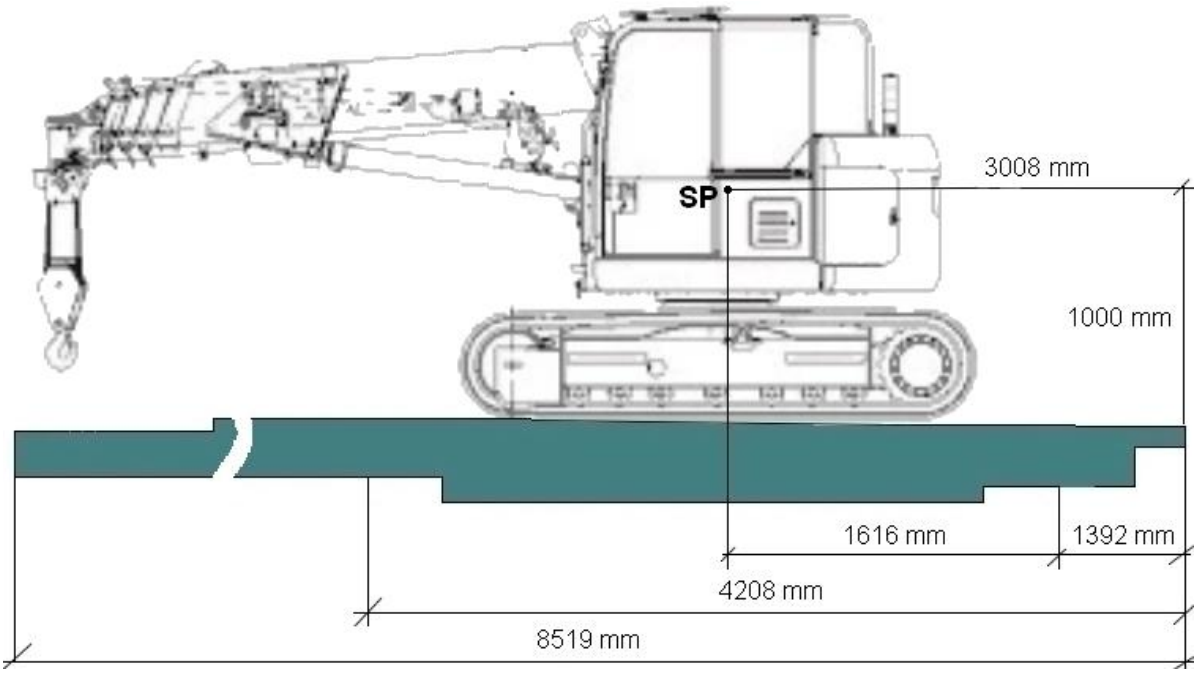
Part 28: Heavy duty trailer with a weight and inertial force

An excavator with a dead weight of 15t is transported with a heavy-duty trailer. How high are the deformations and stresses when braking hard on the beam girders of S355 structural steel.

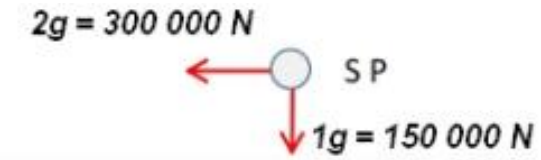
15t excavator with heavy-duty trailer

Dimensions for the center of gravity SP and for the middle part



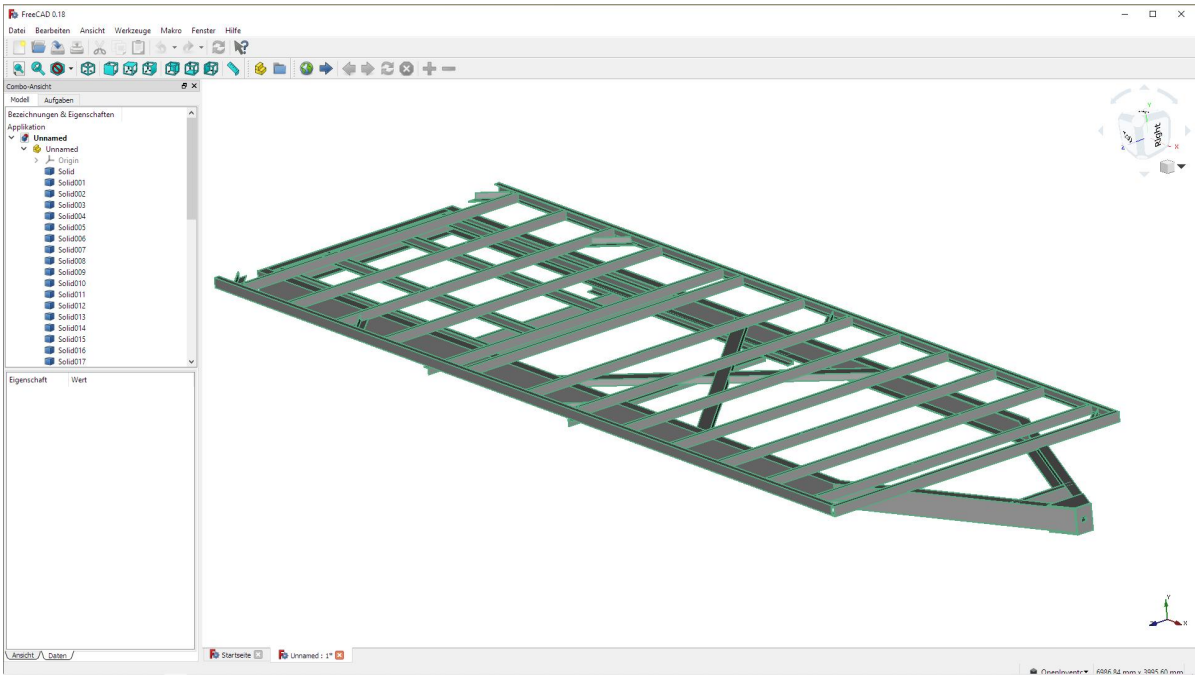
Weight and Inertial Force

Emergency braking in the direction of travel with a forward mass inertia of 2g and a downward weight of 1g.

