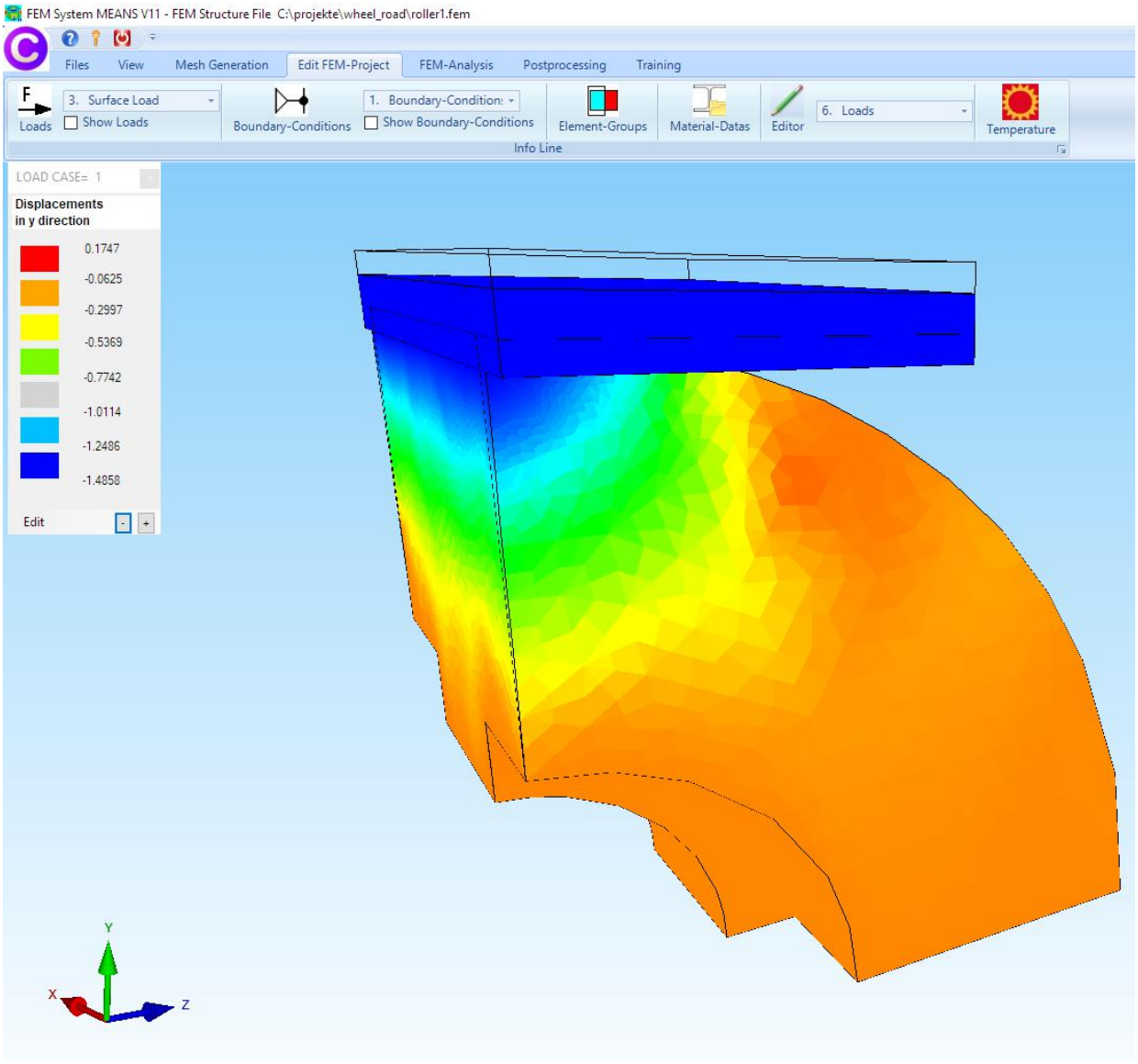
FEM-System

MEANS V11

FEM-Contact-Analysis

of a Polyurethane Wheel



www.femcad.de

www.fem-infos.com

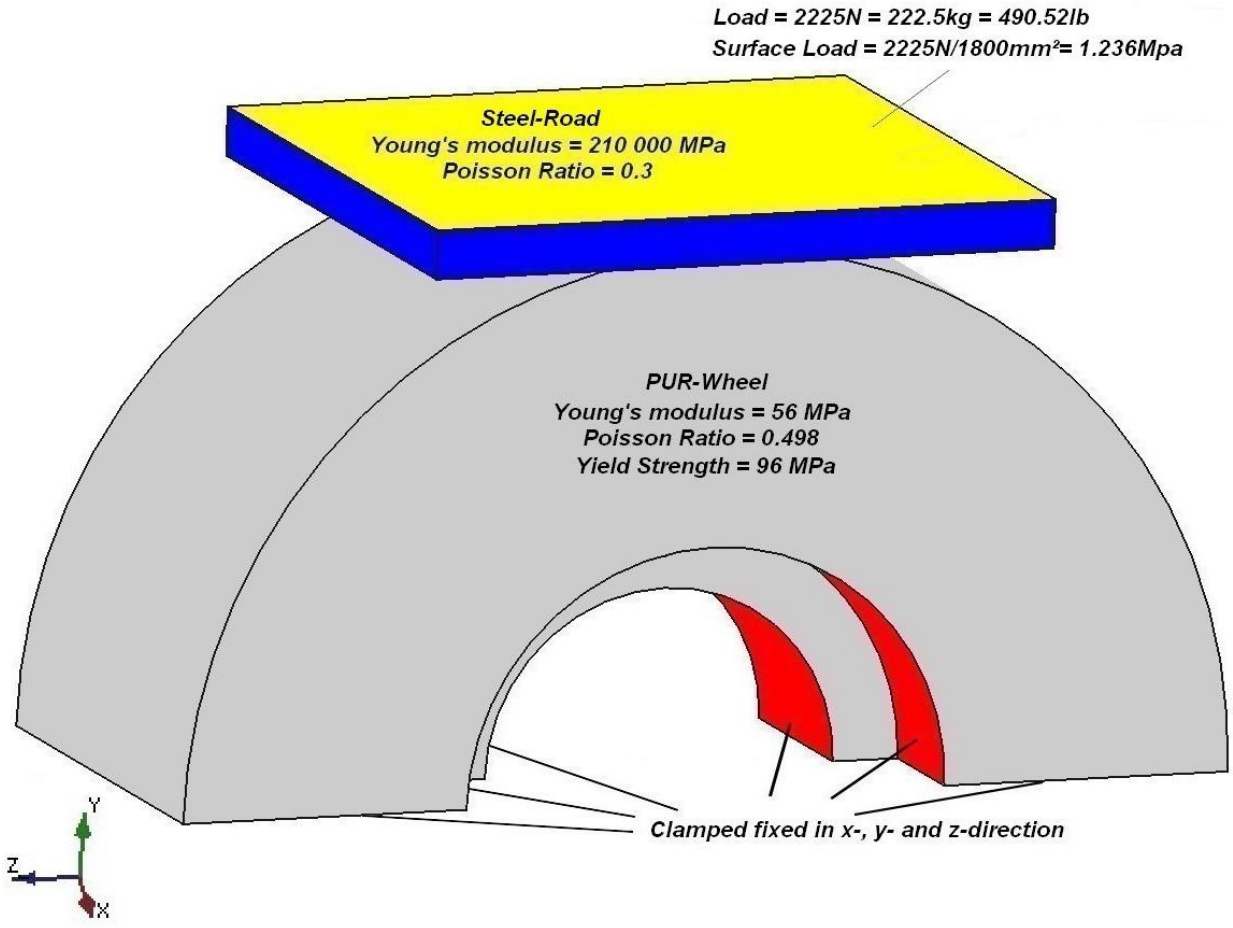
Part 19 - Contact-Analysis of a hot cast Polyurethane Wheel with MEANS V11

The model is of a small 95 Shore A hot cast polyurethane wheel with 56 MPa Young's Modulus, 0.498 Poisson's Ratio and 96 MPa Tensile Yield Strength. To keep the FEM model as small as possible we analyzing 1/4 of the wheel with a symmetry surface.

Full Model

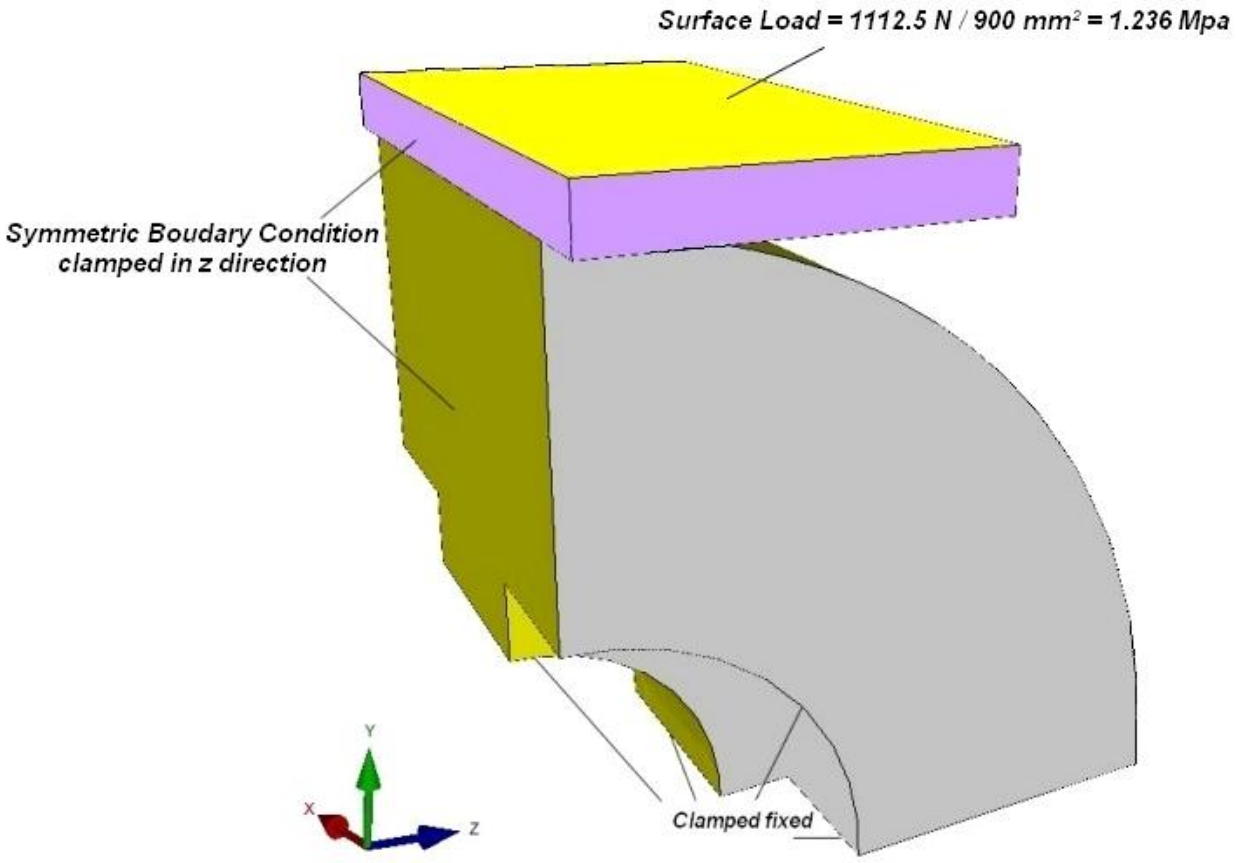
The inner diameter is fixed, it would be bonded to ball bearings. The wheel is loaded with a steel road at 2225N.

