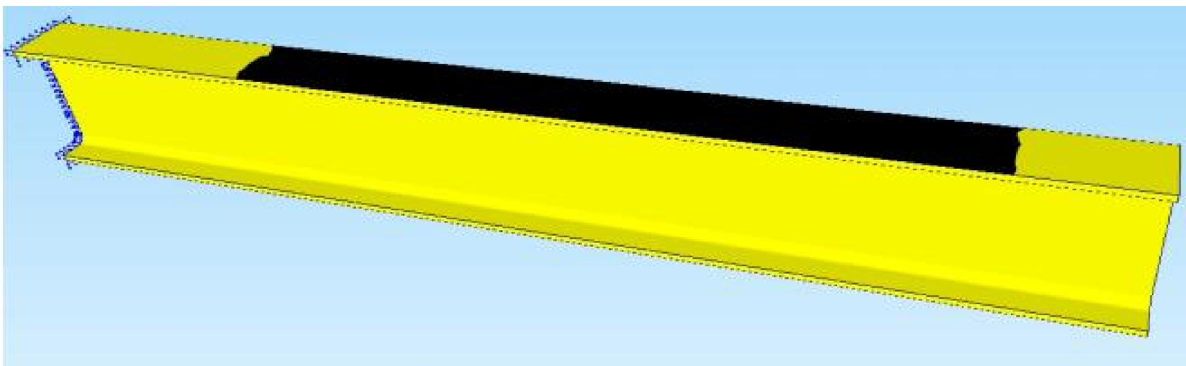
$W_{1\ \text{Surface Load}} = W_{1\ \text{Line Load}} * \ b = 30\ \text{N/mm}\ /\ 120\ \text{mm} = 0.25\ \text{N/mm}^2$

$W_{2\ \text{Surface Load}} = W_{2\ \text{Line Load}} * \ b = 50\ \text{N/mm}\ /\ 120\ \text{mm} = 0.42\ \text{N/mm}^2$

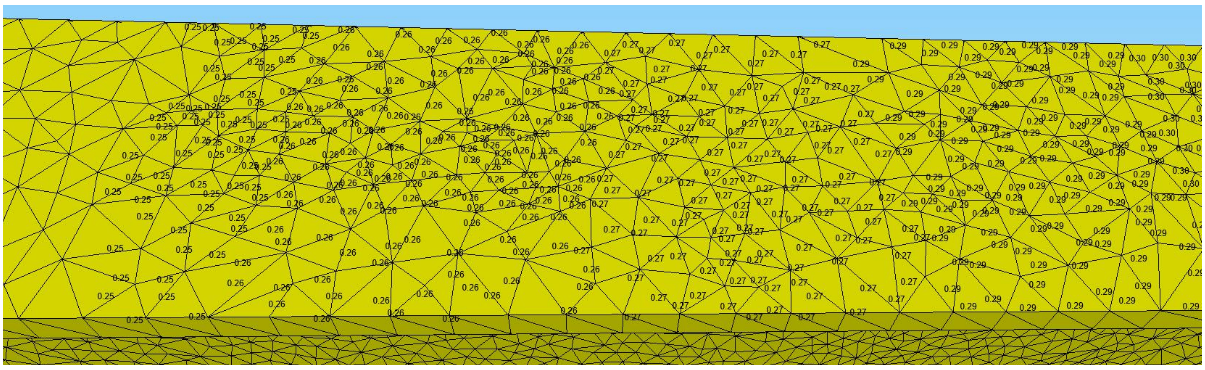
with "unequal along Z-axis" and with the selection "all showing nodes"



The result is the following trapezoidal surface loading



With "Show load values" in the Node Modus, the load values can be displayed and checked:

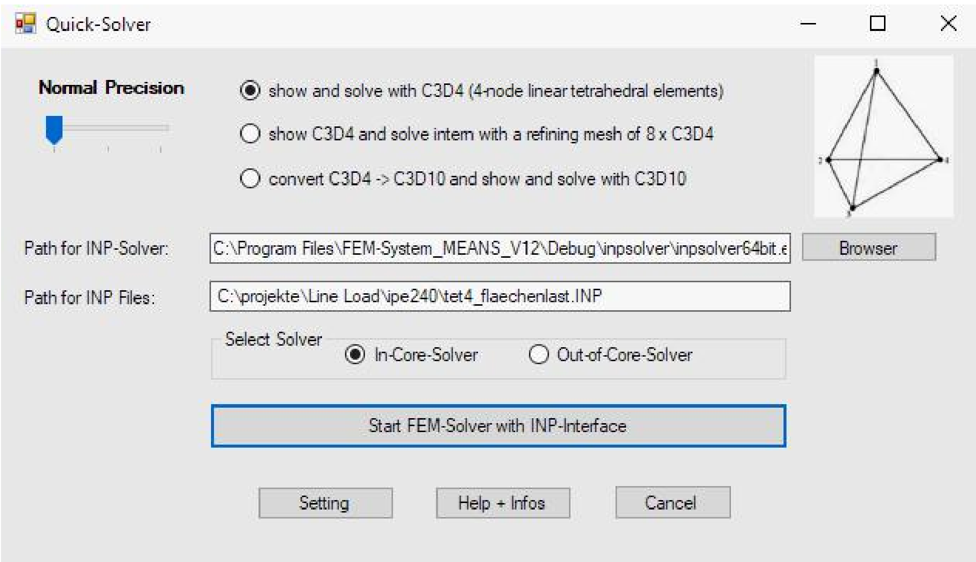


Material Datas

Since the material steel is always preset with a modulus of elasticity of 210,000 N/mm², no material data is required.