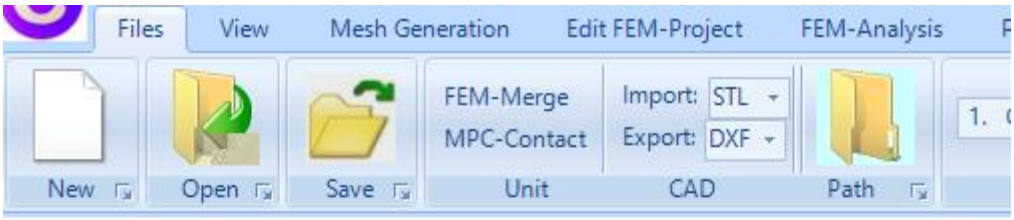


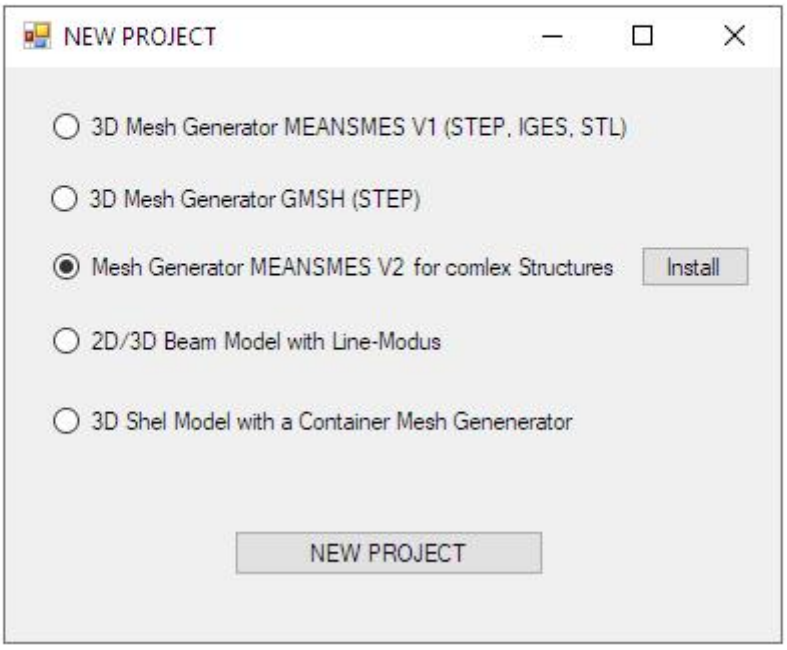
FEM mesh generation

First the CAD assembly is converted into a part since only a part and not an assembly can be meshed and export it in STEP format

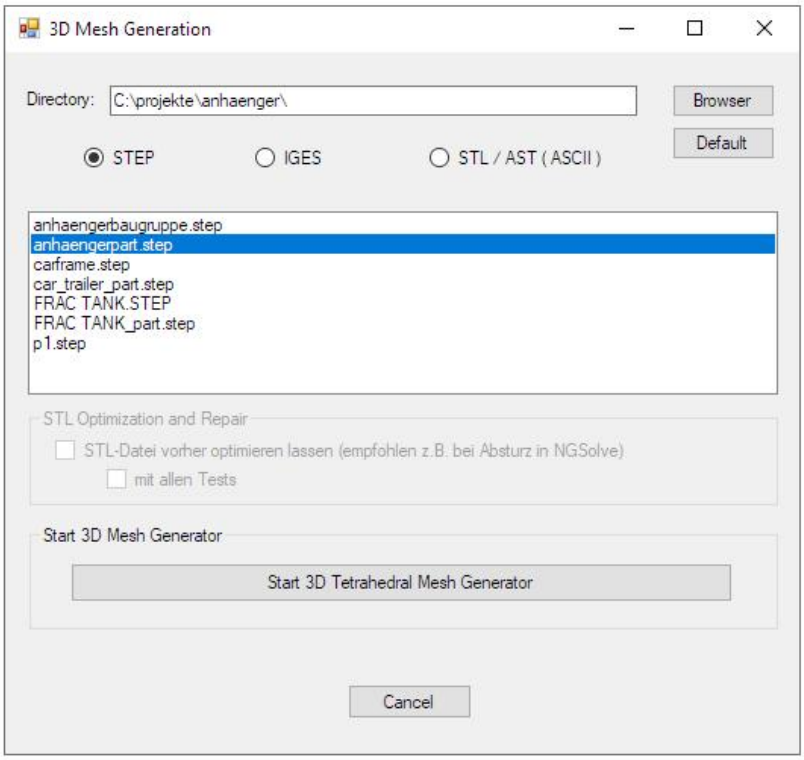


Start MEANS V12 via the desktop icon and select "New" to use the 3D mesh generator "Mesh Generator MEANSMES V2 for complex Structures" available from August 1st, 2022 and select the desired STEP file.