Half Model:



$$\text{Surface Load of 1/2 model} = \frac{\text{Load}}{\text{Surface Area}} = \frac{2225\ \text{N}}{1800\ \text{mm}^2} = 1.236\ \text{MPa}$$

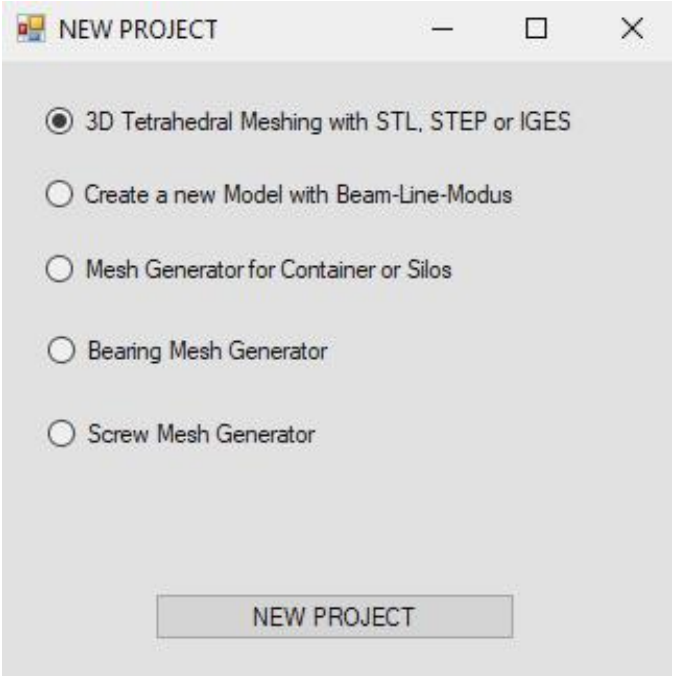
Quarter Model



$$\textbf{Surface Load of 1/4 model} = \frac{\textbf{Load}}{\textbf{Surface Area}} = \frac{\textbf{1112.5 N}}{\textbf{900 mm}^2} = \textbf{1.236 MPa}$$

Mesh Generation

First an FEM mesh is generated from the CAD Step model assembly.



Select the "Files" tab and select "New" to create a new FEM project.

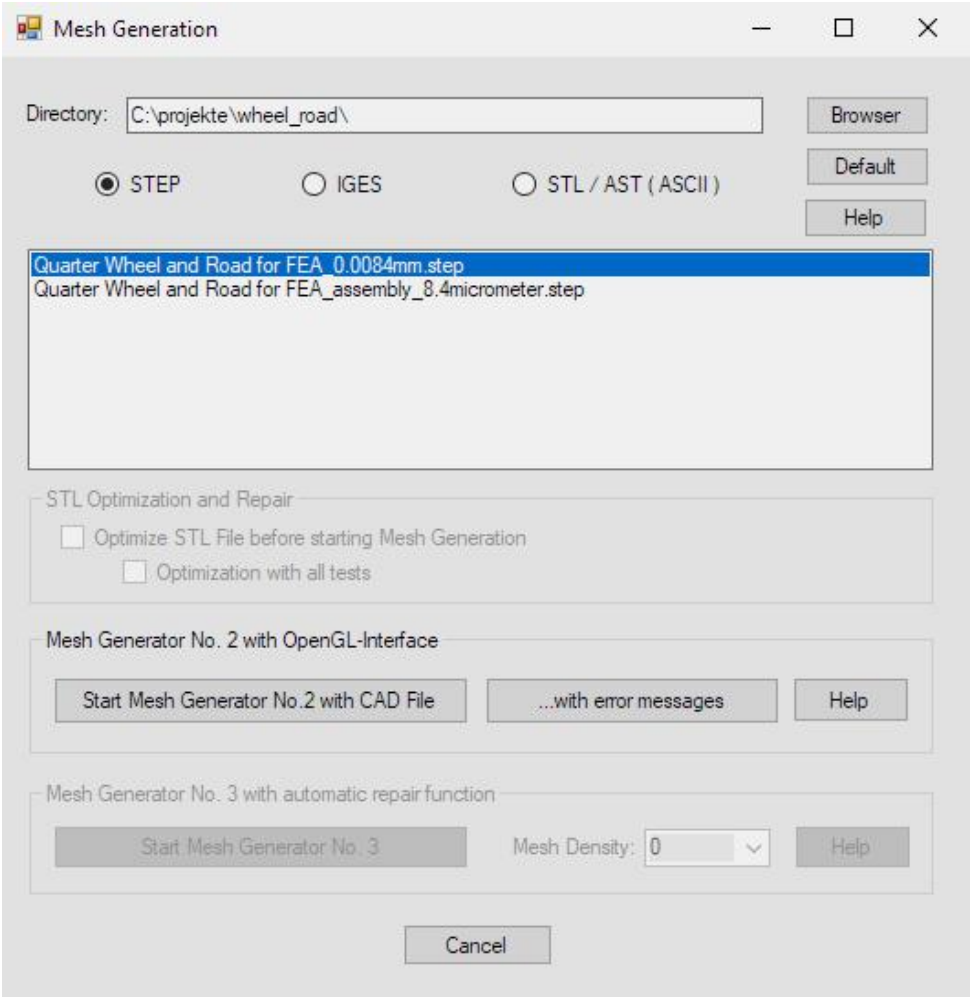
Select "3D Tetrahedral Meshing with STL, STEP or IGES for the following formats:

STL this 3D model consists of a triangular outer shell for the 3D Mesh generation, this flexibel format can also be imported and exported. STL Files with holes or gaps, it can also be repaired before meshing.

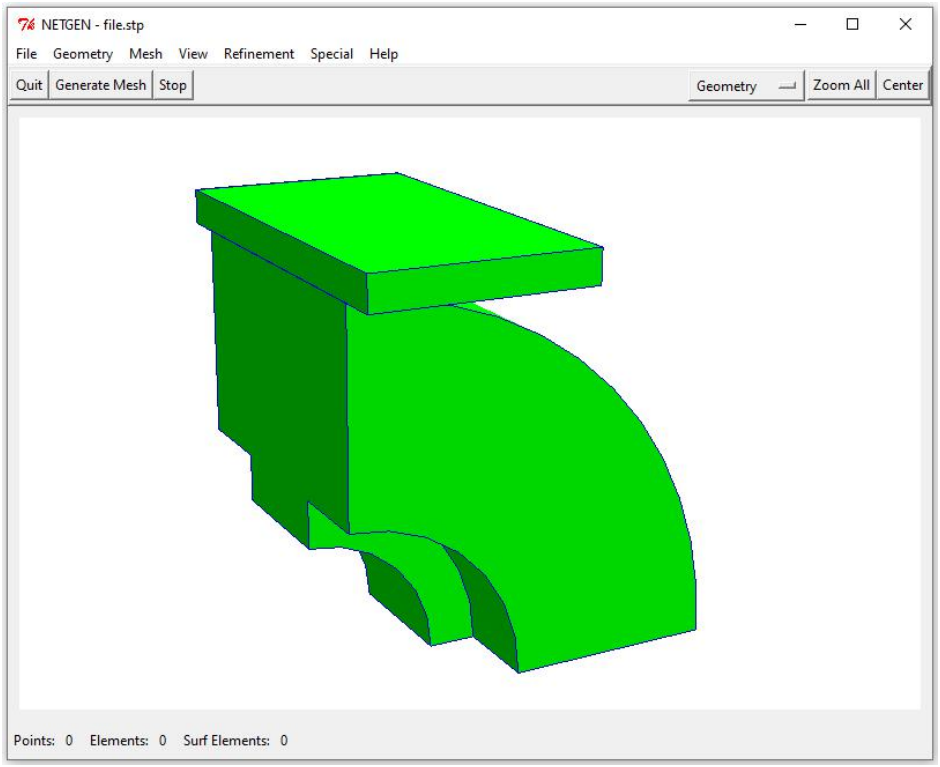
STEP today it is the Standard format, note that no CAD assemblies but only single parts can be meshed. Assemblies can be combined via the “Boolean operations” or with “Screw Models” into one part.

IGES like STEP format but is not as common anymore

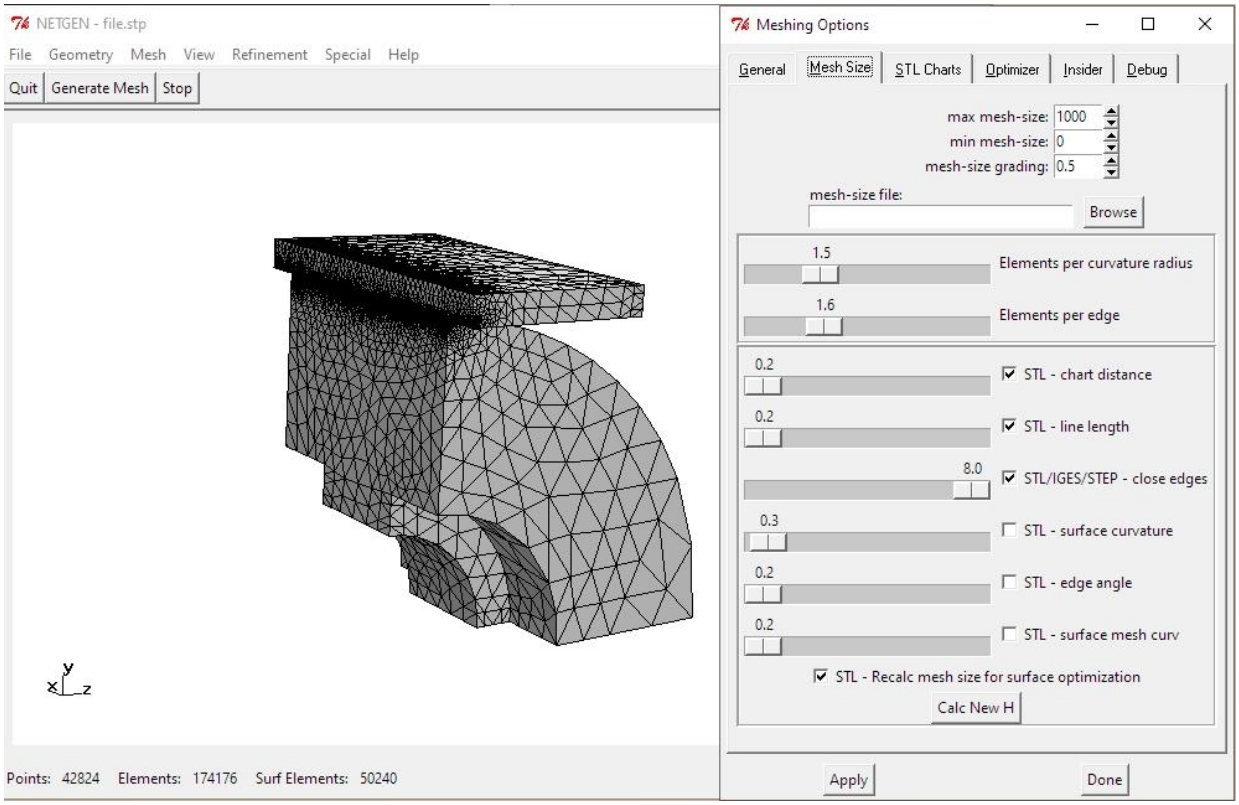
Use the "Browser" button to select the "wheel_road.step" STEP file and click on menu "Start Mesh Generator No. 2 with CAD File" to display it in the mesh generator.



The model can now be seen in the mesh generator and can be rotated as required.