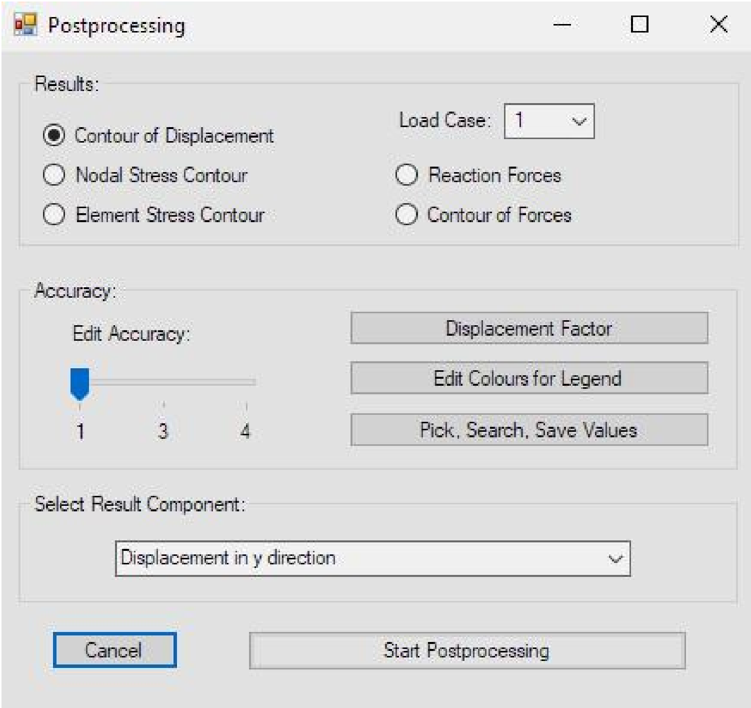
FEM Analysis

First save the FEM model under any name on the hard disk, select the “FEM Analysis” tab and calculate a static analysis with the Quick Solver.

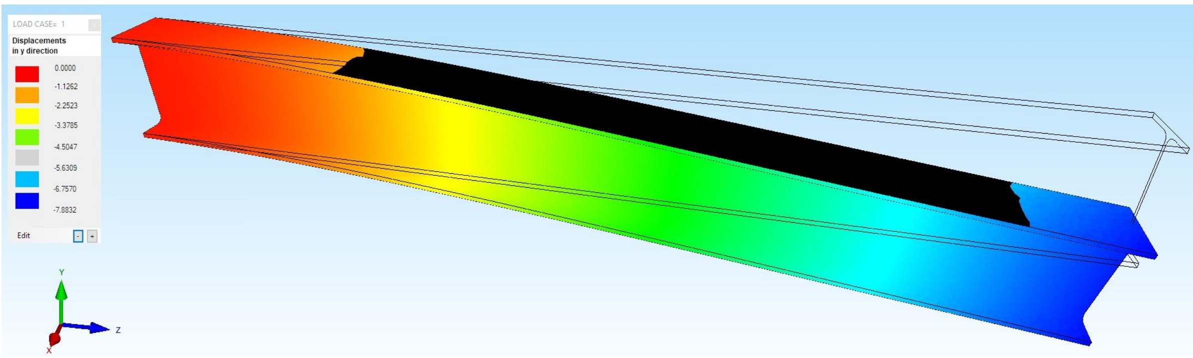


Postprocessing

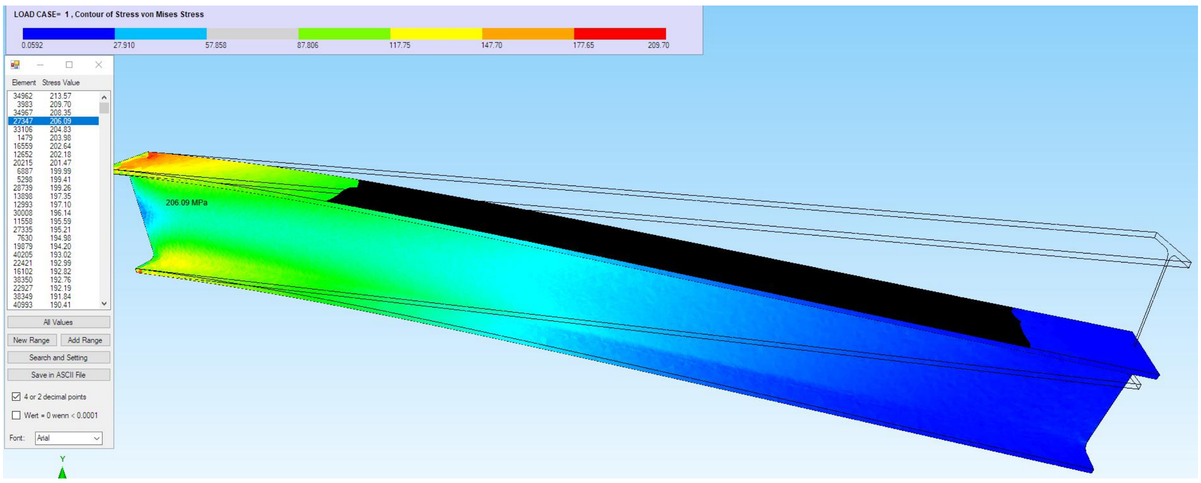
After the FEM Analysis, the results can be evaluated using the icon  with register „Postprocessing“.