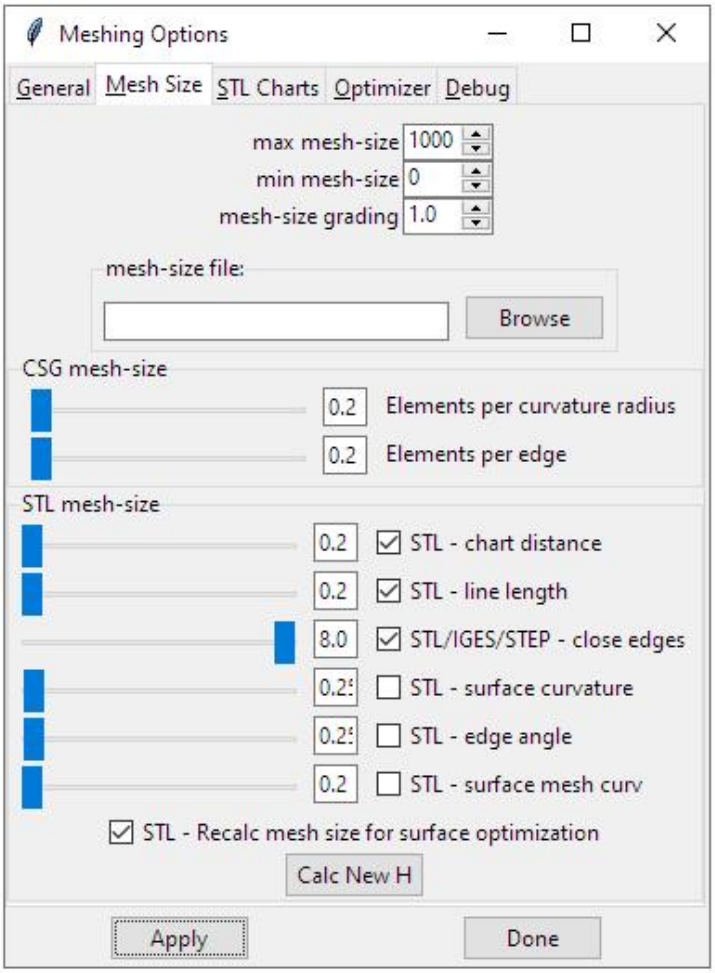




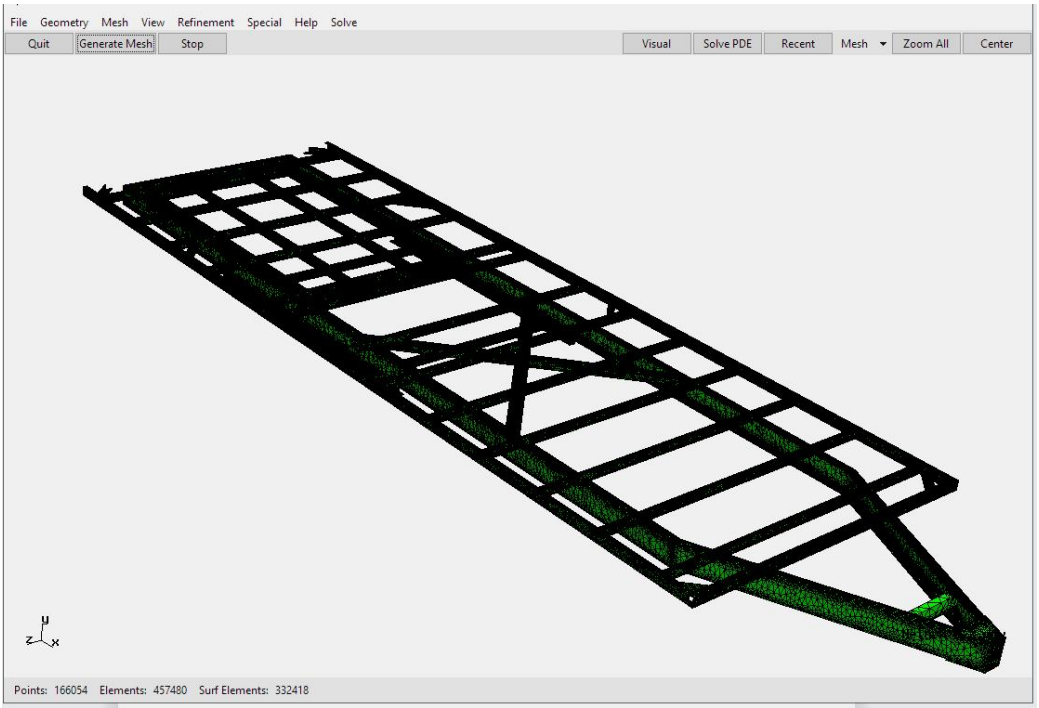
Select the "Start mesh generator with CAD file" menu to display the STEP model in the 3D mesh generator in a new frame window.



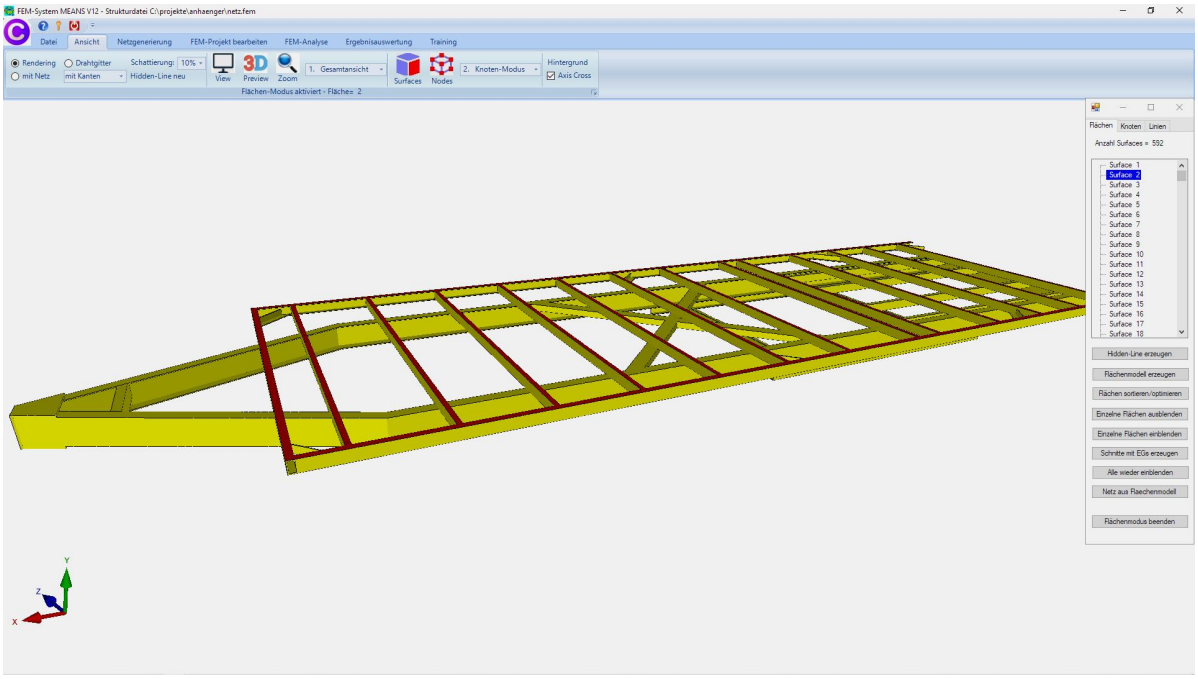
Since the trailer consists of very thin U-profiles and a moderate FEM mesh of approx. 450,000 tetrahedrons is to be generated, the following setting must first be set with the Mesh/Meshing Options menu:



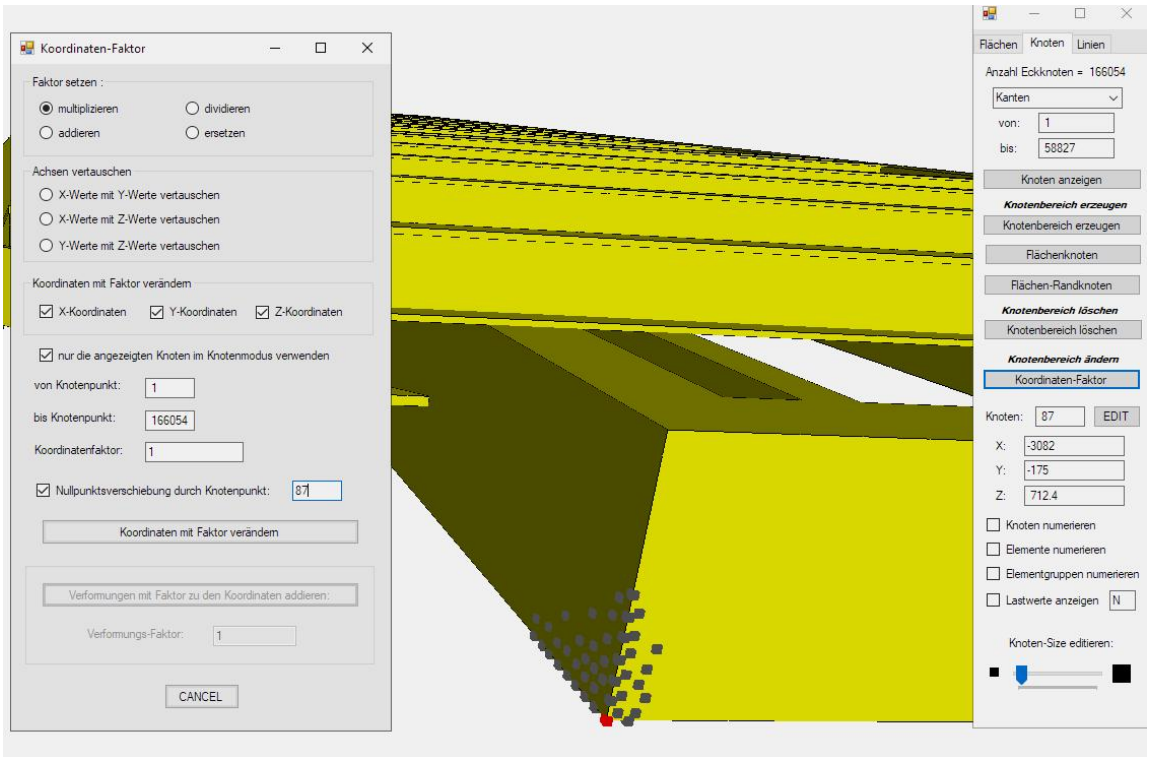
Select "Generate Mesh" to create a mesh with 166054 nodes and 457400 elements.