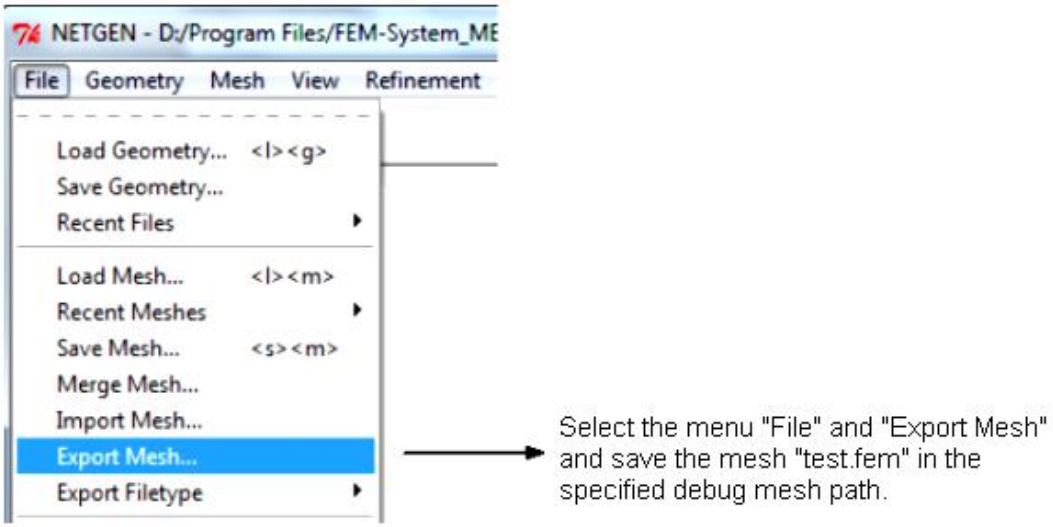
First select the "Mesh" menu and "Mesh-Size" tab and generate with the following setting and a Refinement an FEM Mesh with 174 176 tetrahedral elements with 42 824 nodes. The contact surfaces is also very fine meshed by a contact surface distance of 0.0084 mm or 8.4 Micrometer (could be too much).



Export Mesh

After generating the mesh, the FEM mesh with the name "test.fem" must be exported to MEANS V11. Select the menu "File" and "Export Mesh" and save the mesh "test.fem" in the specified debug mesh path to start MEANS V11 automatically with the FEM-Mesh and the Surface Model for selecting Nodes, Edges and Surfaces.

.



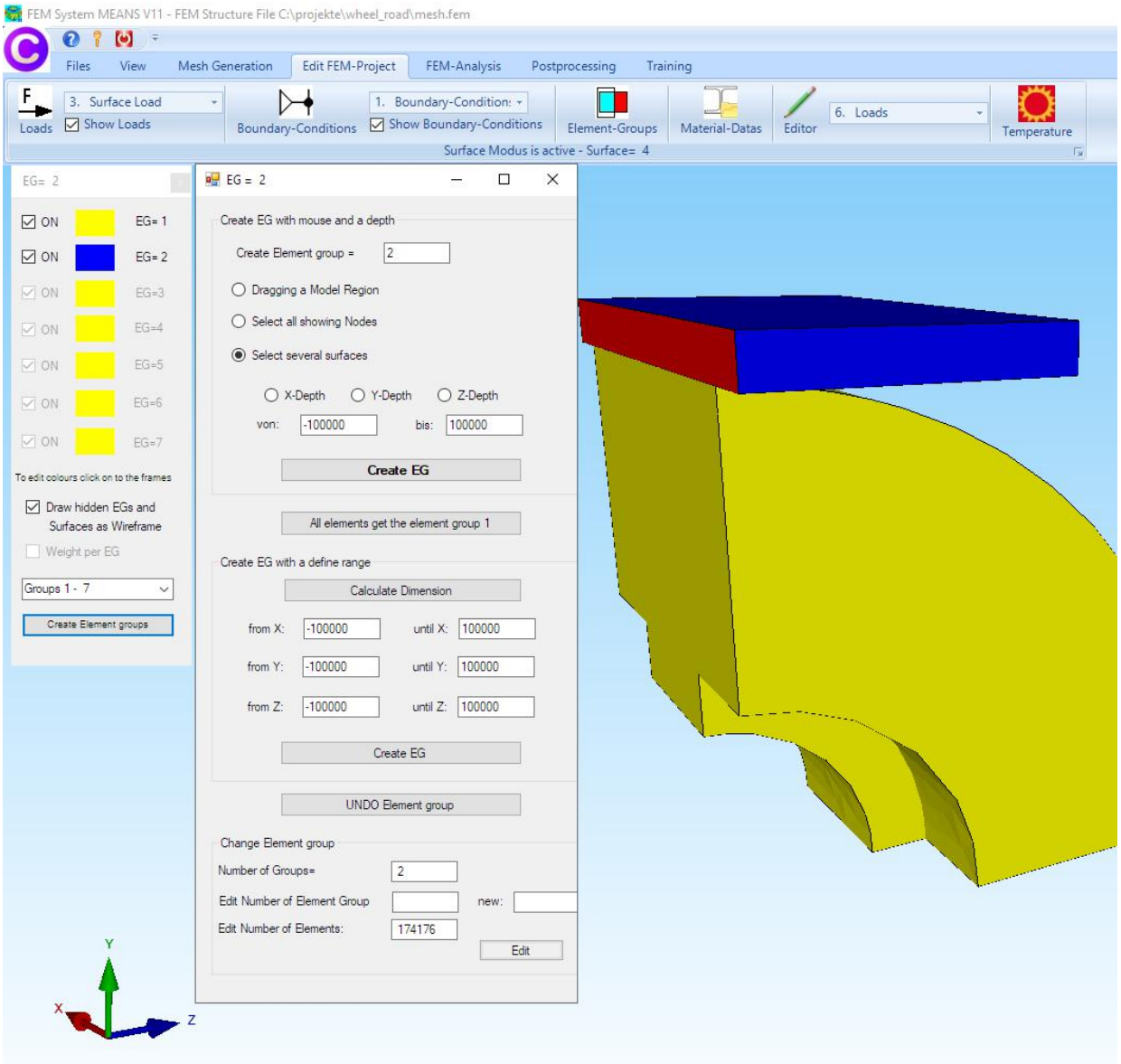
Create Element Group 2

At first we must created Element Group 2 because we are calculated with the two different material datas:

Element Group 1 with the PUR-Wheel and Element Group 2 with the Steel-Road.

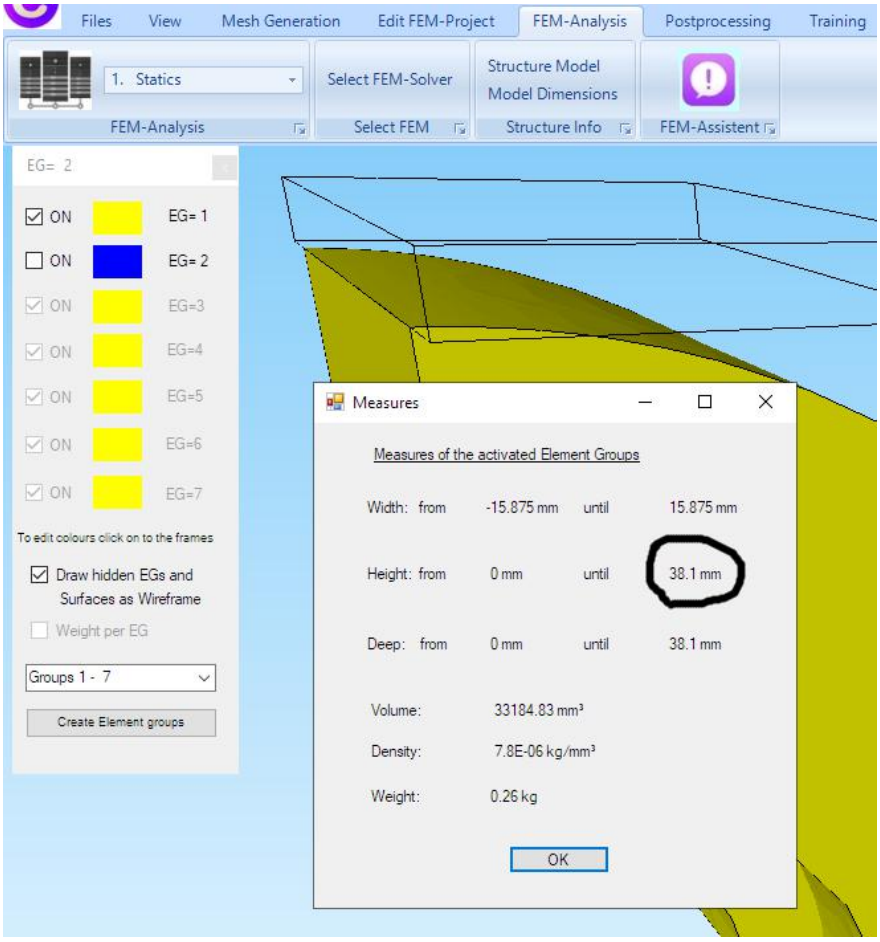
Select the “Edit FEM-Project” and “Element-Groups” tab and choose in the new element group menu “Create Element groups” and select in the new dialogbox the option “Select several surfaces” to create EG 2.

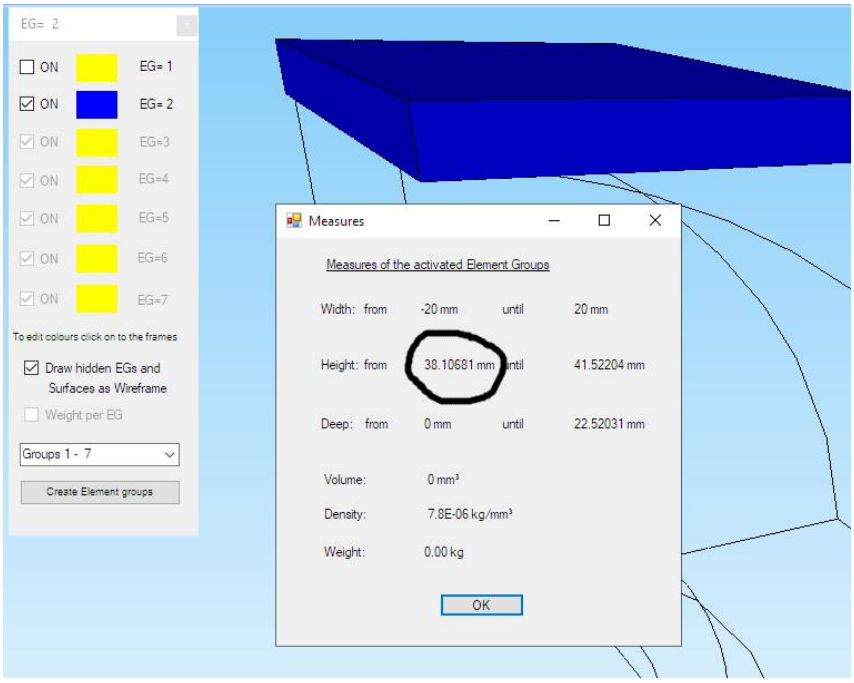
Choose “Create EG” and create Element Group 2 with a click on Surface 4 and 8. At least click on the color-box for a blue EG 2.



Check the Contact-Distance

The Contact-Distance is very important for the Contact-Analysis. Now we check it easy with the FEM-Analysis and Model Dimension tabs for every Element Group.