Max. Displacement in y direction = -7.88 mm (exactly = -8.19 mm)



Max. v.Mises-Stresses = 206 N/mm² (exactly = 204 N/mm²)

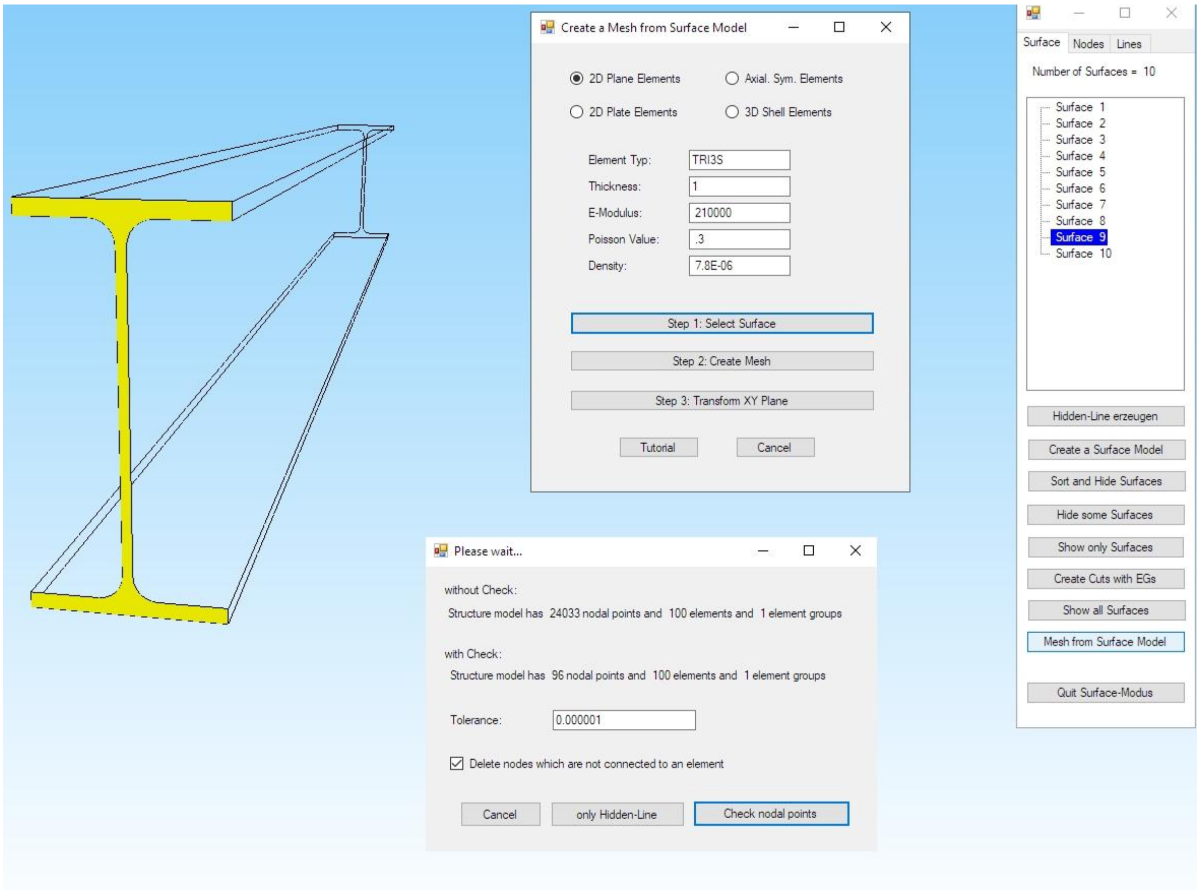


Pentahedral model with a trapezoidal line load

To get a pentahedron model, simply extruded the tetrahedron front surface in Z-direction.