Export the FEM mesh either in Neutral format with *.FEM or in Abaqus format with *.INP. Then switch back to MEANS V12 and open this FEM mesh with the "Open" menu. The surface model will only be needed later for the middle section.



So that the dimensions can be adopted on page 1, a „Node of Range“ must first be created in the lower right corner with the icon from the View tab in order to be able to read corner node 87, then carry out a zero point shift with the "Coordinate Factor" menu.

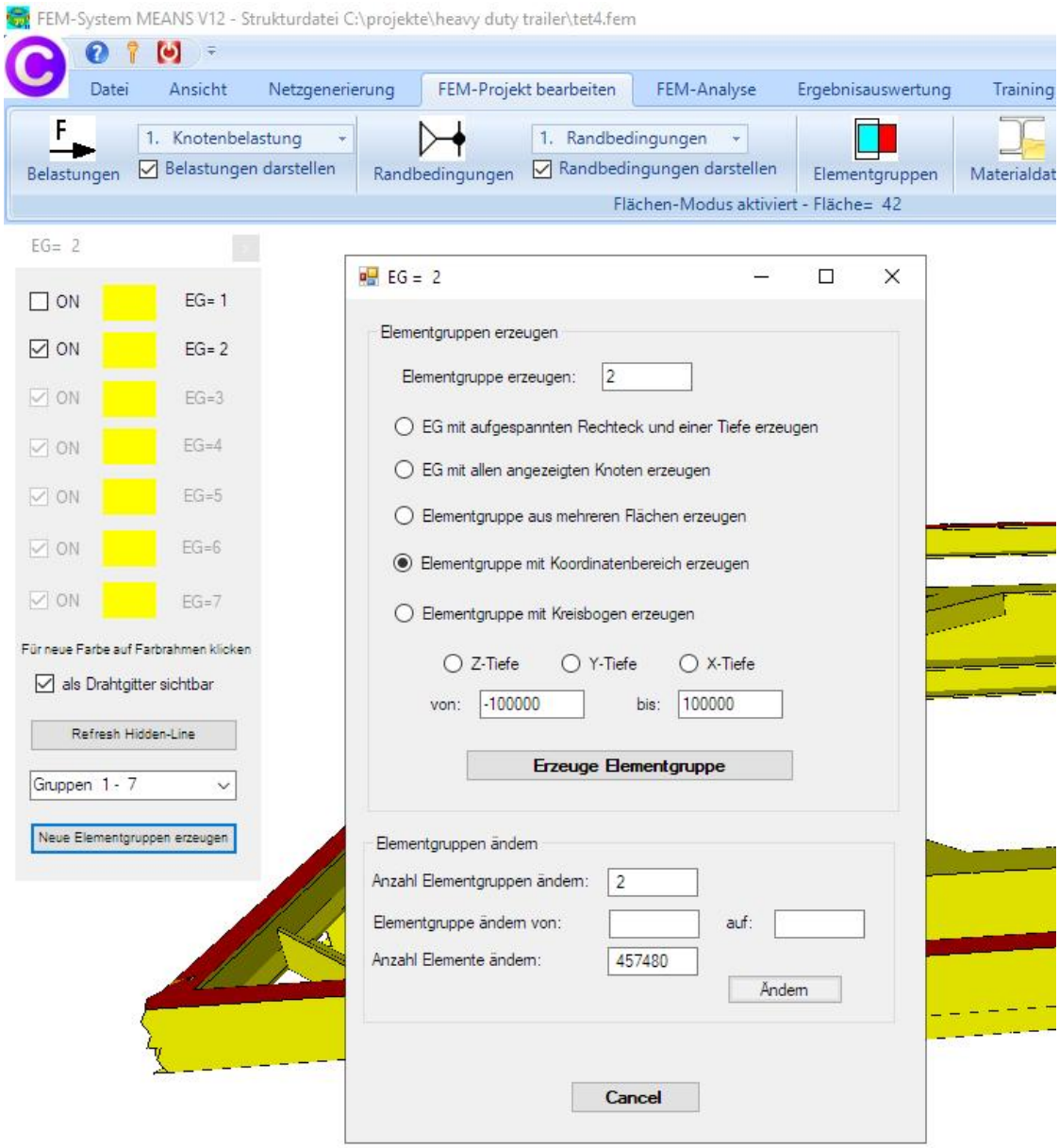


MEANS version

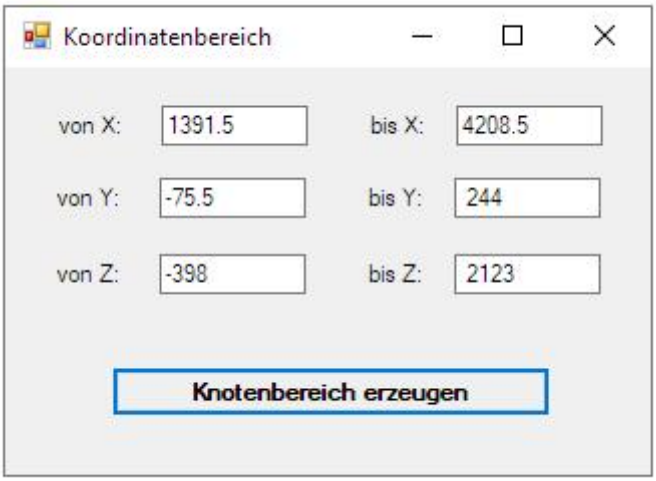
Unfortunately, this model is too large for the user with a **MEANS LITE** version with up to 400,000 nodes and elements. In the CAD the user would have to try to decimate the trailer in non-relevant parts or switch to a higher MEANS version such as **MEANS DESIGN** up to 999,000 nodes and elements or **MEANS HIGH END** with unlimited nodes and elements.