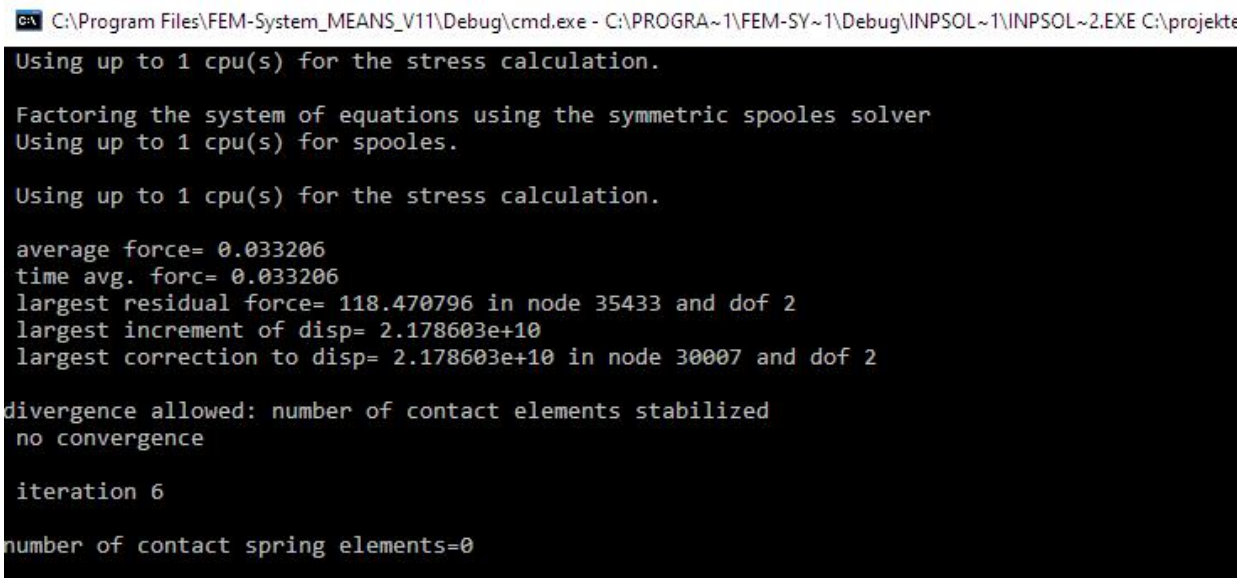




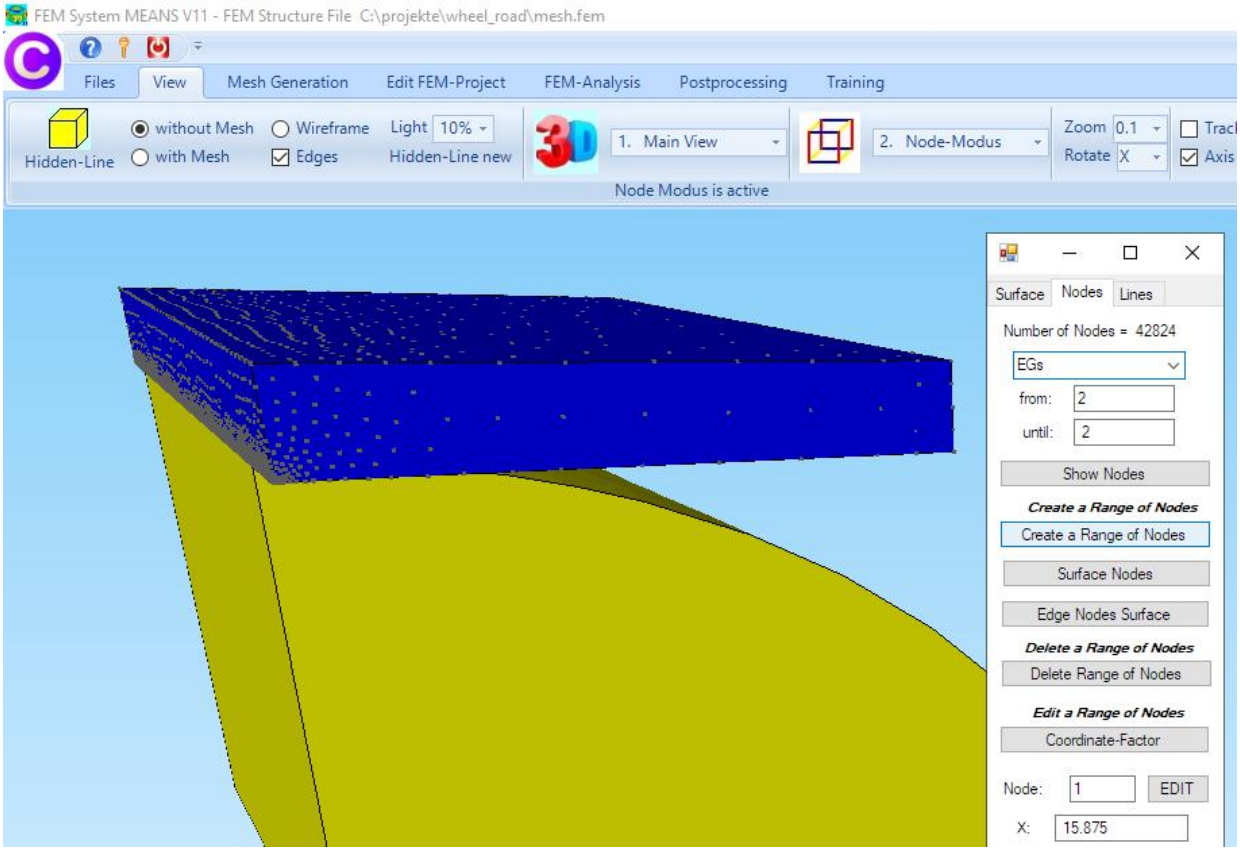
Now we are able to read the Contact-Distance 38.10681mm - 38.1mm = **0.00681mm**

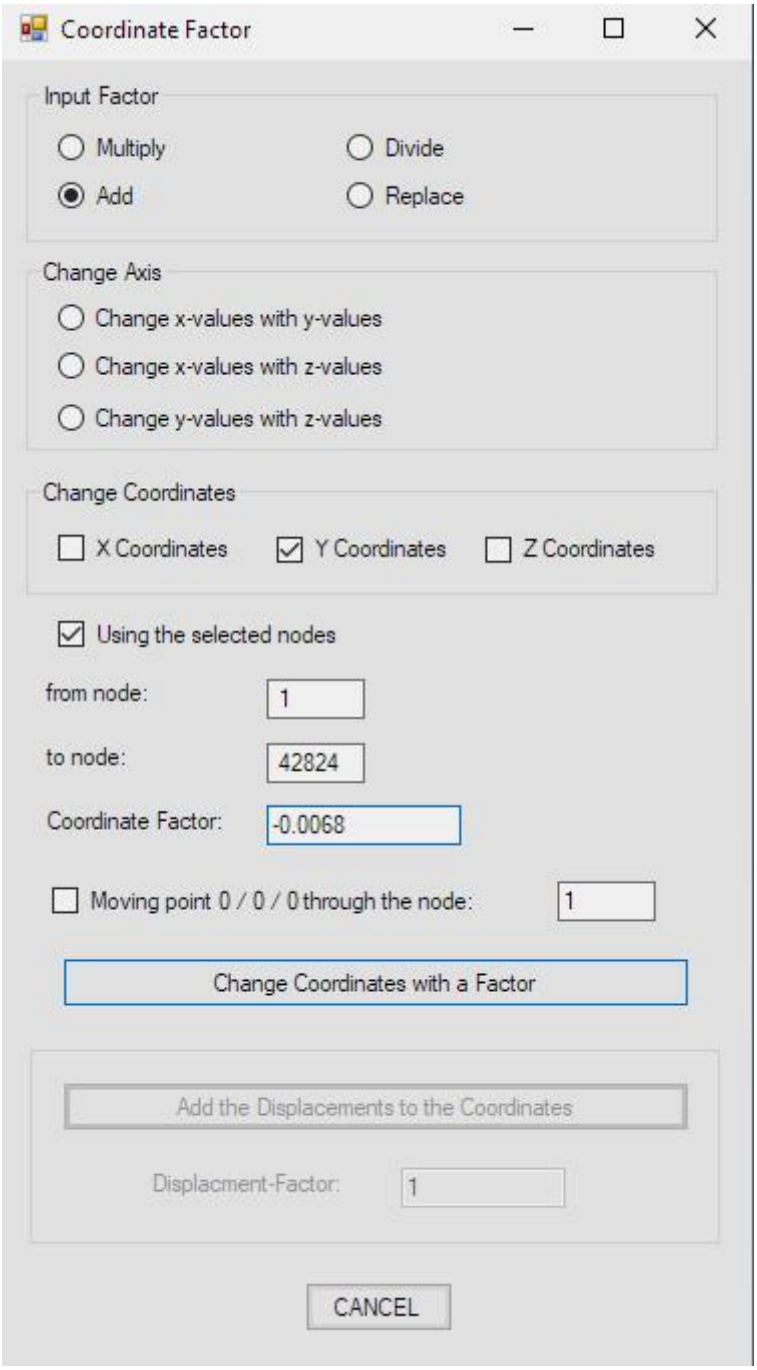
Edit the Contact-Distance

But it is just as important that you know how to do it to reduce the distance. If the contact distance is a little bit too much the Contact-Solver does not convergence and canceled the calculation.



At first you must show all Nodes of Element Group 2 with the View and “Node-Modus” tab. Then choose menu “Coordinte Factor” to add a Coordinate-Faktor of -0.0068mm.



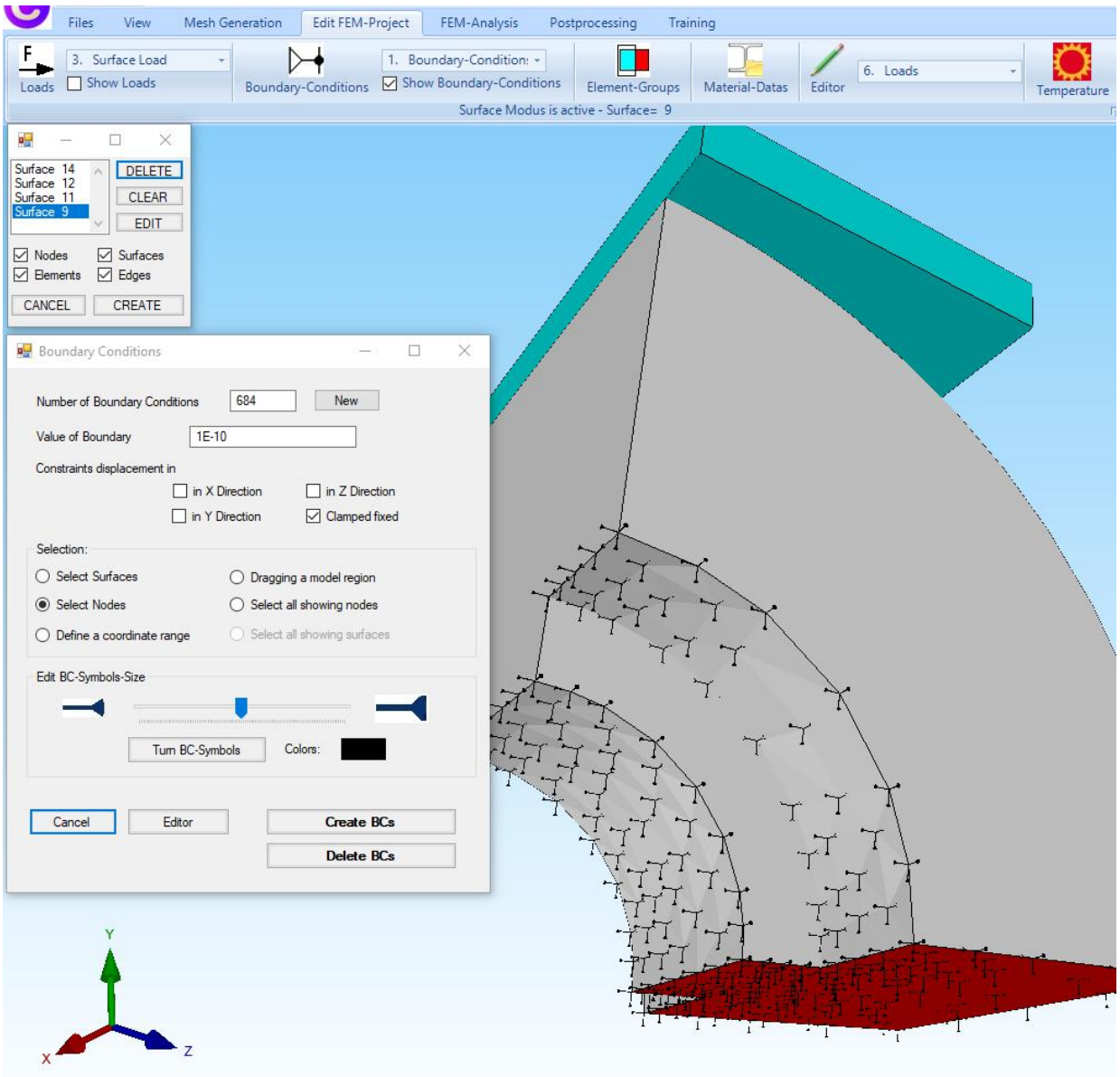


Now the Contact-Distance is 0.000mm.