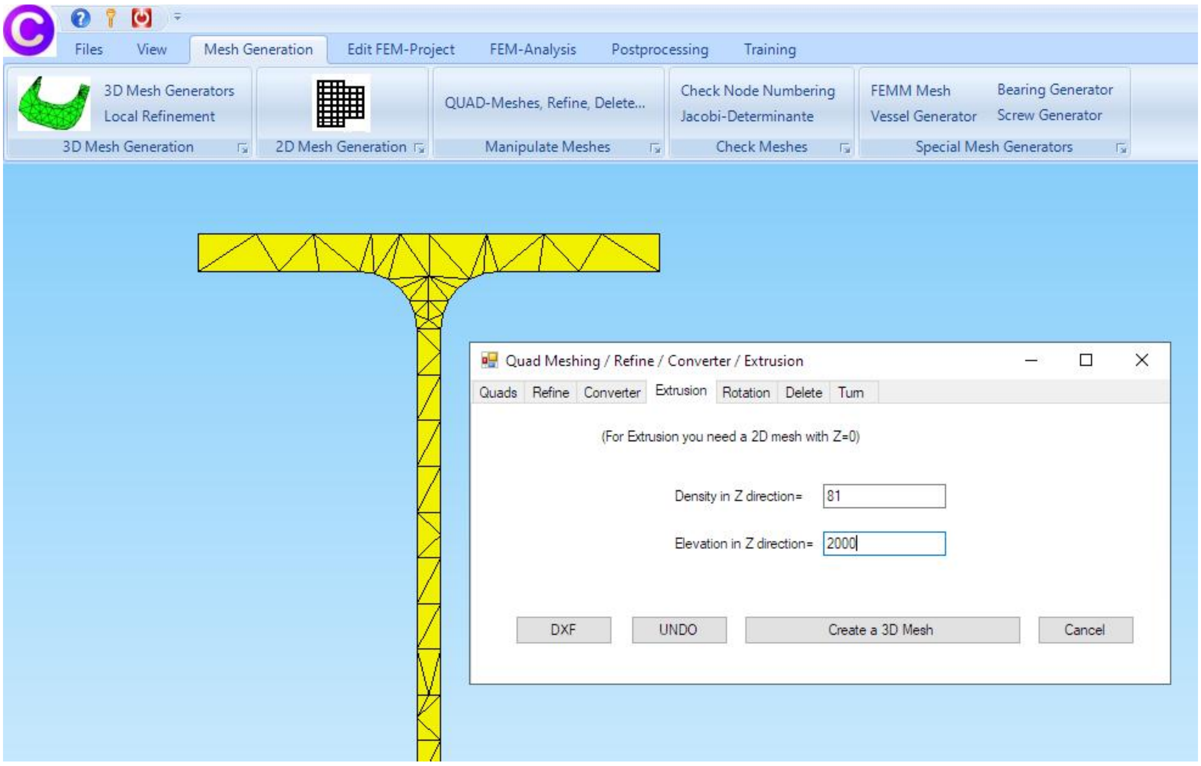
Create a 2D Mesh from the front surface

In Surface Modus, select menu “Mesh from Surface Model”. In the first step, select the profile surface 9. In the second step you create the triangular mesh with 96 nodes and 100 TRI3S elements with a node check and the option “Delete nodes which are not connected to an element”. With the third step the z-axis can be set from 2000 to zero and the x-axis can be swapped with the y-axis

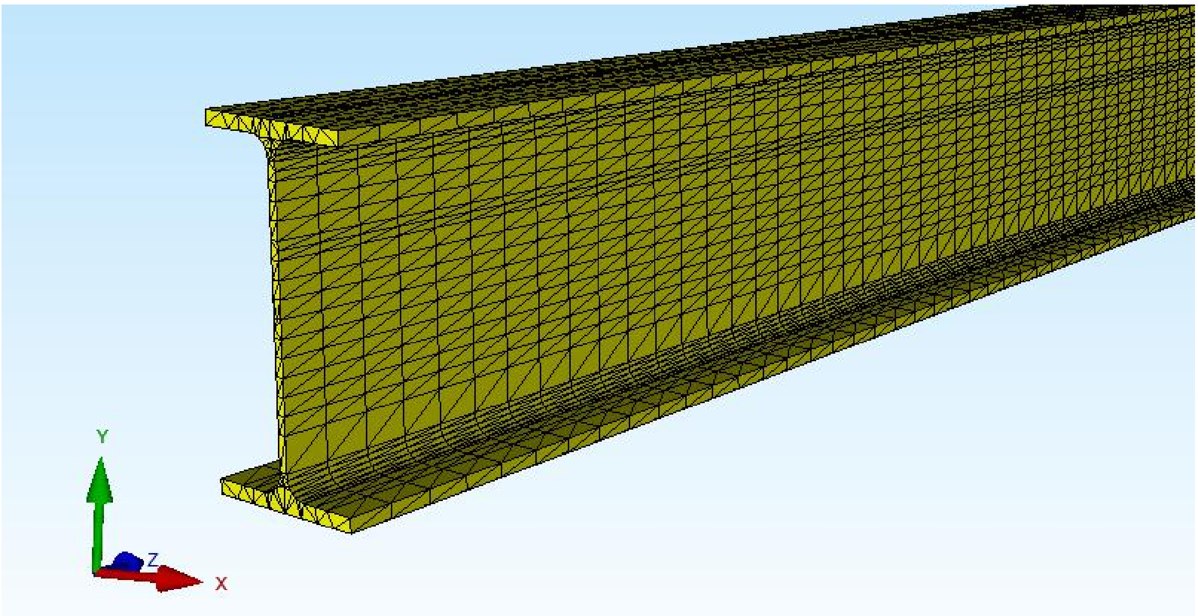


Extrusion

Select register "Mesh Generation" and menu "QUAD-Meshes, Refine, Delete...“ as well as in the next dialog box the register "Extrusion" and create the pentahedron mesh with a number of nodes in the Z direction = 81 and a Z object height = 2000 mm.

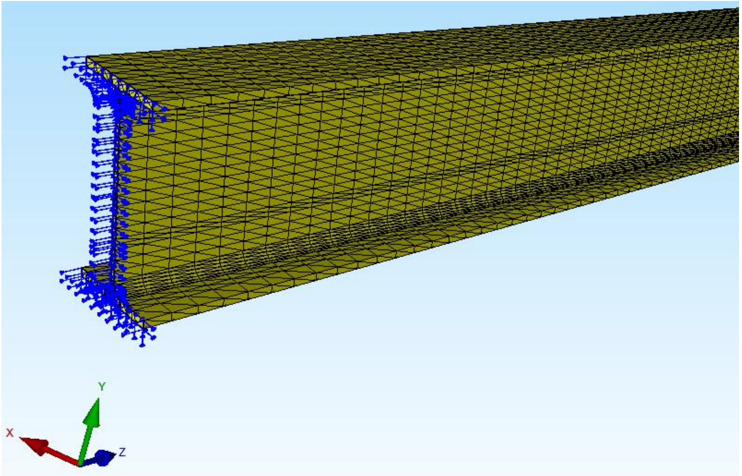


Select the menu "Create 3D Mesh" to create the pentahedron model with 8240 PEN6 elements and 8019 nodes..