Create Element Group 2

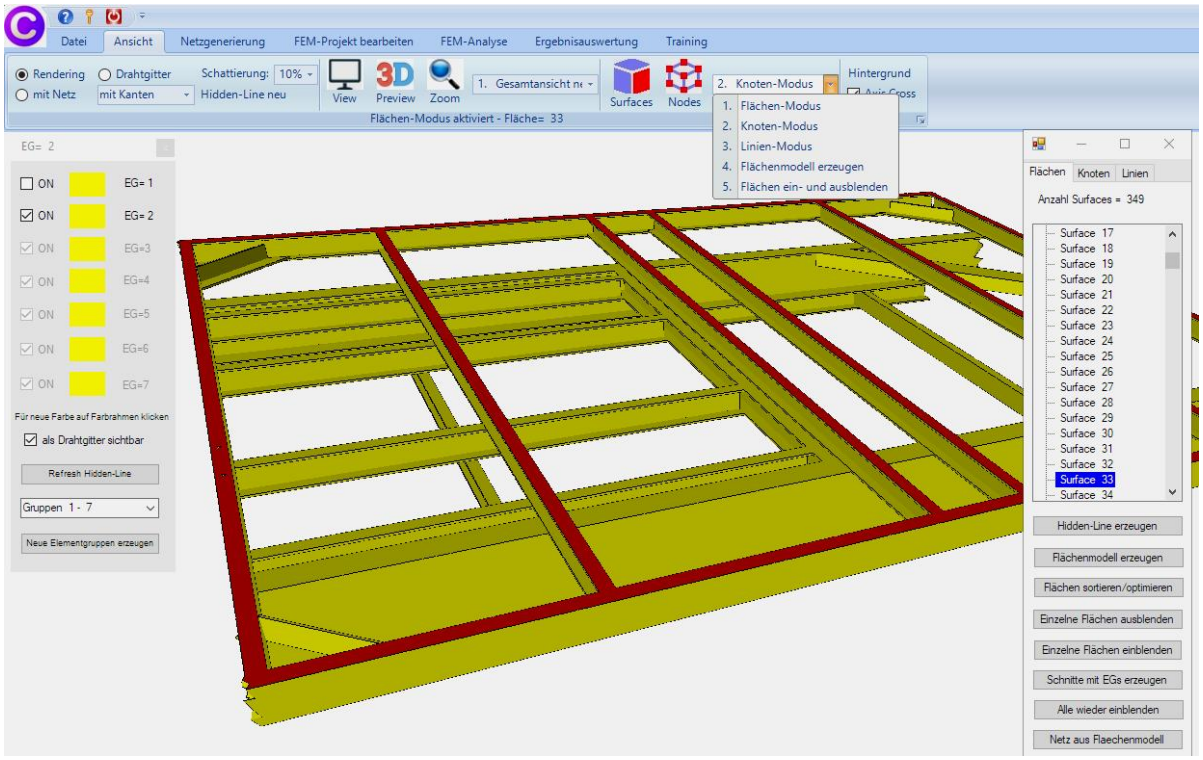
Only the most heavily loaded center section from X= 1392 mm to X= 4208 mm is now used. To do this, element group 2 is created from the middle part and the rest of the trailer is hidden. Select the "Edit FEM project" tab and the "Element groups" menu and create element group 2 using a coordinate range.



Define the coordinate range with the following limits values plus a tolerance of 0.5:

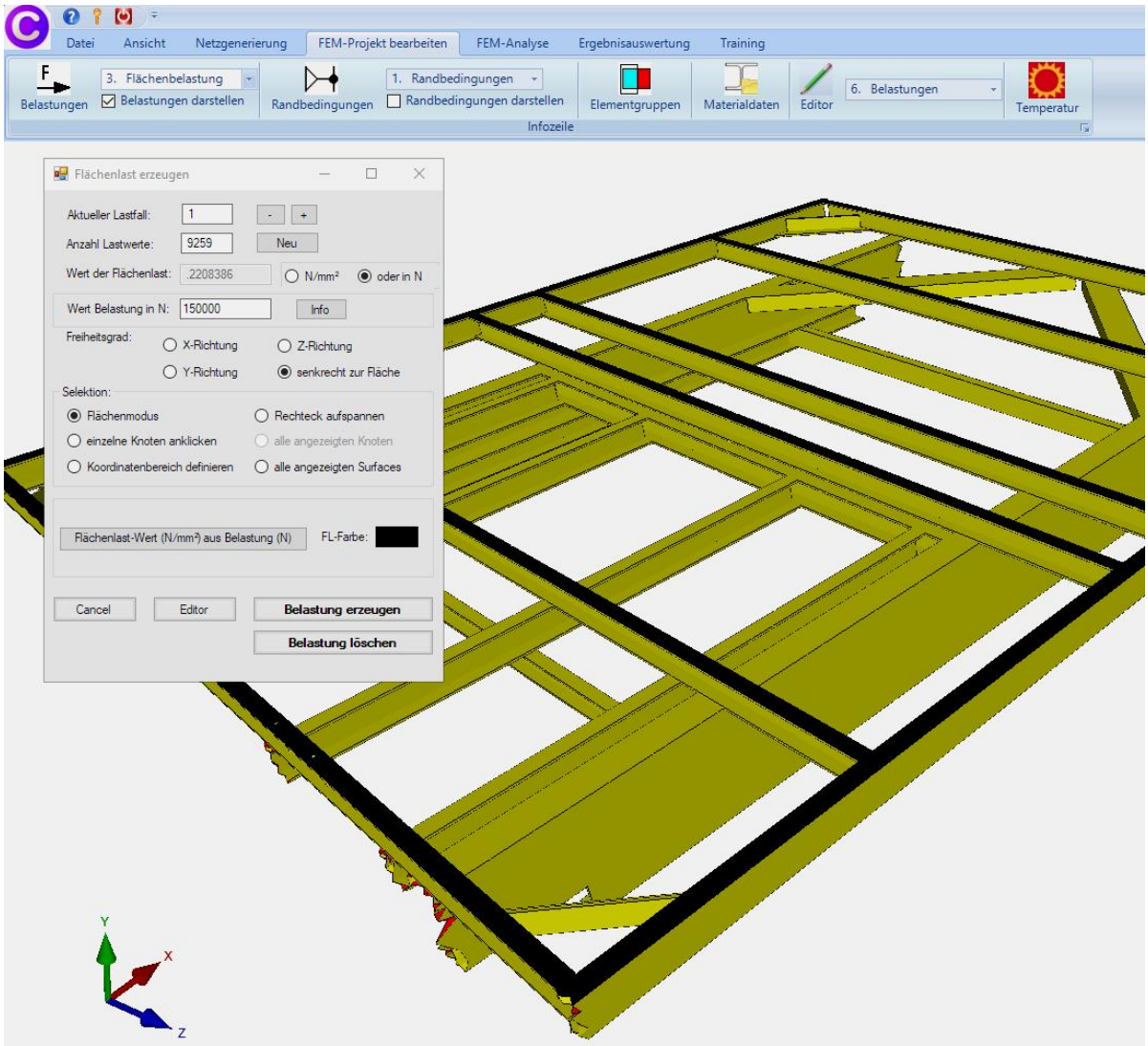


Then first hide element group 1 and create the surface model of element group 2 with the "View" tab and Menü „Surface Modus“.