Create Boundary Conditions

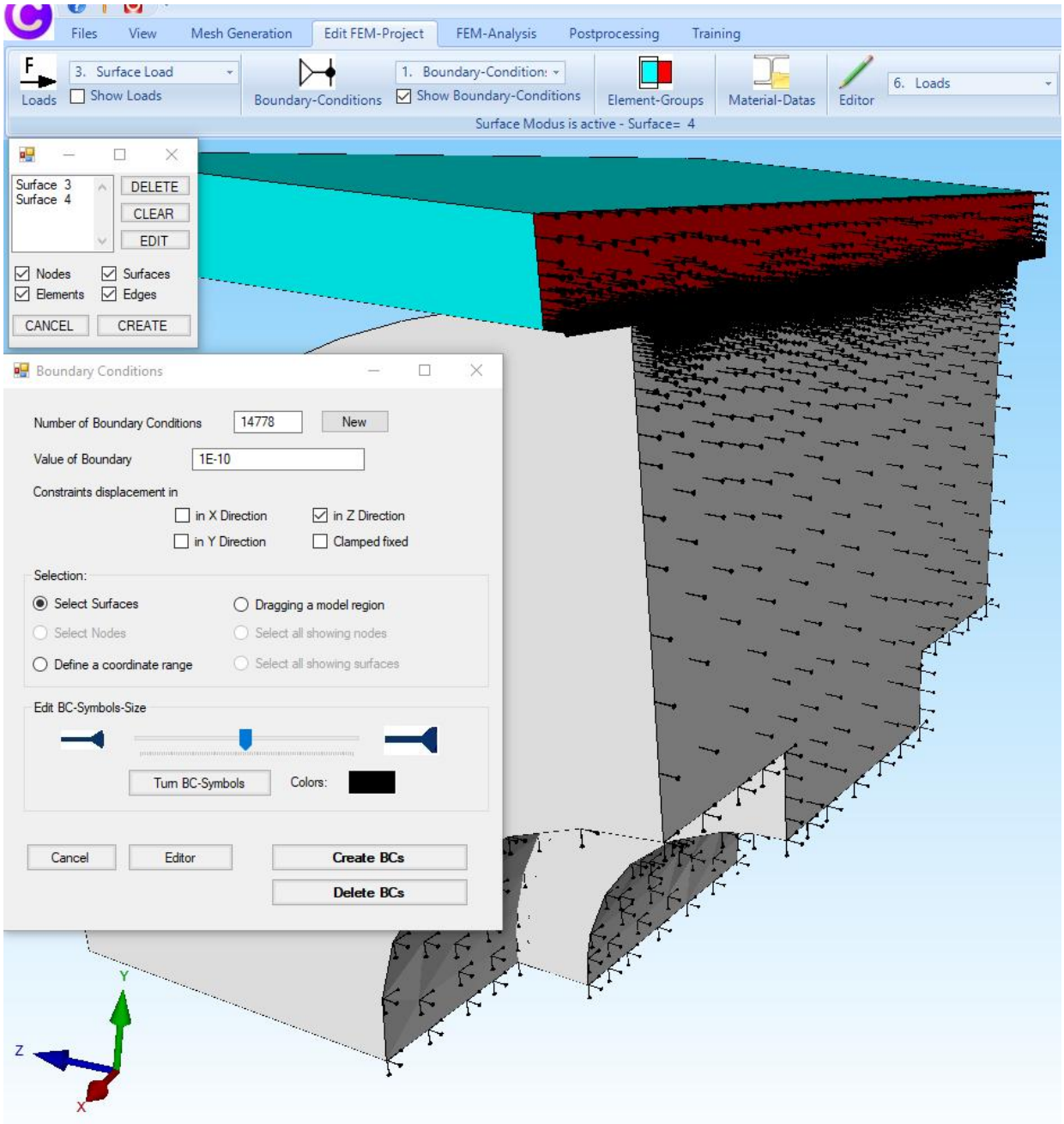
Clamped fixed

The inner diameter of the wheel is fixed, it would be bonded to ball bearings. Select the tab „Edit FEM-Project“ and „Boundary-Conditions“ and create the fixed BCs on Surface 9, 11, 12 and 14 with „Create“ in the selectbox.



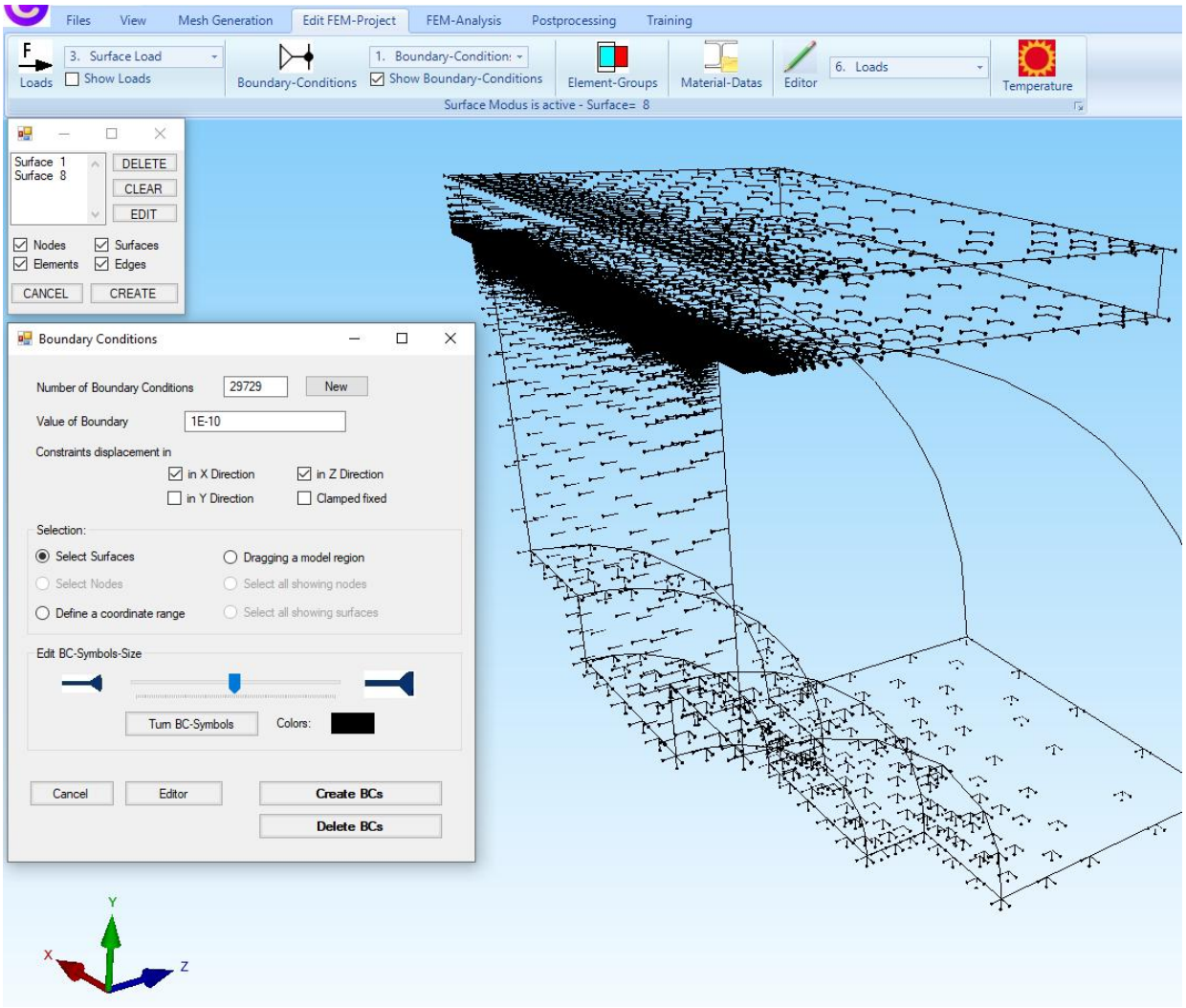
Symmetric-Boundary-Conditions

The nodes of Surface 3 and 4 must be constrained in z direction in order to take advantage of the symmetry of the quartered model.