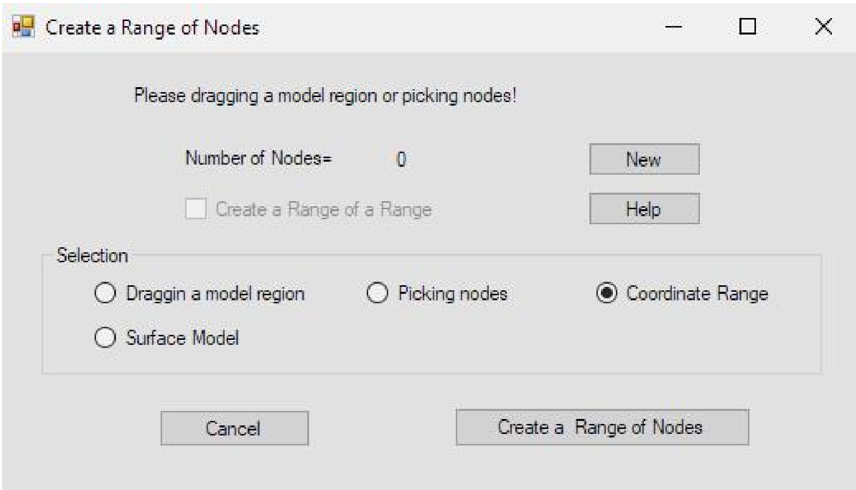
Boundary Conditions

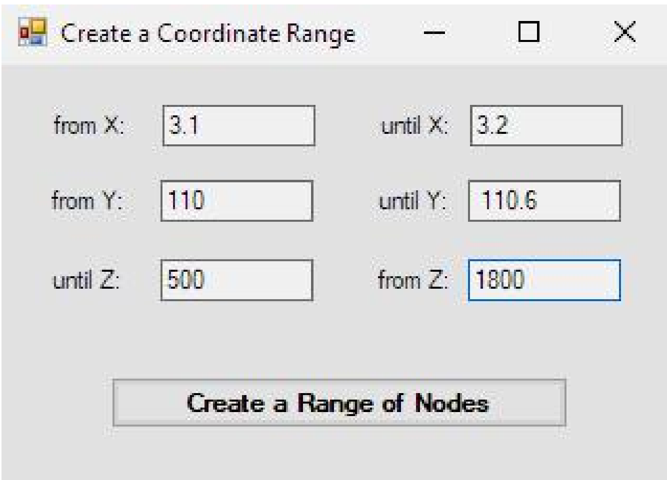
The IPE is clamped fixed on the left side, the BCs are created with register „Edit FEM-Projects“ and menu Boundary Conditions“ by clicking on to Surface 6.



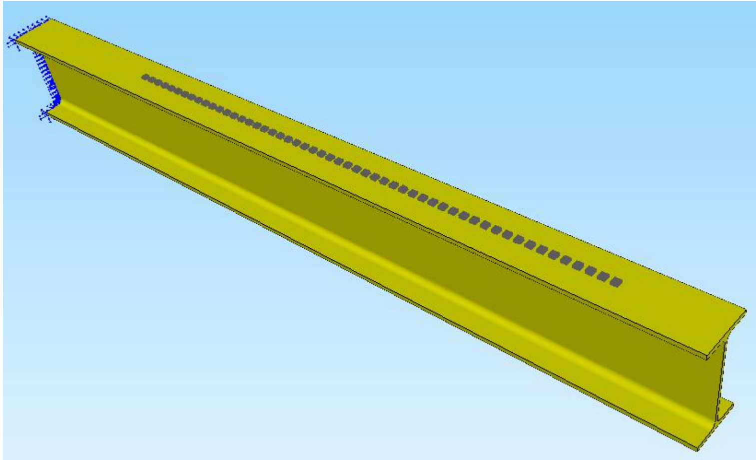
Create a Range of Nodes for Line Load

In order to generate a trapezoidal line load, a range of nodes must first be defined in the Node-Modus with "Create a Node of Range" "and „Coordinate Range":