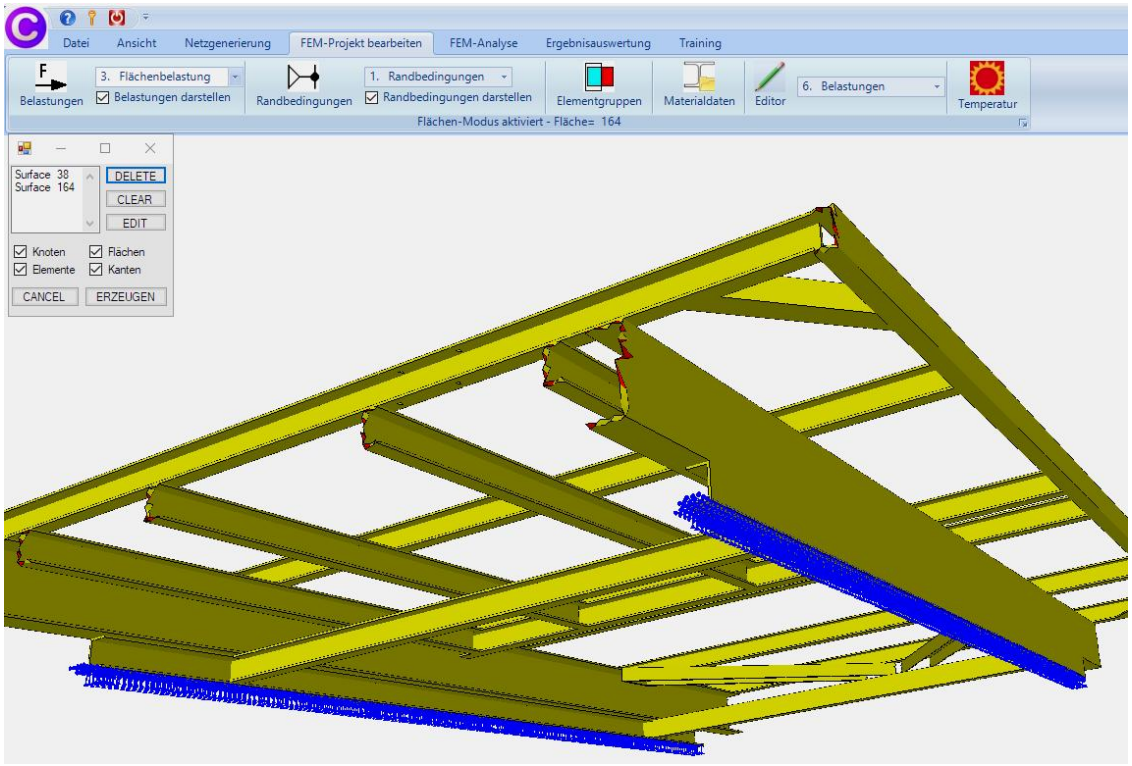
Create a Surface Load

The weight load of 15t can be generated as a surface load. Select the tab "Edit FEM project and "Surface Load" and enter the value of the loading with "150000" N and select menu "Create Load" and click on the surface 33.



Create Boundary Conditions

The trailer is clamped fixed on the lower surfaces. Select the "Edit FEM Project" tab and create the Boundary Condtions on the Surface 38 and 64.

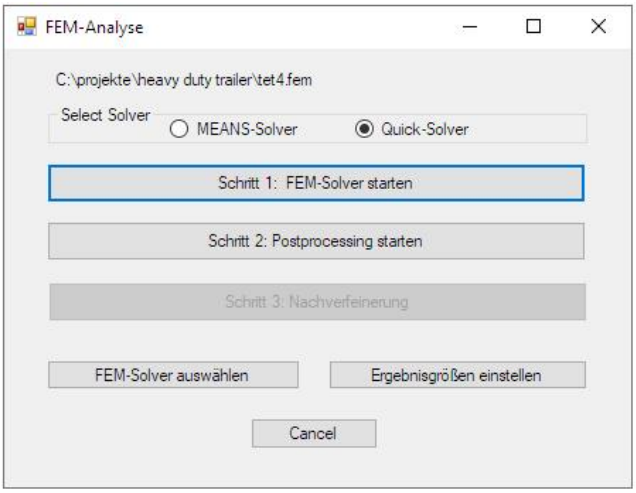


Material Data

The material steel S355 is always preset with a modulus of elasticity of 210 000 MPa and a Poisson number of 0.3 like all other types of steel.

FEM Analysis

After saving the FEM model under any name, select the "FEM Analysis" tab and the "Statics" menu to use the Quick Solver to calculate the deformations and stresses.

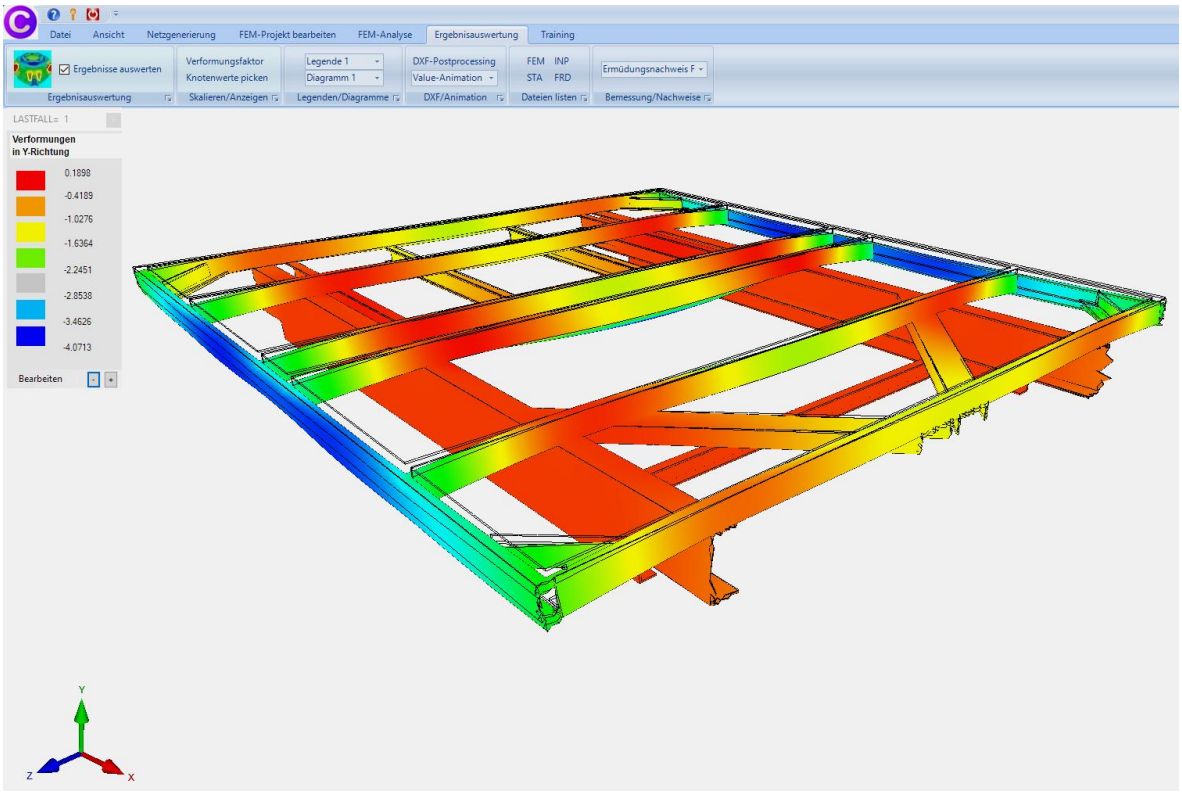


Postprocessing

The displacements and stresses are evaluated graphically with the "Postprocessing“ tab.