Road clamped

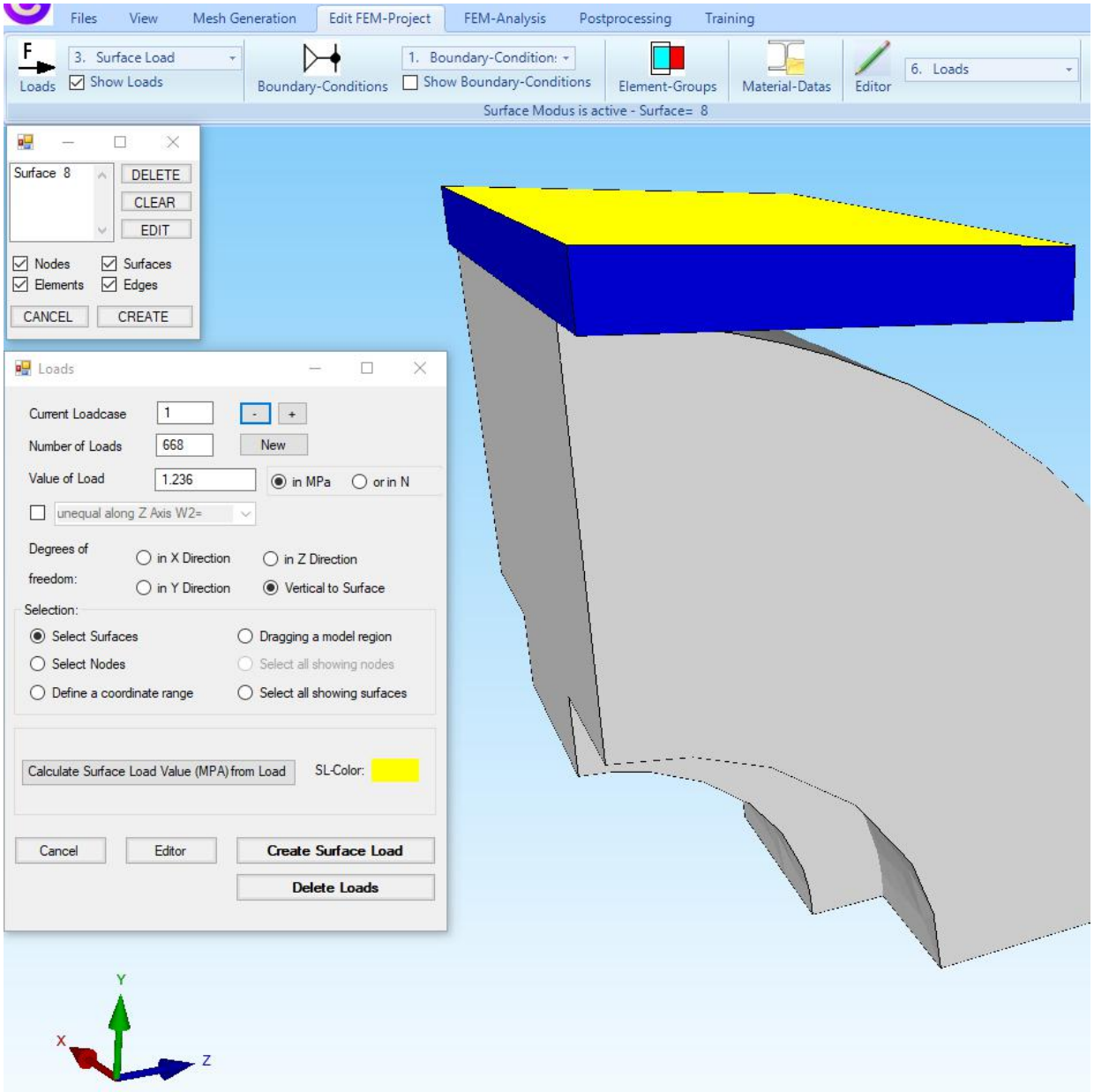
The road is only moved in y-direction so we must constrained the displacements in the x- and z-direction at the top and bottom side.



Create the Surface Load

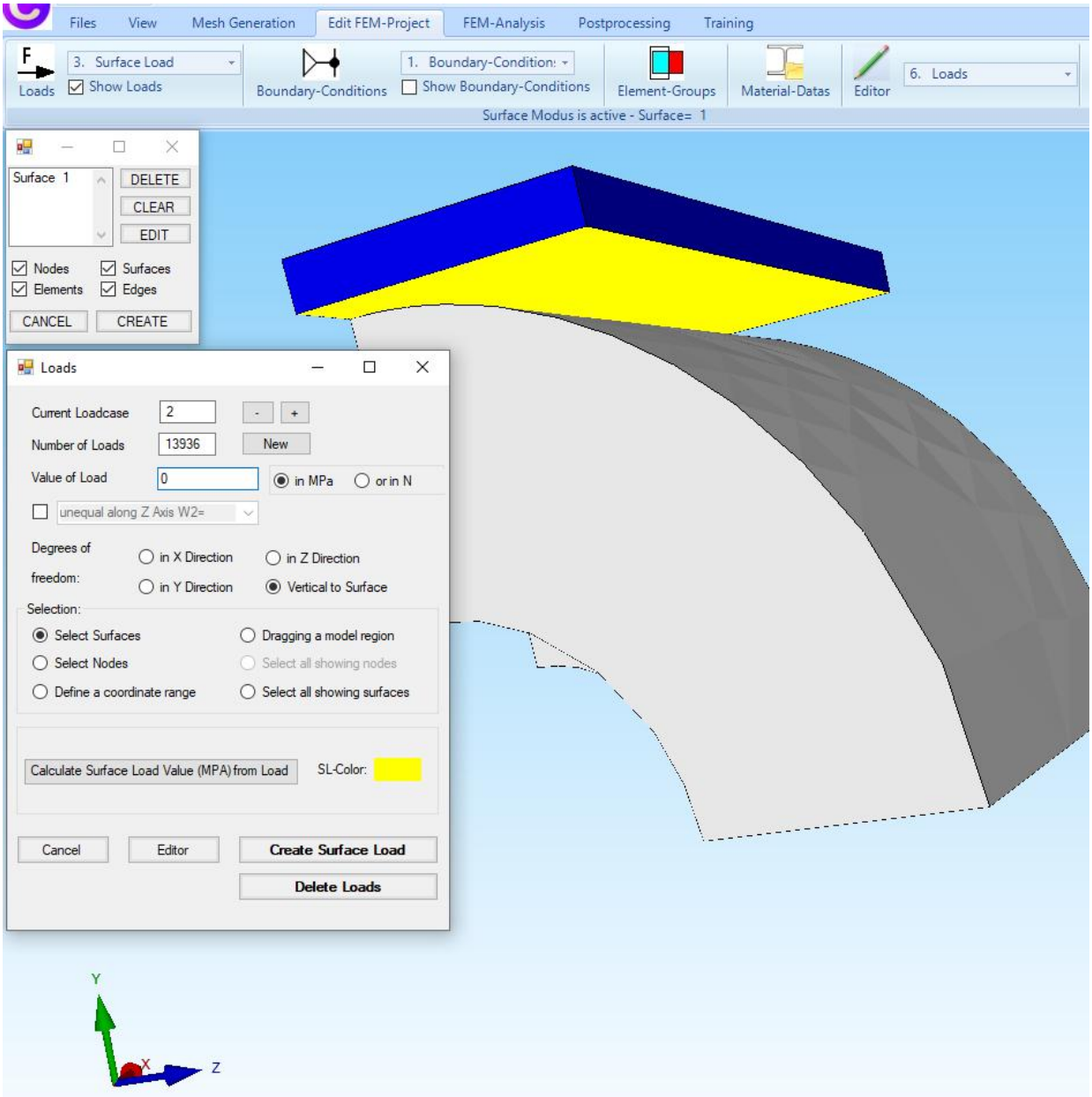
The roller is loaded with 1.236 Mpa at the top side of the road in y direction.

Select the tab „Edit FEM-Project“ and „Surface Load“ to create Load Case 1 with a Surface Load of 1.236 Mpa and with a mouseclick on Surface 8 and „Create“ in the selectbox.



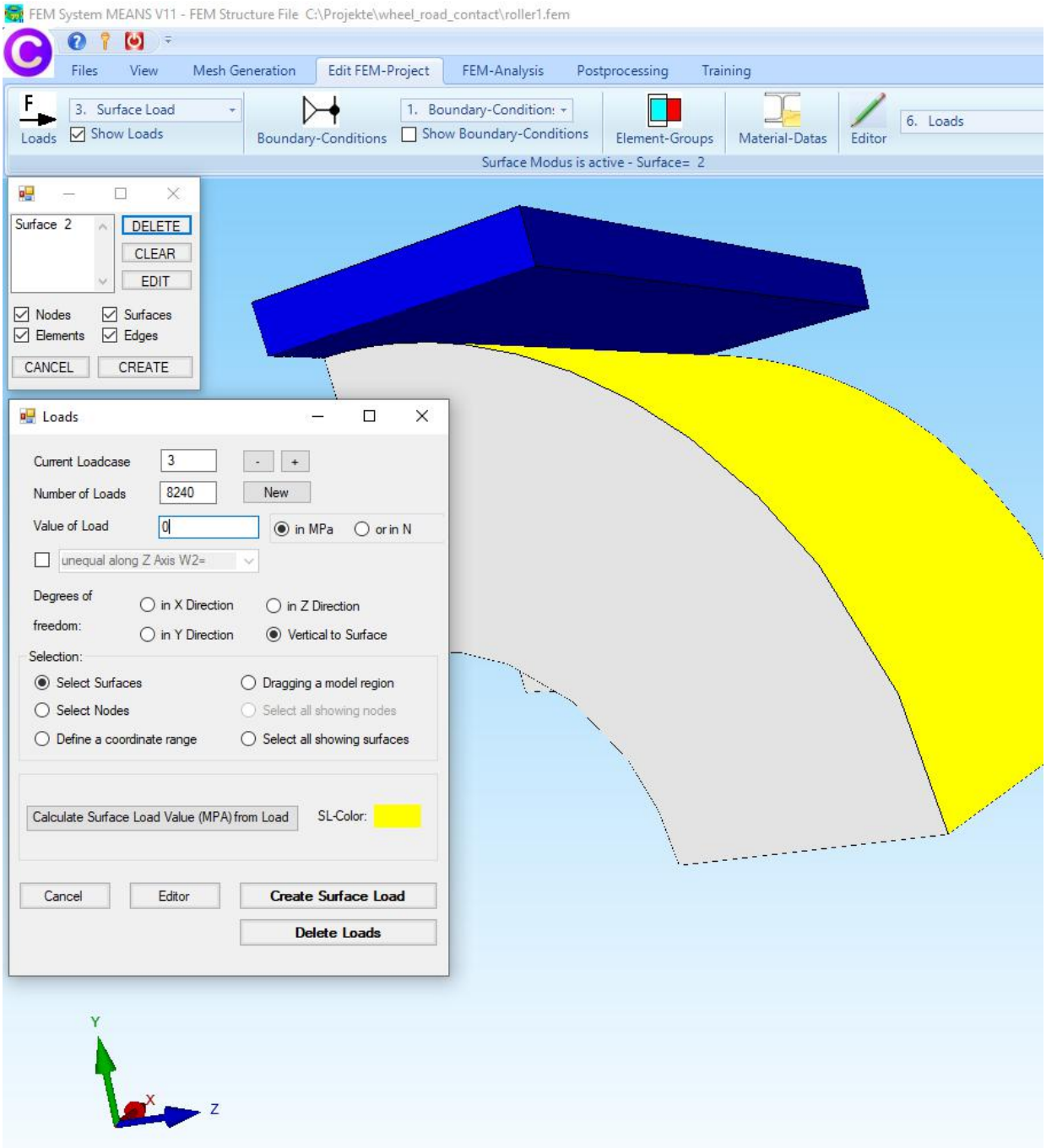
Create the Master-Contact-Surface

Now we must create the Master-Contact-Surface with Load Case 2.
Select the tab „Edit FEM-Project“ and „Surface Load“ to create Load Case 2 with a Surface Load and with a mouseclick on Surface 8 and „Create“ in the selectbox. A value is not necessary.