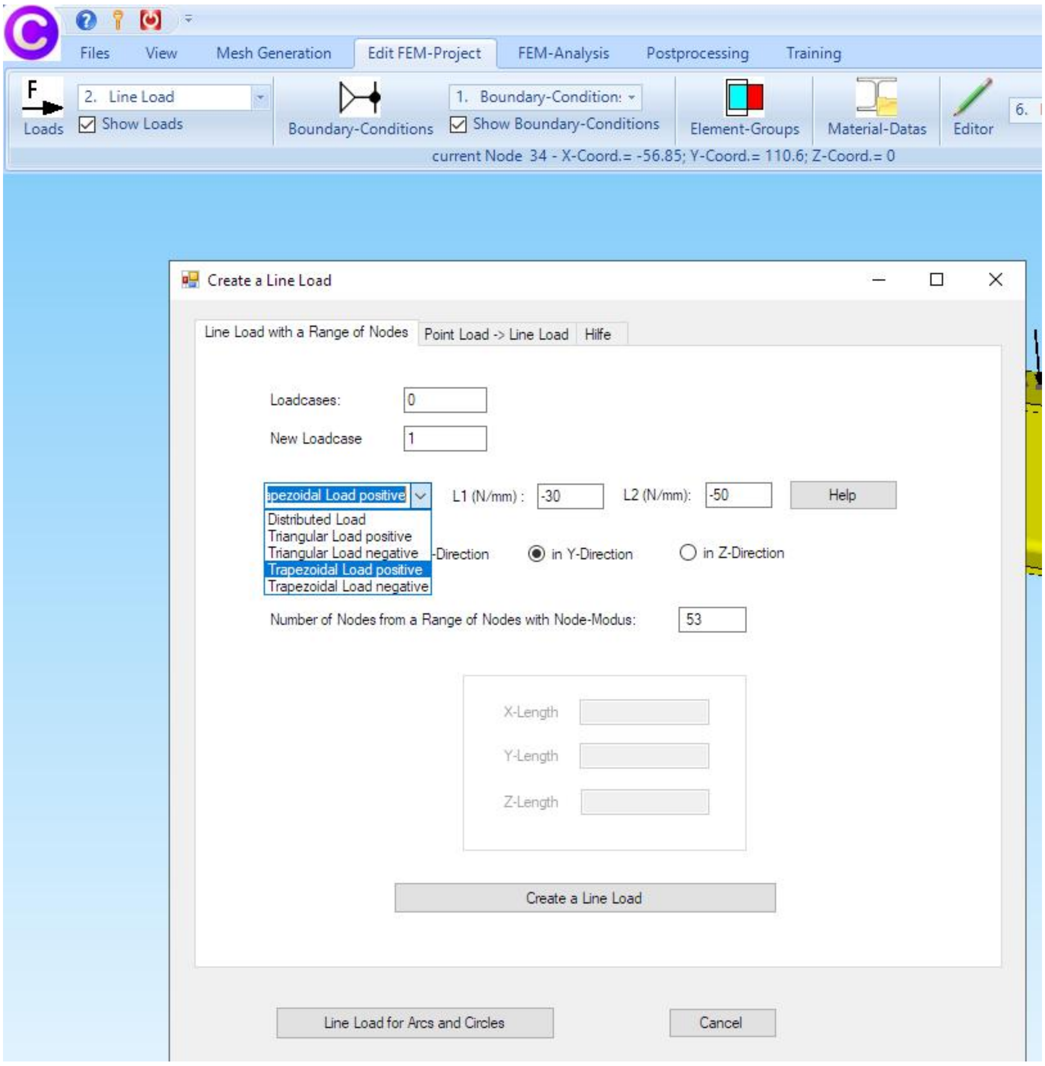


You should see the following node range with 53 nodes from 500 mm to 1800 mm:

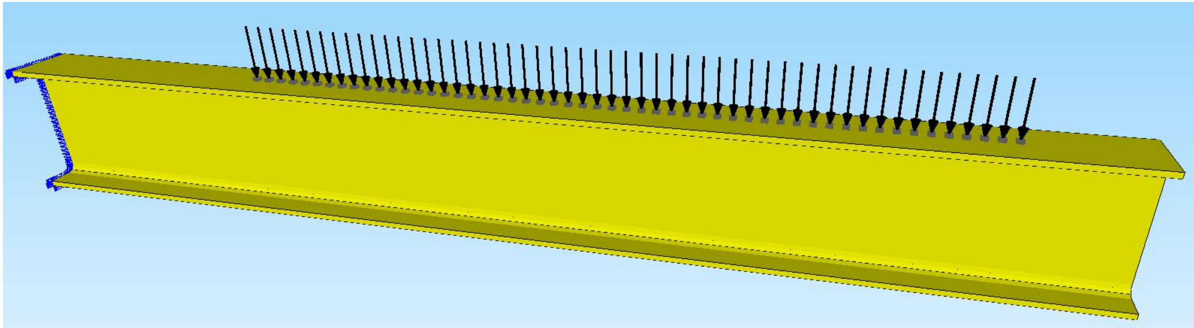


Create a trapezoidal line load

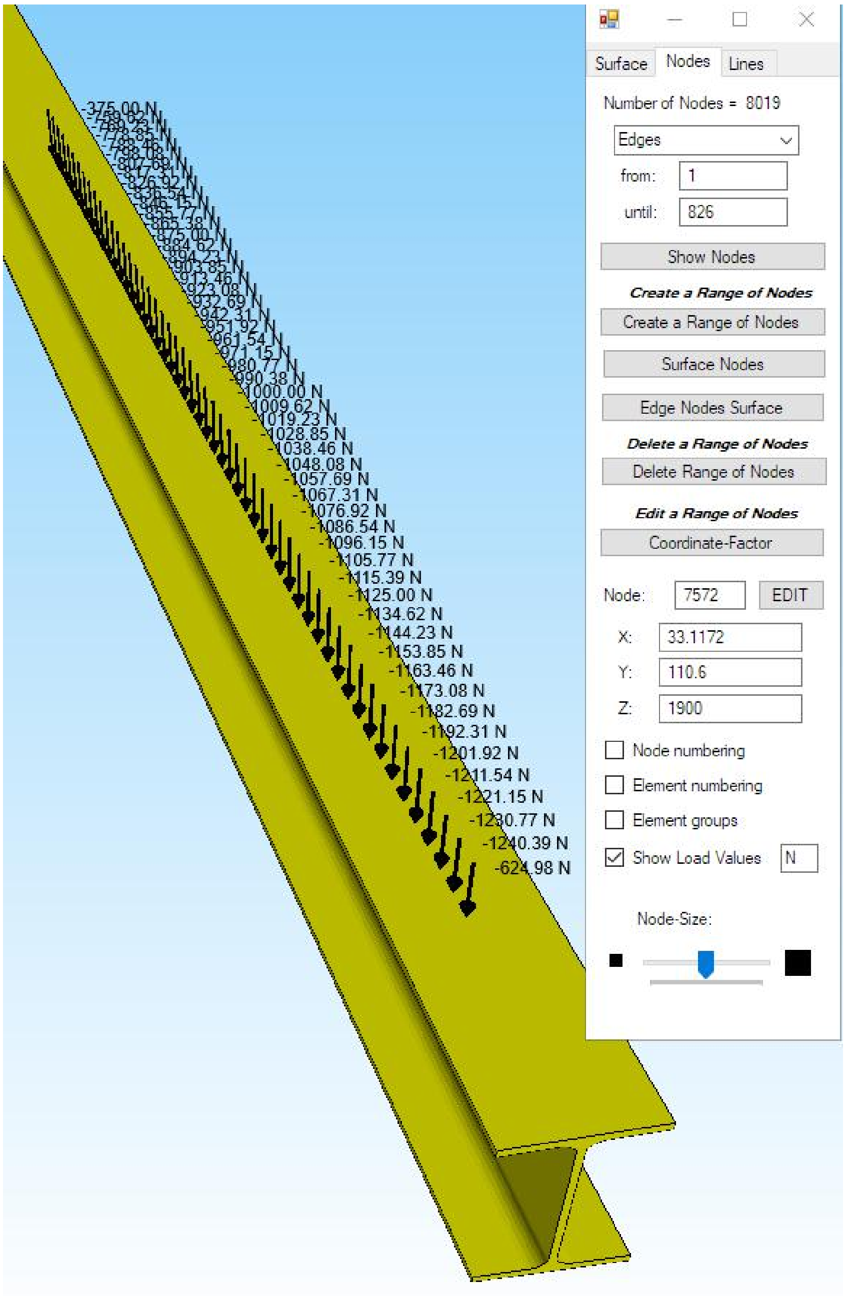
Select the "Edit FEM Project" tab and the "Line load" menu and select the drop-down menu "Trapezoidal load positive" in the dialog box to generate a line load with L1 = -30 N/mm and L2 = -50 N/mm at the node range.



Then you should see the following node load with 53 load nodes:

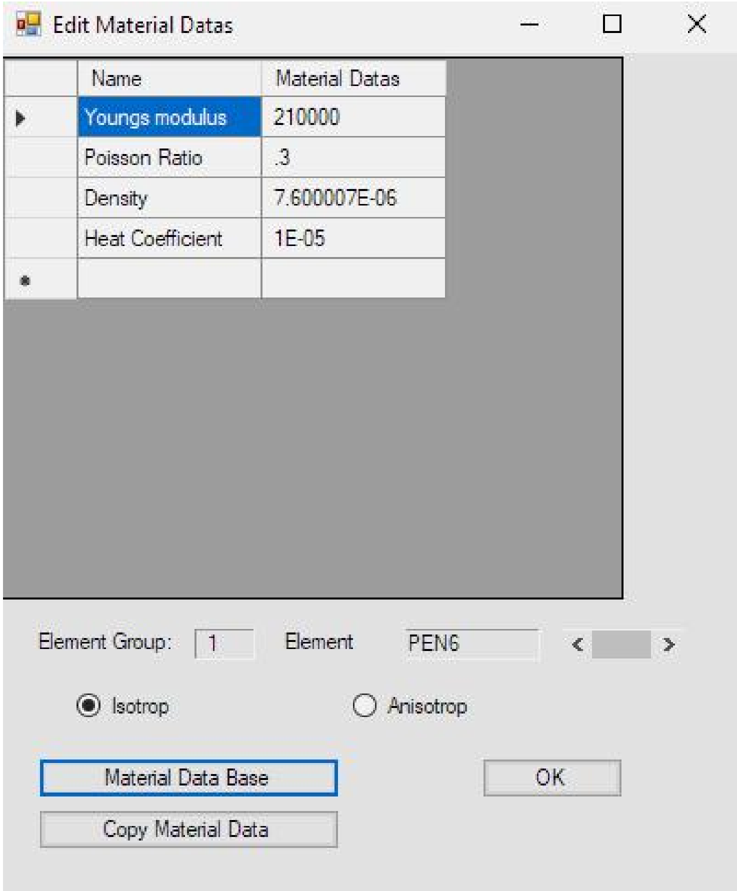


With "Show load values" in the Node Modus, the load values can be displayed and checked:



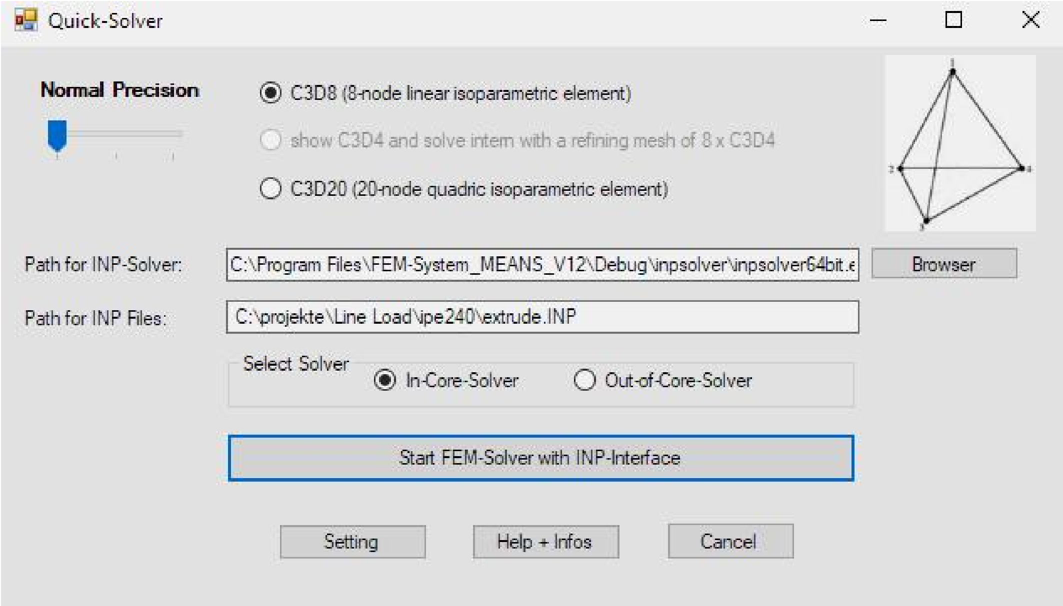
Material Datas

Select register „Edit FEM-Project“ and „Material Datas“ to enter the material datas for Steel with an E-Modulus of 210 000 N/mm² and a Poisson-Value = 0.3.



FEM Analysis

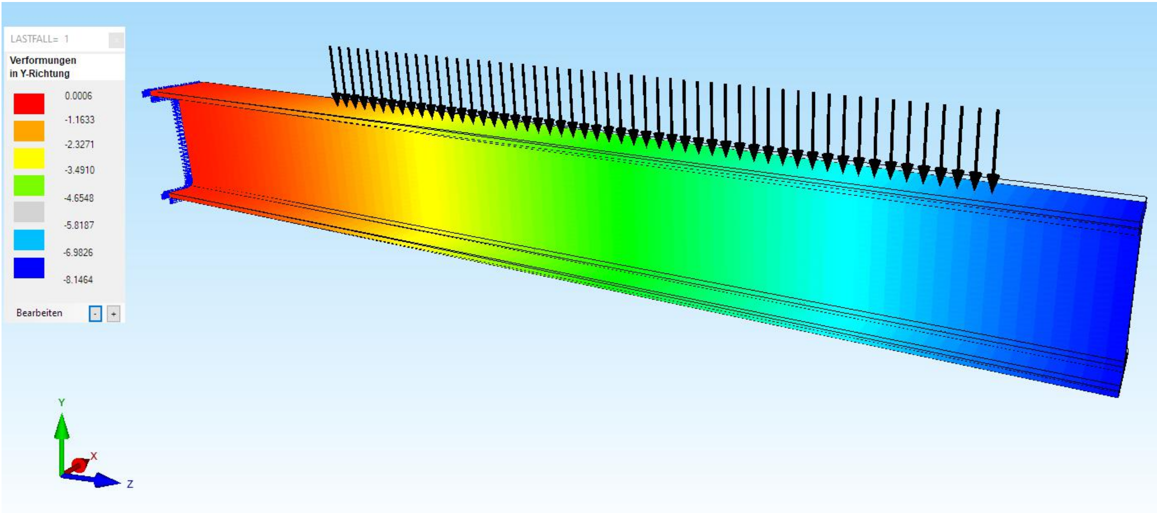
First save the FEM model under any name on the hard disk, select the “FEM Analysis” tab and carry out a static analysis with the Quick-Solver.



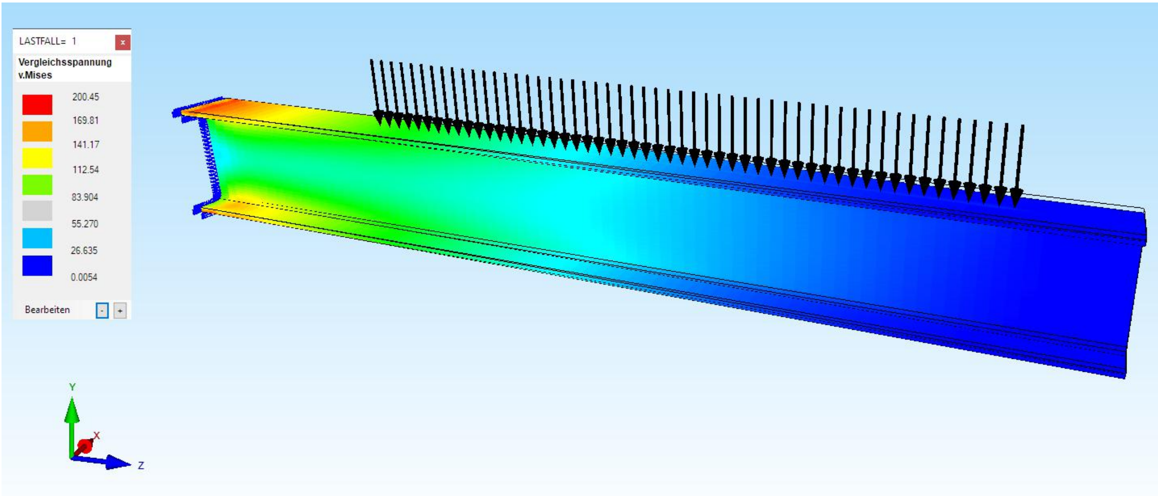
Postprocessing

After the FEM Analysis, the results can be evaluated using the icon  with register „Postprocessing“.

Max. Displacements in Y Direction = - 8.14 mm (exactly = -8.19 mm)



Max. v.Mises-Stresses = 200.4 N/mm² (exactly = 204 N/mm²)



Sum of Reaction Forces = 52000.05 N (exactly = 52 kN)