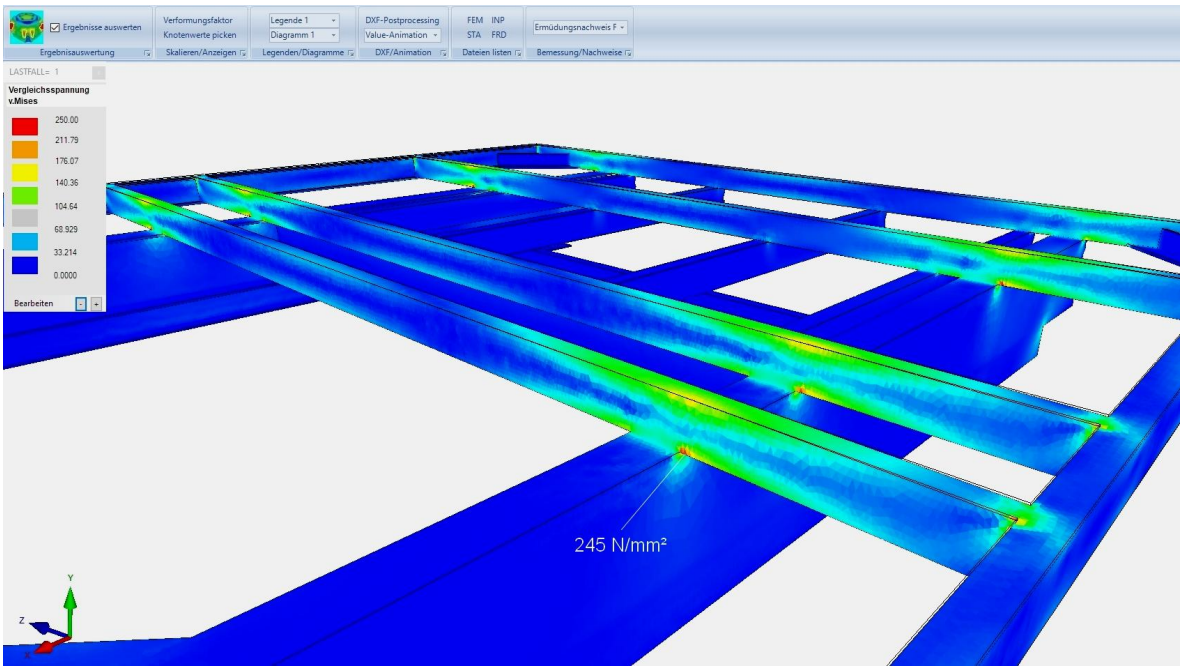
Displacements

max. Displacements in Y direction = -4.0713 mm



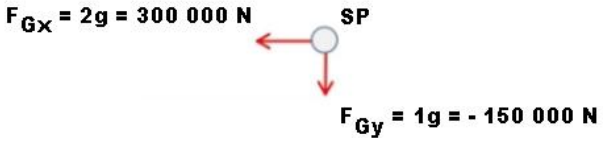
v.Mises-Stresses

The max. v.Mises stress is 245 MPa and is below the Yield Strength of 355 N/mm².

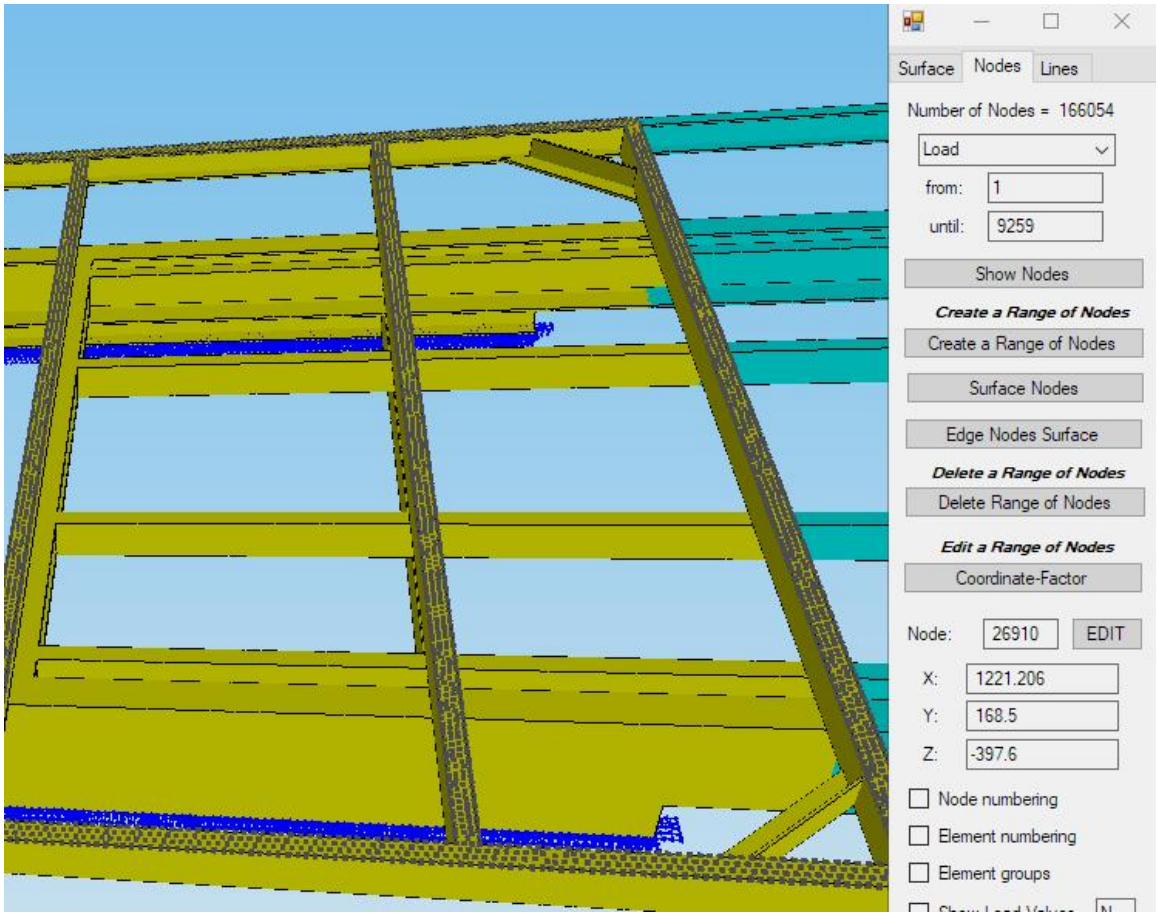


Generate mass inertia forces

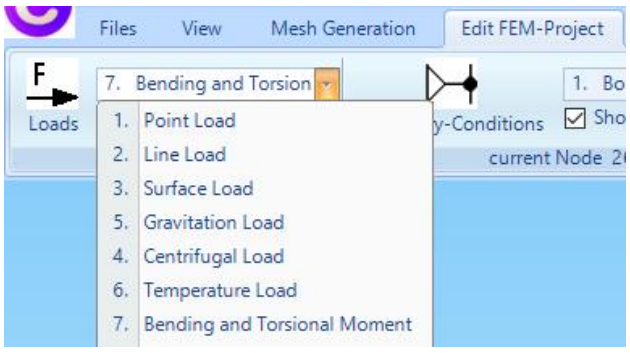
The mass inertia forces act in the center of gravity SP and must be evenly projected onto the center part of the trailer using MPC elements. The downward Y inertia forces are 1g with FGy = -150,000 N and the forward X inertia forces are 2g with FGx = 300,000 N.