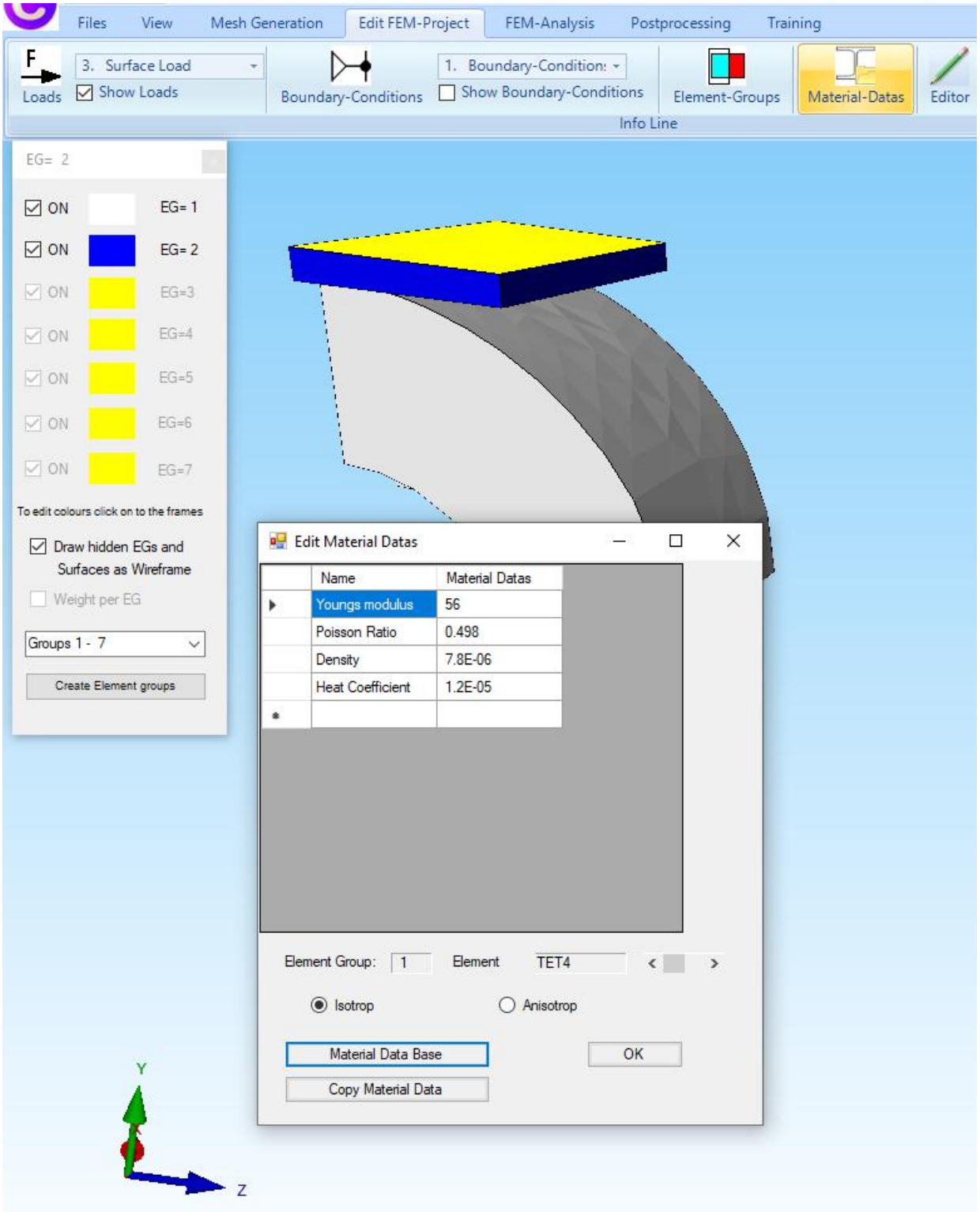
Create the Slave-Contact-Surface

Now we must create the Slave-Contact-Surface with Load Case 3.
Select the tab „Edit FEM-Project“ and „Surface Load“ to create Load Case 3 with a Surface Load and with a mouseclick on Surface 2 and „Create“ in the selectbox. A value is not necessary.



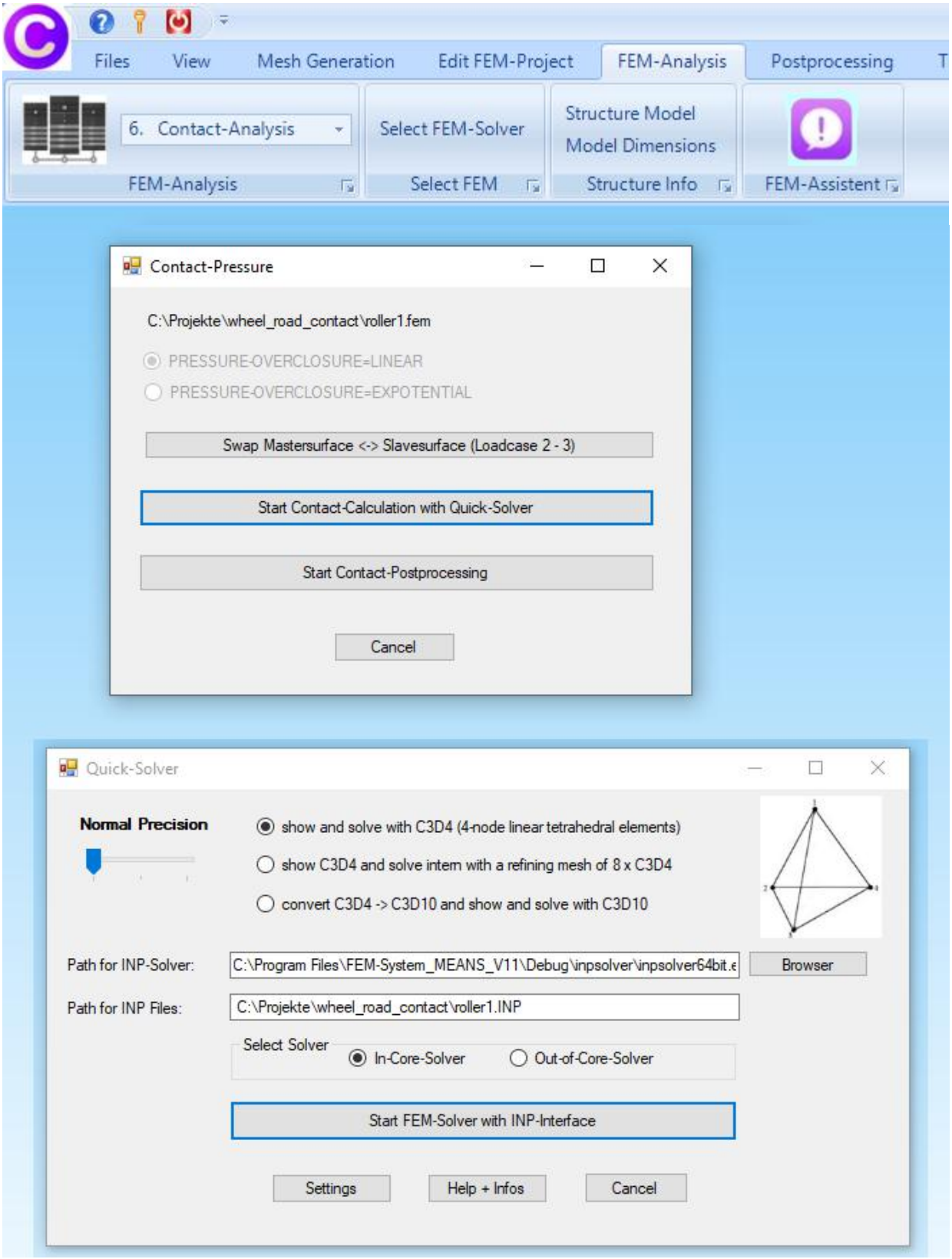
Material Datas

Now we must input the Material Datas with the tab „Edit FEM-Projects“ and „Material-Datas“. Input for EG 1 with the Polyurethane Wheel the Youngs modulus of 56MPa and Poisson Ratio of 0.498 and for EG 2 with the Steel-Road the Youngs modulus of 210000MPa and Poisson Ratio of 0.3.



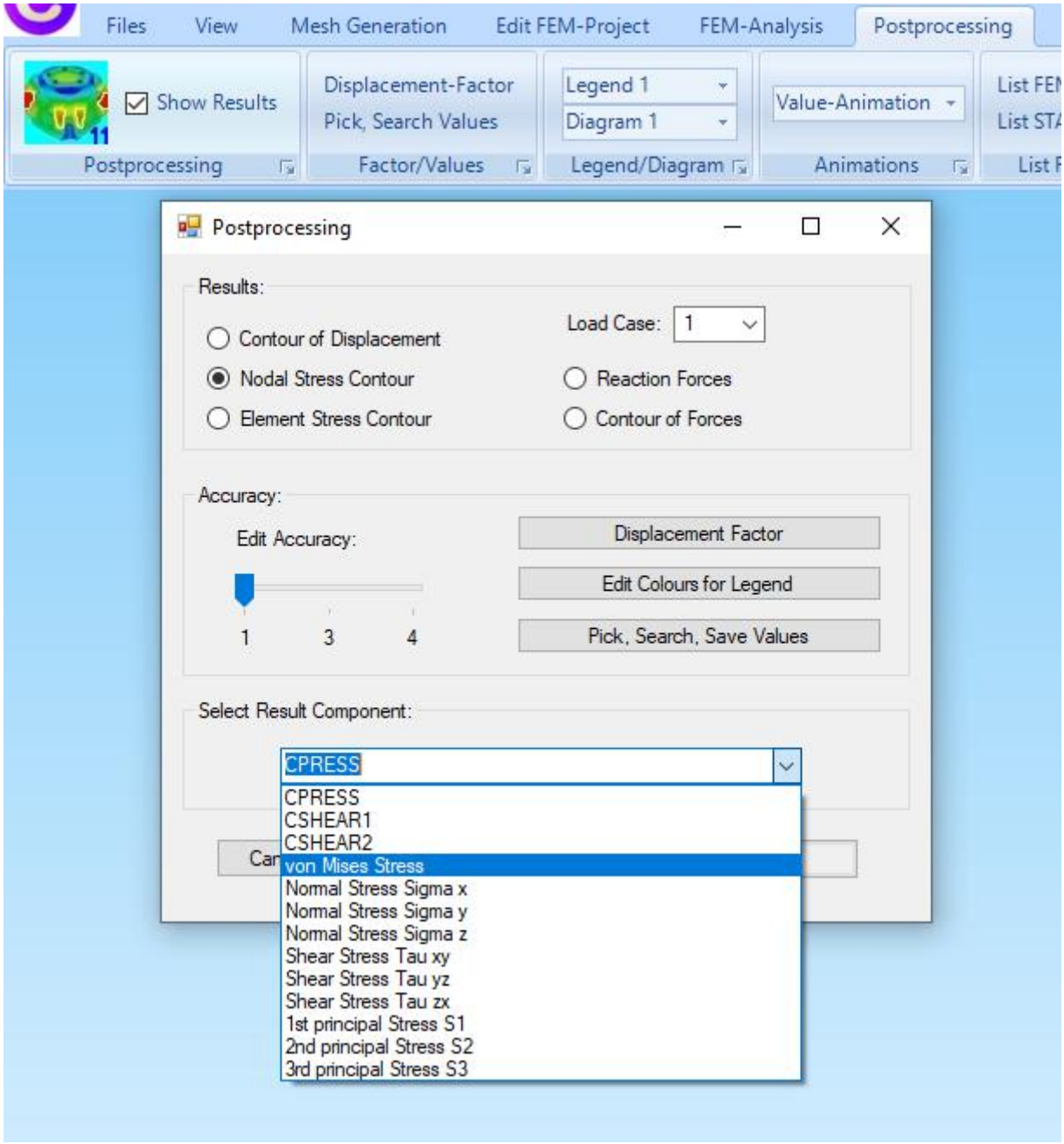
FEM-Contact-Analysis

The contact model is ready for the FEM-Analysis. Select the tab „FEM-Analysis“ and „Contact-Analysis“ and start the calculation to solve the displacements and stresses with the TET4 tetrahedral elements in a few minutes.



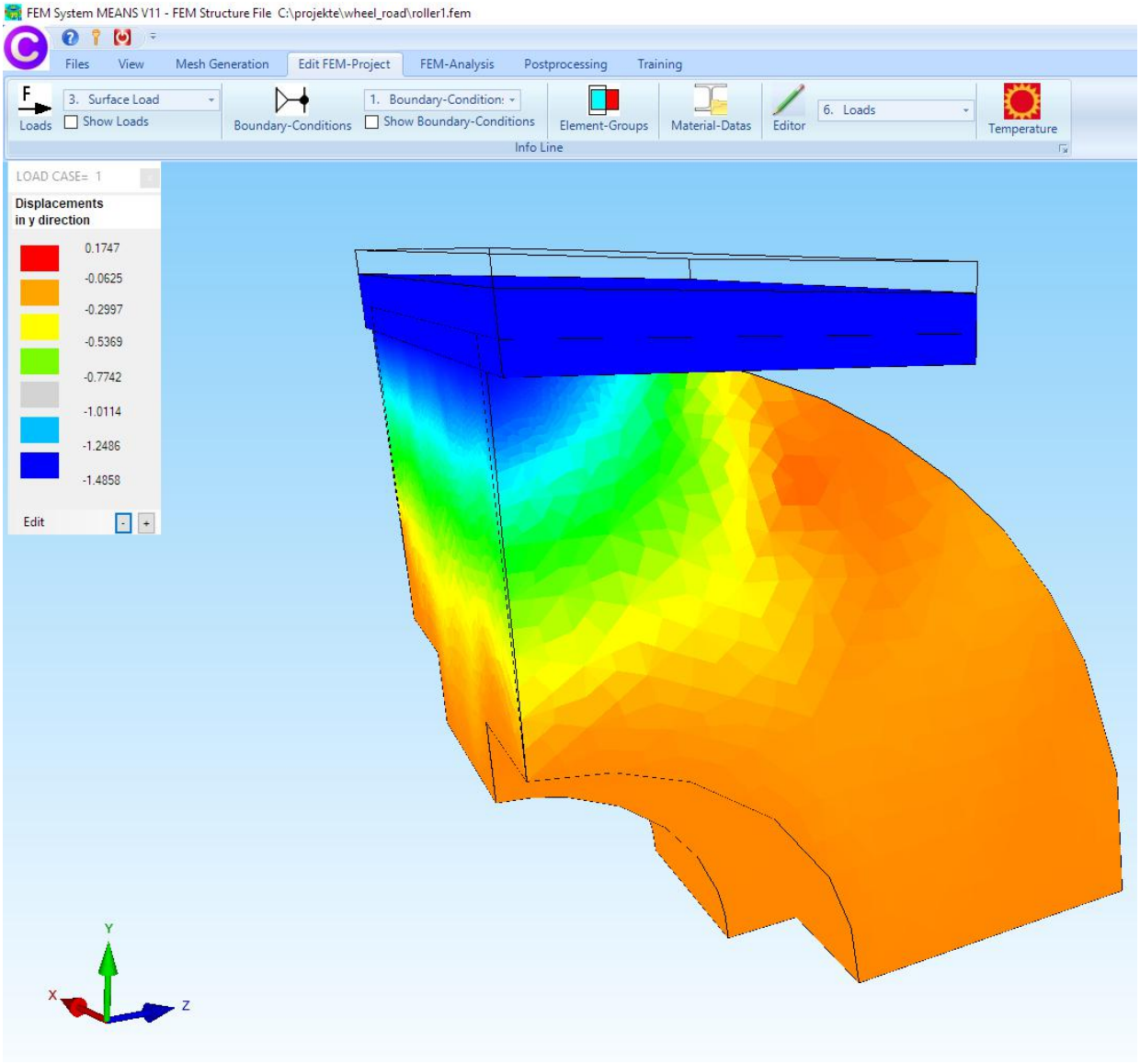
Postprocessing

After the calculation the results can be evulated. Select the tab „Postprocessing“ and choose the „Contour of Displacement“ or „Nodal Stress Contour“.



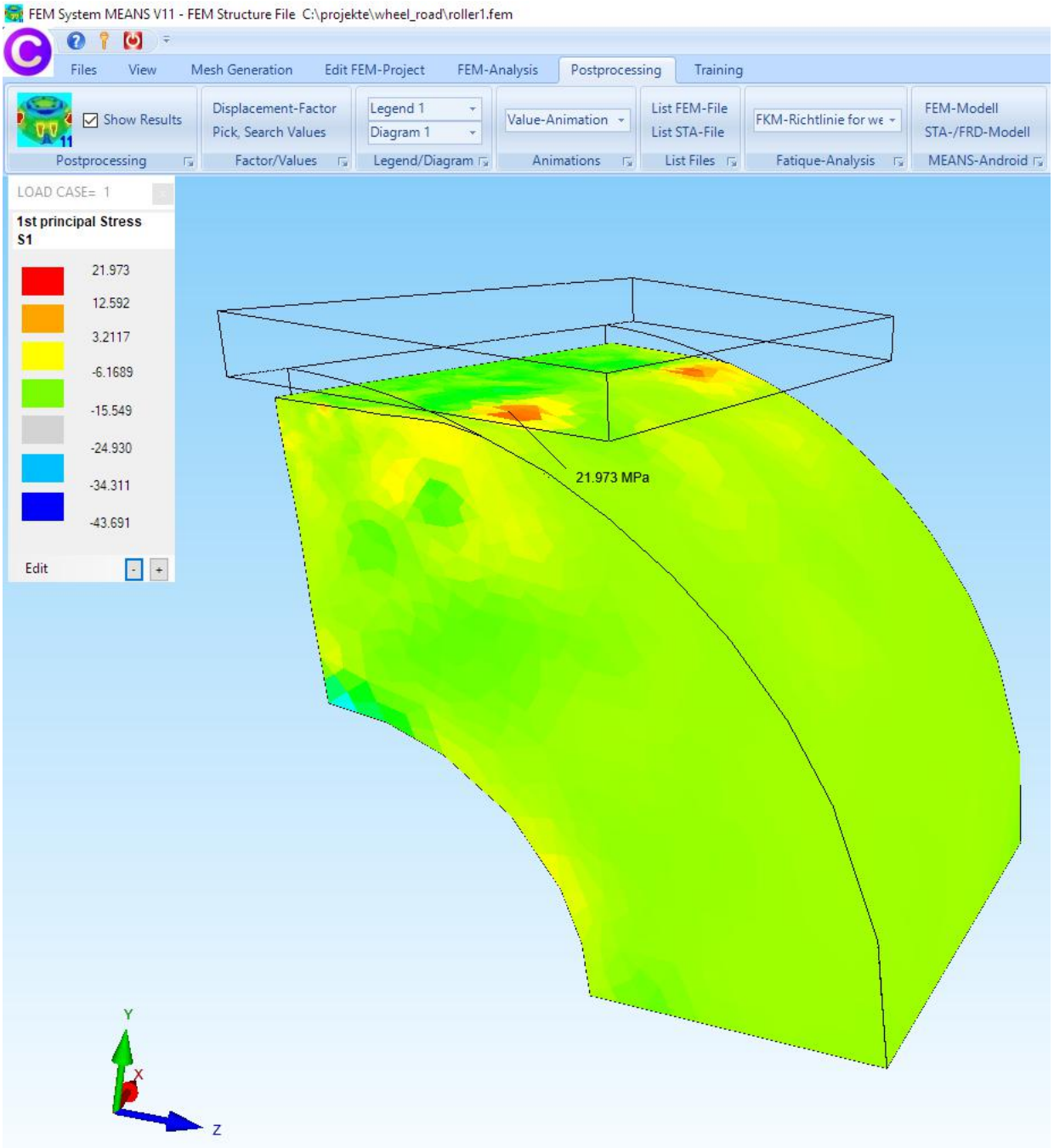
Displacements

Max. Displacement in y direction is -1.458 mm



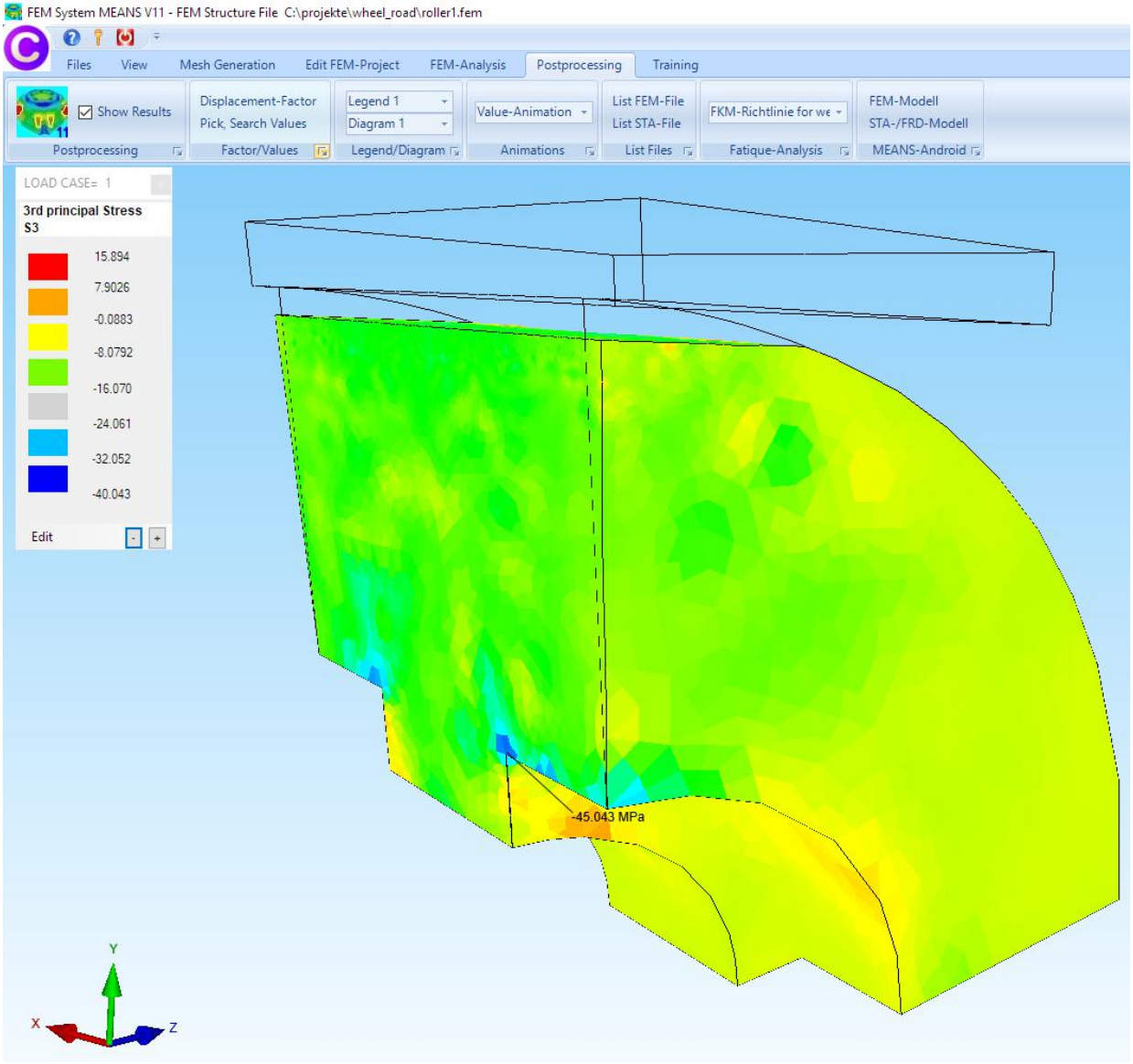
1st principal Stresses S1 of the Wheel

Maximale 1st principal Stresses S1 of the Wheel is 21.9 MPa



3rd principal Stress S3 of the Wheel

Minimale 3rd principal Stresses S3 of the Wheel is -40.04MPa



v.Mises Stresses of the Road

Maximale v.Mises Stress of the Road is 98.6 MPa