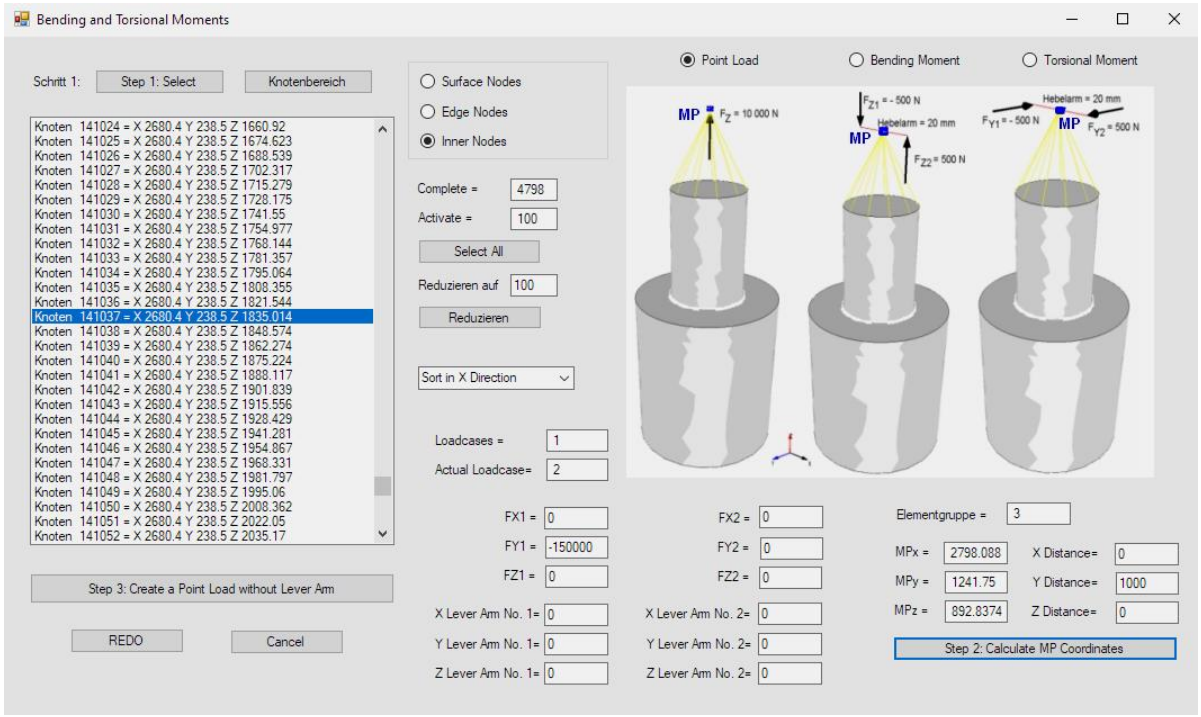
First, the surface load is shown with the View tab and Node-Modus menu, as well as the side menus „Load“ and „Show Nodes“ to create an „Range of Nodes“ of the load.



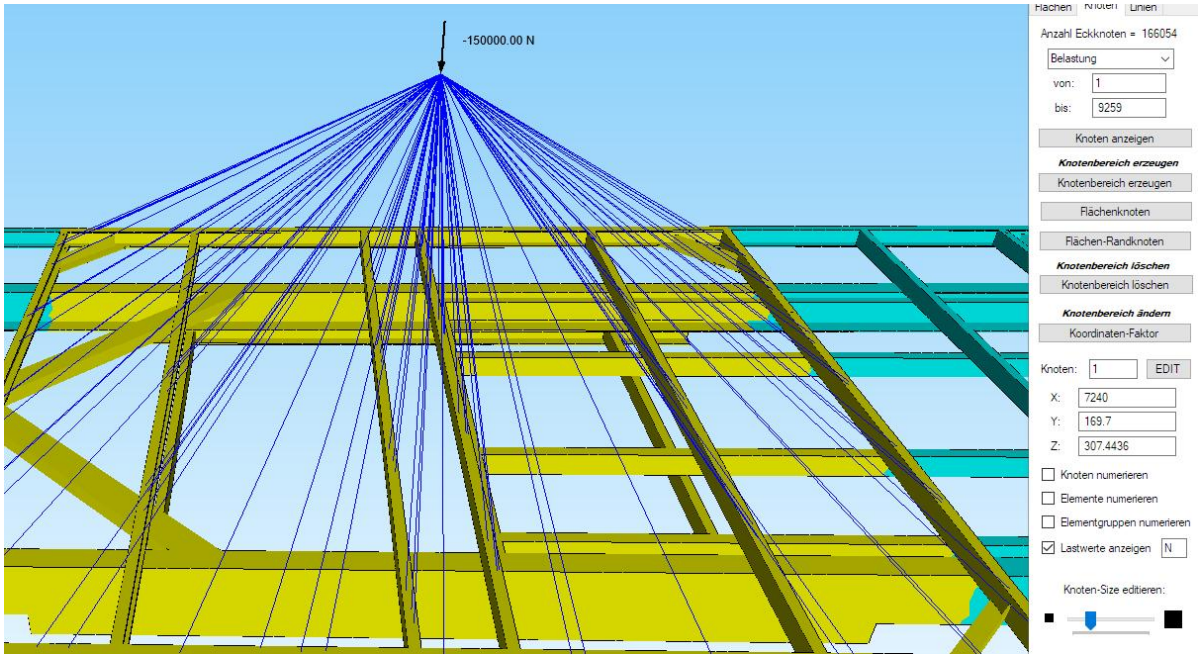
With the „Edit FEM-Project“ tab and the„Bending and Torsional Moment“ menu



all nodes of the „Range of Nodes“ are listed. With "Inner Nodes" only the 4798 inside nodes are shown. The internal nodes must then be reduced to 100 nodes using the "Reduce" menu, since only 100 nodes can be calculated with an MP node in the FEM solver.
Enter a "Point Load" in Y direction of "-150000 N" and a MP distance in Y direction = 1000 mm and select menu "Step 2 - Calculate MP Coordinates" and menu "Step 3 - Create a Point Load without Lever Arm“ to generate a Point Load.



A Point Load of -150000 N is generated at the MP node, which is connected to the 100 selected nodes via MPC elements.



However, the MPC elements can also be created individually in the Line-Modus by connecting the 100 nodes with lines to the MP node.

Repeat the input and use a second MP node to create load case 2 with a Point Load of "300,000 N" in the X direction and carry out an FEM analysis with the Quick Solver.

The result is load case 2 with superimposed load cases 1+2 with the following v.Mises stress contour: