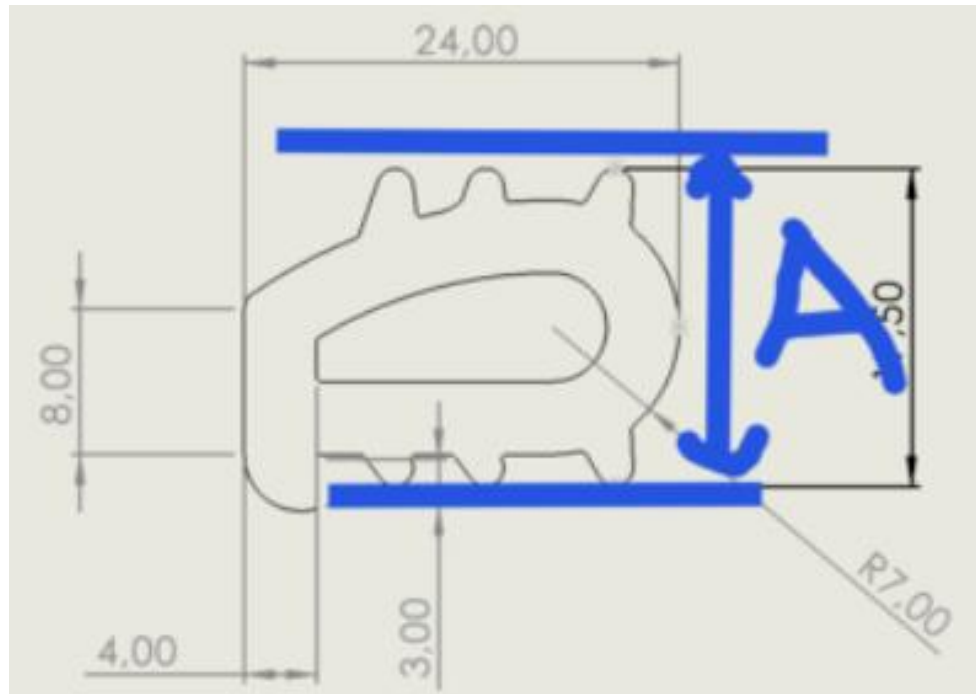


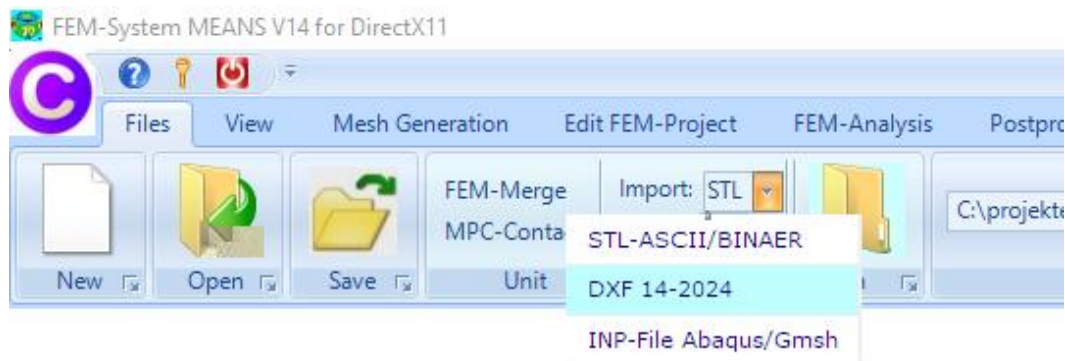
Part 41 - Nonlinear Contact-FEM-Analysis of a Rubber Seal with FEM-System MEANS V14

1.) Import Cross-Section DXF-File

First import the Cross-Section DXF-File with Register „Files“ and menu „Import“ in MEANS V14 to create a line model of 558 nodes and 552 BEAM2 elements.

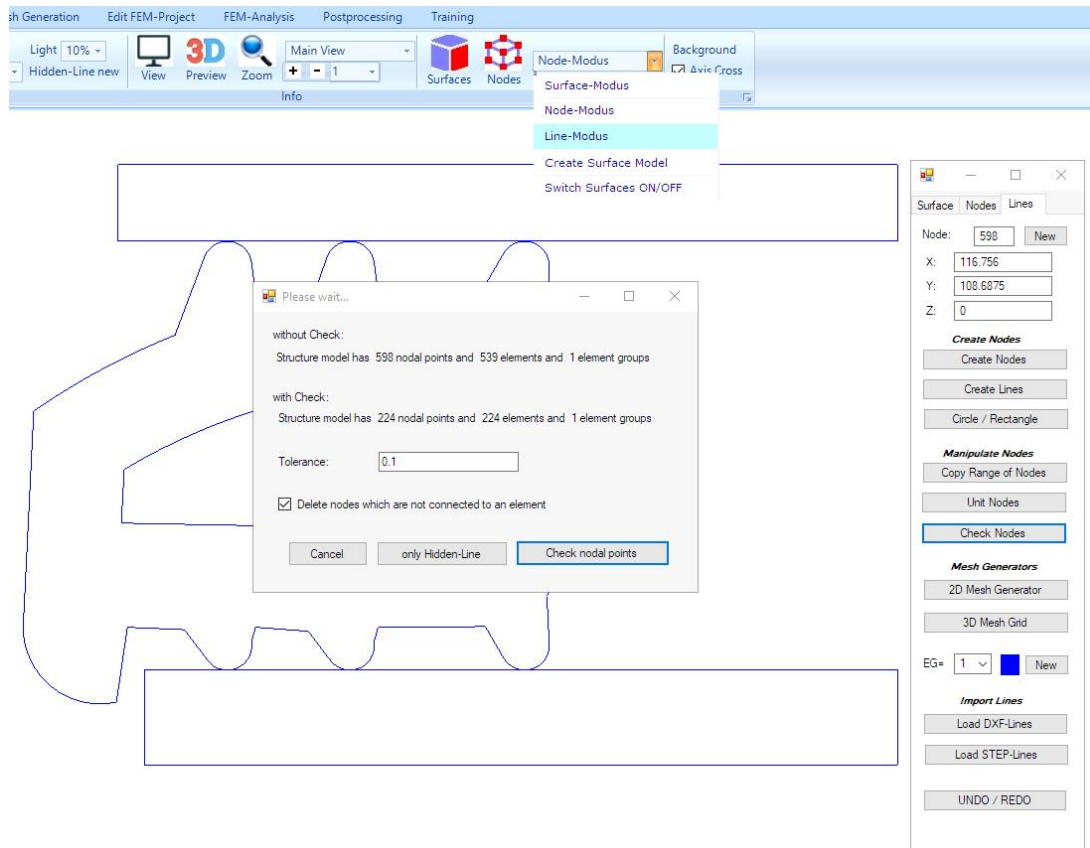


If you have only a DWG file, it must first be converted to a DXF file using any 2D-CAD system.



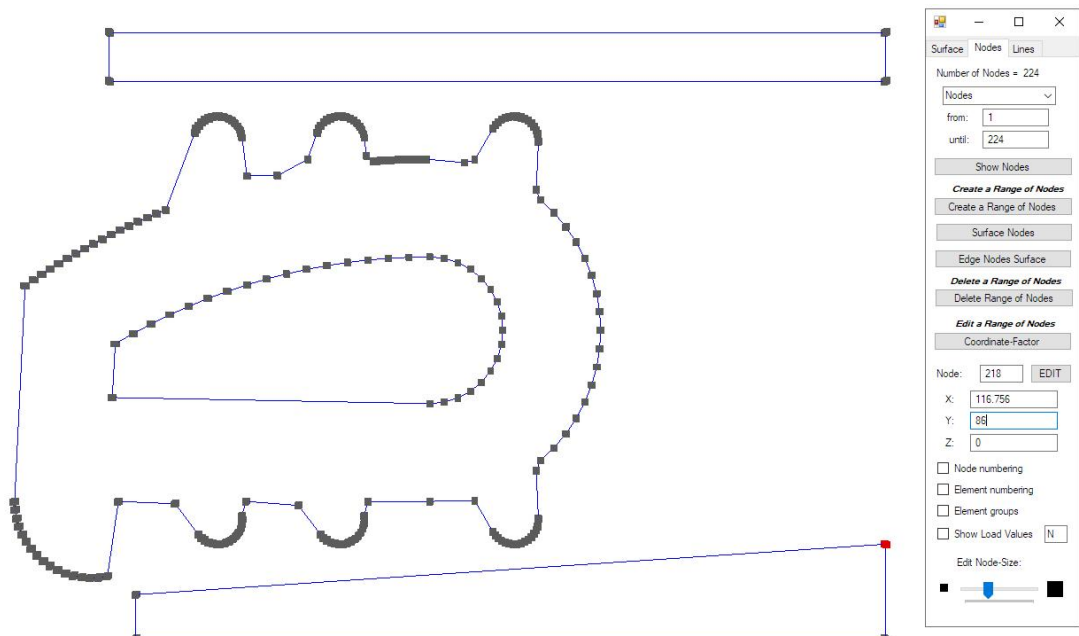
2.) Model-Check

Select Register „View“ and menu „Line-Modus“ and using the right side menu „Check Nodes“ to delete the overlaid nodal points from 598 to 224 and the lines from 539 to 224 with a Snap Radius of 0.1.



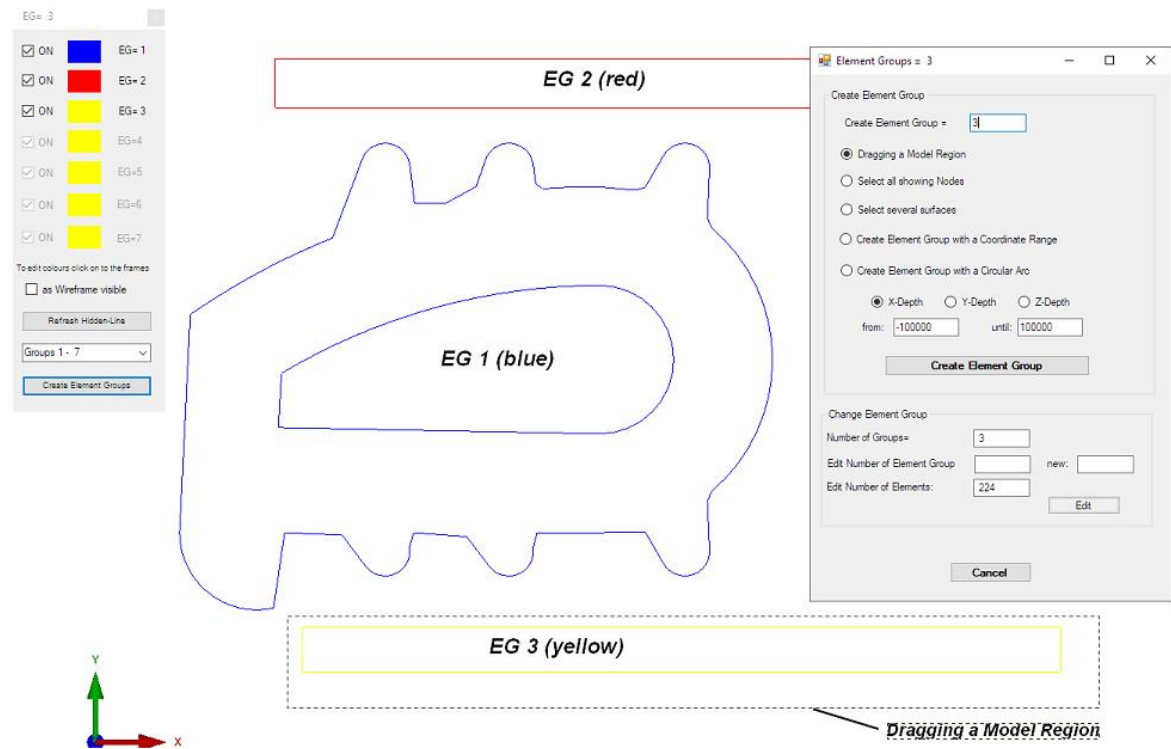
3.) Reduce Plates

Switch to Node-Modus and enlarge the nodes size with „Edit Node-Size“ and reduce the high with menu „Edit“ on the top nodes 221 and 222 from 109 to 107 mm and the bottom nodes 217 and 218 from 88 to 86 mm.



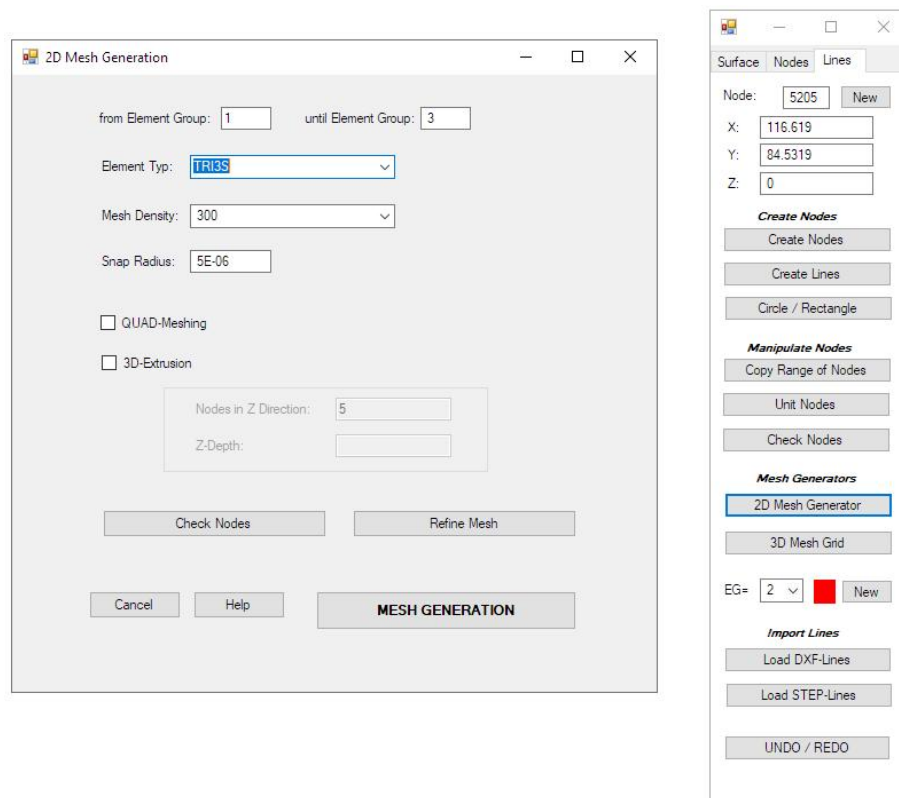
4.) Create 3 Element Groups

Select Register „Edit FEM-Project“ and menu „Element Groups“ to create with the Selection „Dragging a Model Region“ the EG 2 for the Top and EG 3 for the Bottom Plate. Then the Line Model is saved with a name.

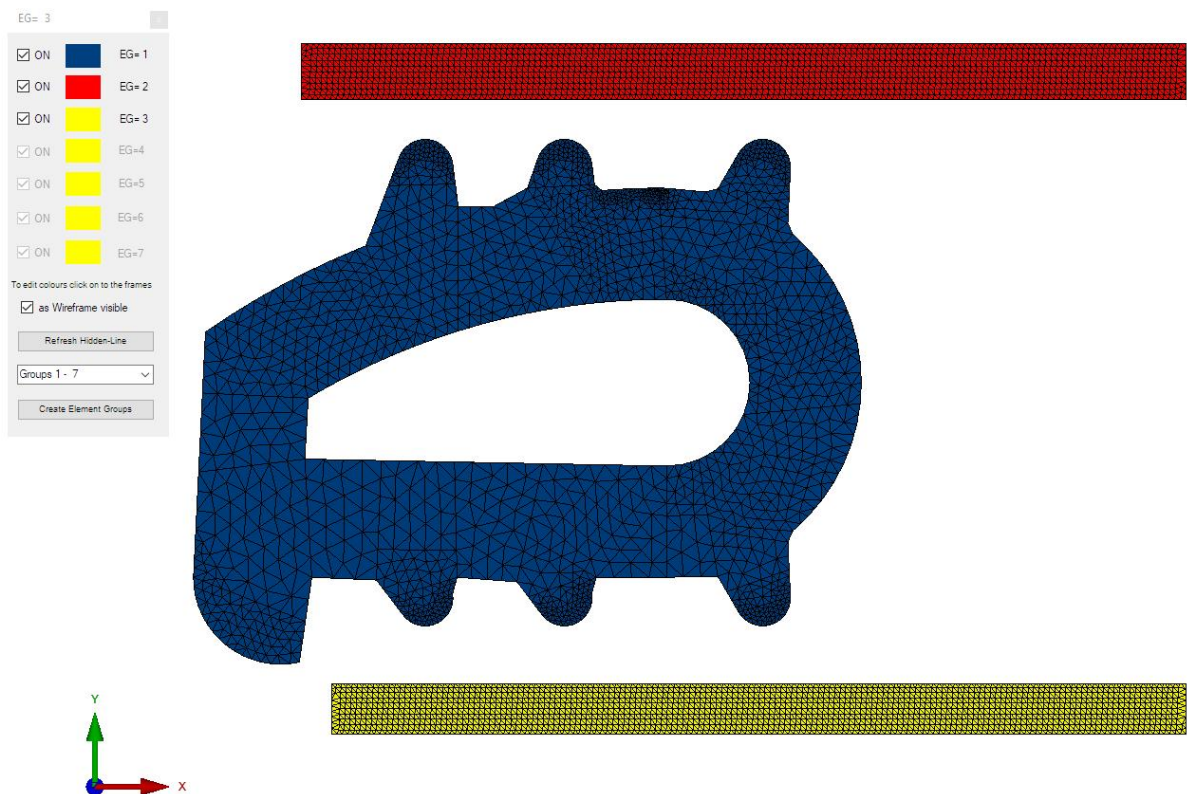


5.) Generate a 2D-Mesh

Switch to the „Line Modus“ and choose menu „2D Mesh Generator“

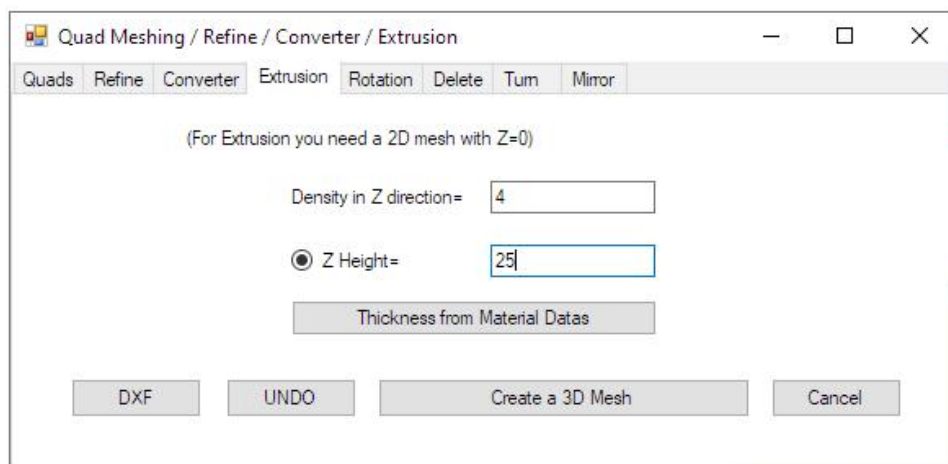
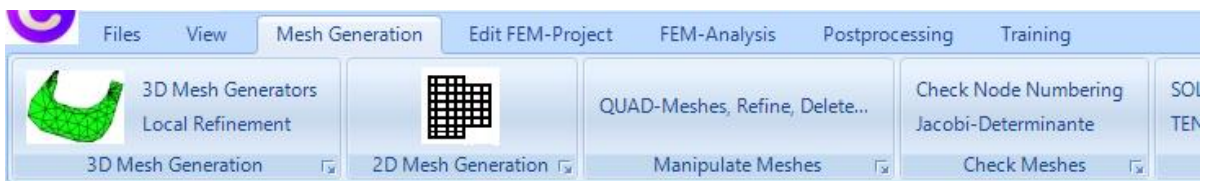


to generate a 2D-Mesh with 9 354 TRI3S elements and 5 205 Nodes.

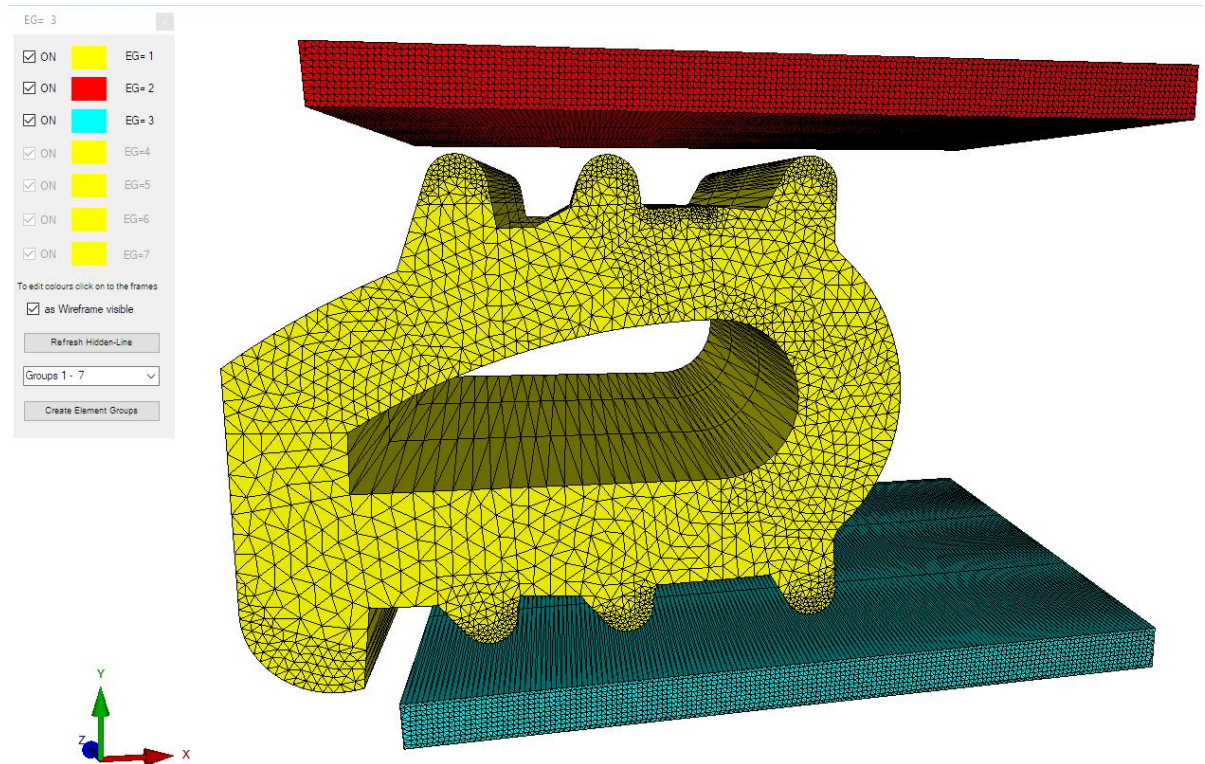


6.) Extrude a 3D-Mesh

Select Register „Mesh Generation“ and menu „QUAD-Meshes, Refine, Delete..“ and switch to Register „Extrusion“ to extrude the 2D mesh with a Density in Z direction of 4 and with a Z-Heigh of 25.

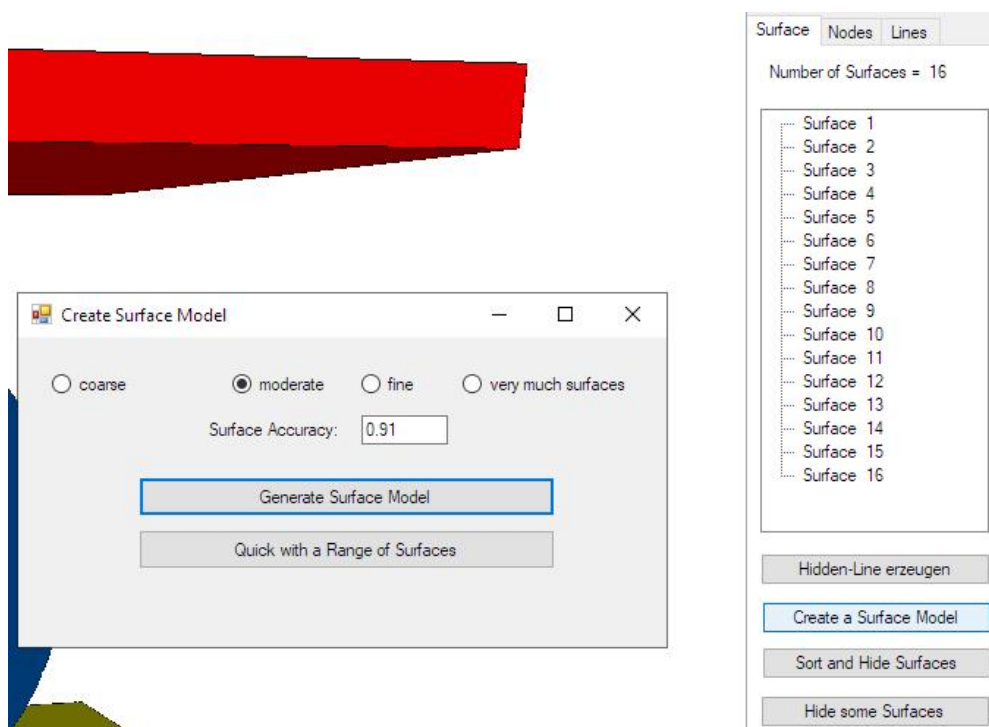


The Mesh consists of 28 062 PEN6 elements, 20 820 nodes and 3 element groups.

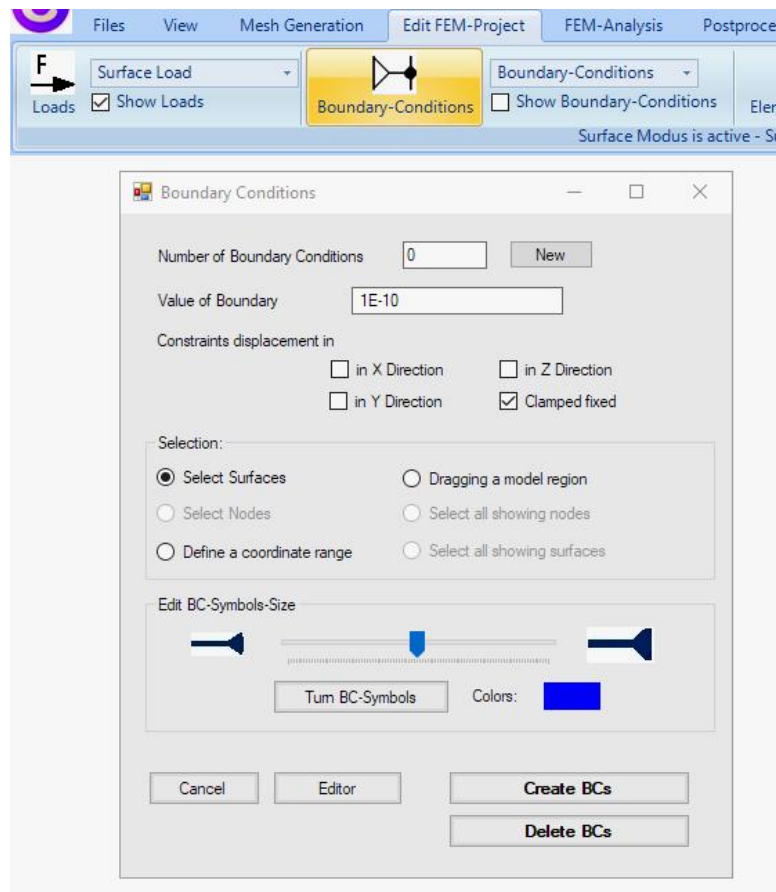


7.) Clamped fixed the Bottom-Plate

Switch to the „Surface-Modus“ and select menu „Create a Surface Model“ to create a moderate Surface Model to select the Boundary Conditions on the bottom plate.

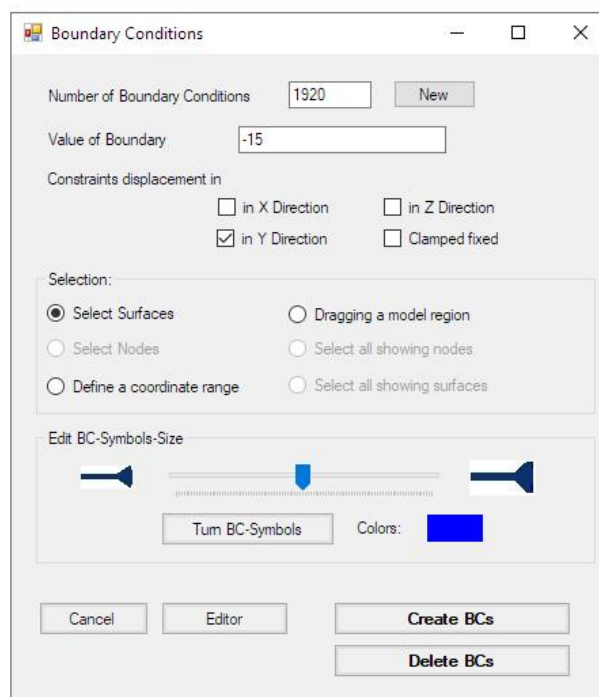


Select Register „Edit FEM-Project“ and „Boundary-Conditions“ and clamped fixed with the Selection „Select Surfaces“ by clicking on Surface 24 on the Bottom-Plate.



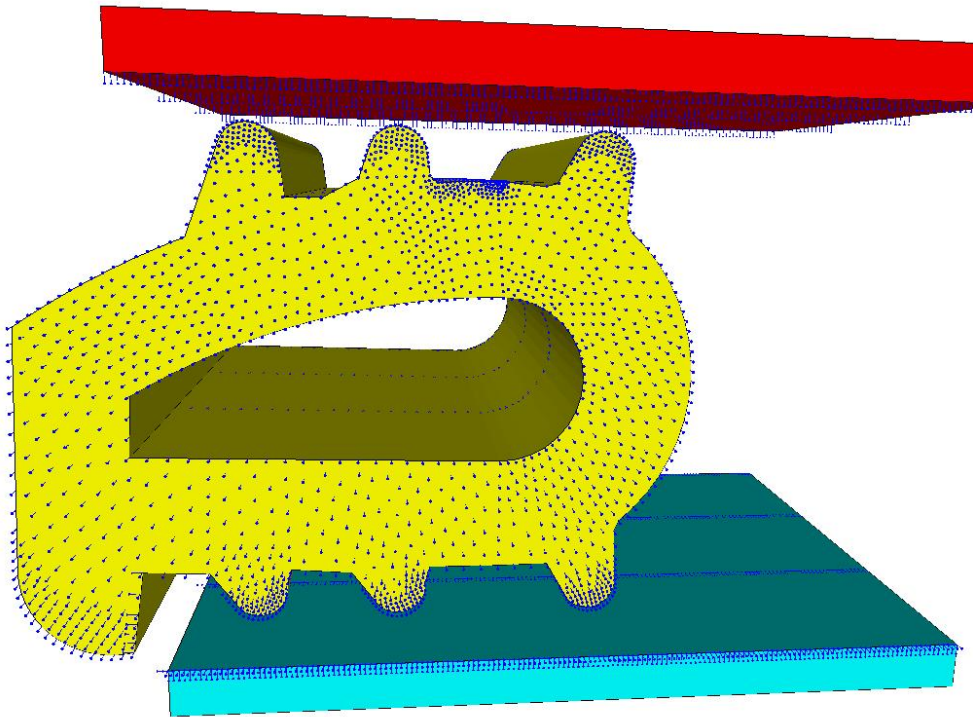
8.) Prescribed Displacements on Top-Plate

Select the „Boundary Condition“ and click on Surface 31 to create the prescribed displacement of -15 mm in y direction on the Top-Plate-



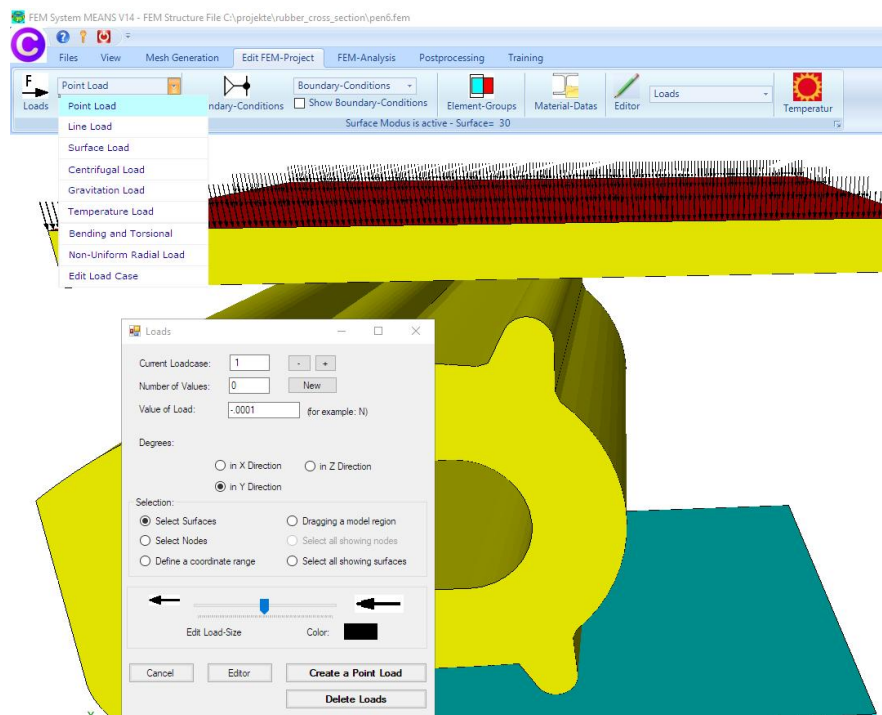
9.) Clamped fixed in z direction all large surfaces

Select „Boundary Conditions“ and clamped fixed in z direction on all large Surfaces. Now the FEM-Model consists over 7450 Boundary Conditions:



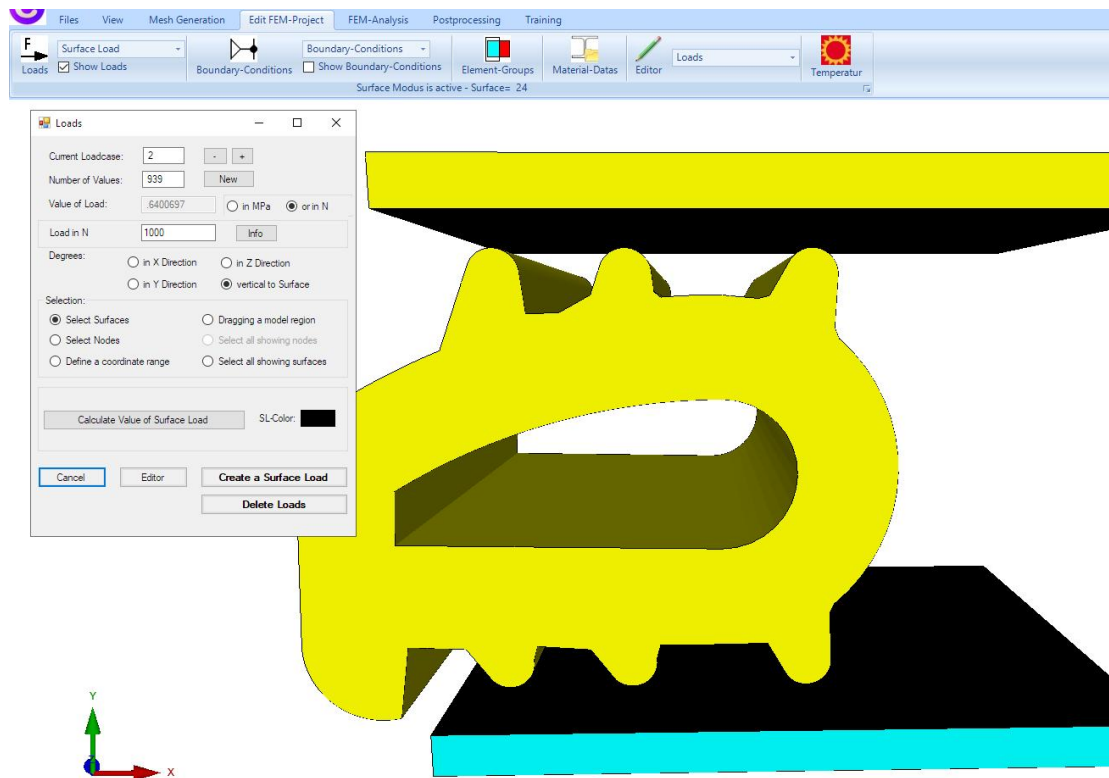
10.) Loadcase 1 with a dummy Point Load

Select Register „Edit FEM-Project“ and „Point Load“ and create Loadcase 1 with a dummy point load of 0.001 N on Surface 30 of the Top-Plate.

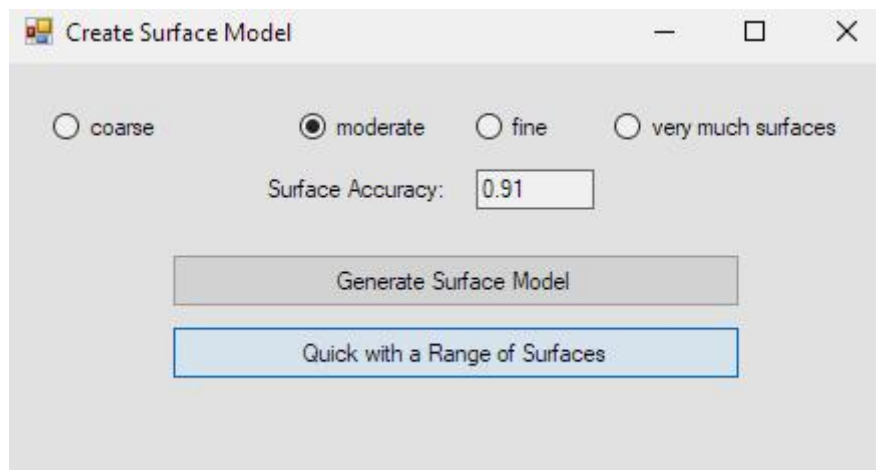


11.) Loadcase 2 with the Master Contact Surfaces

Create Loadcase 2 with a Surface Load and click on the Surface 24 and 31.
A load value is not needed.



Next split the Rubber-Borehole in two Surfaces and select the top one as Master Surface. To that select „Quick with a Range of Surfaces“ and dragging a model region over the borehole.

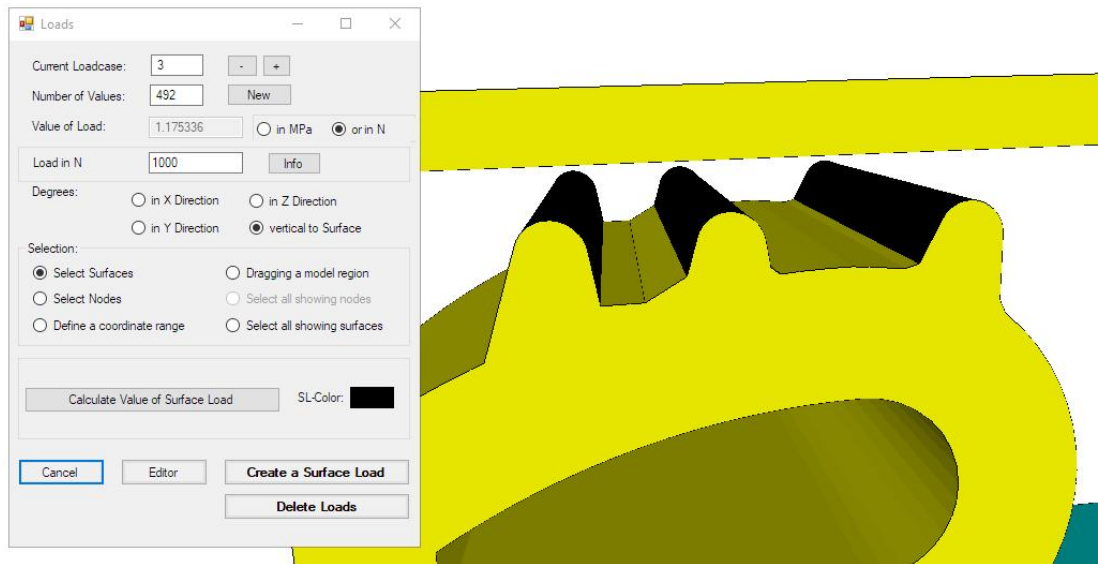




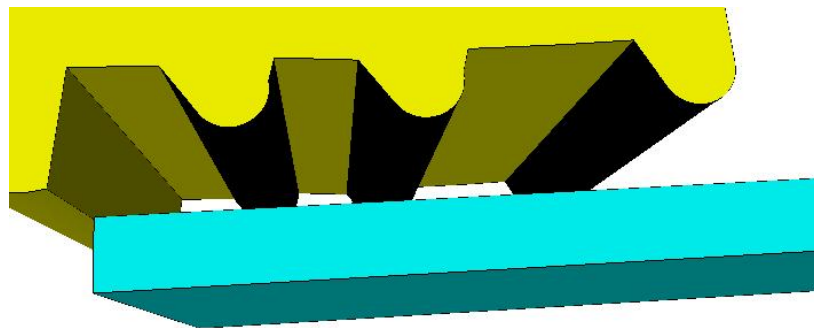
These were the three large Master-Surfaces, now smaller Surfaces need to be created as Master Surfaces, as shown in the Tutorial-Video.

12.) Loadcase 3 with the Slave Contact-Surfaces

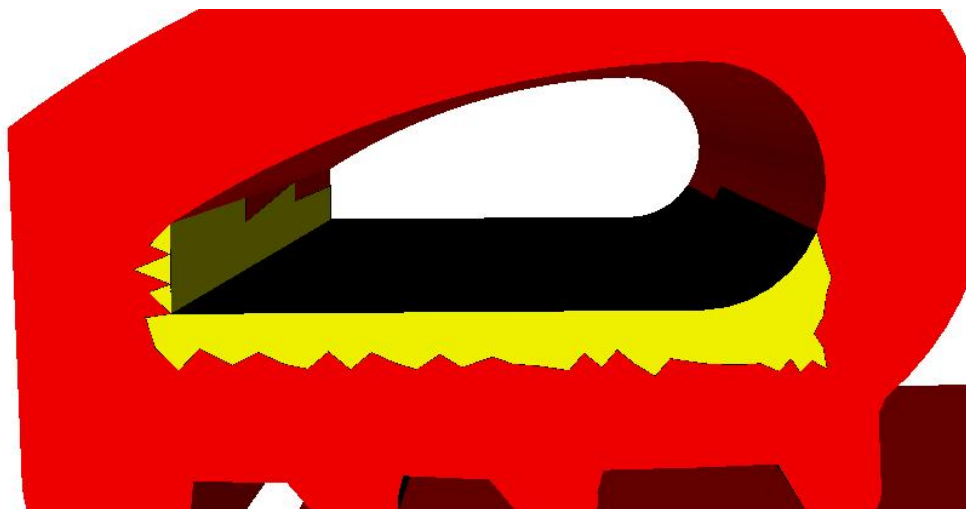
Create Loadcase 3 with a Surface Load and click on the Rubber-Surfaces 7, 10, 12



and below on the Surfaces 2, 3 and 5.



Also create the below Borehole-Surface with „Quick with a Range of Surfaces“.



These were the large Slave-Surfaces, now smaller Surfaces need to be created as Slave Surfaces, as shown in the Tutorial-Video.

13.) Material Datas

Select Register „Edit FEM-Projecs“ and „Material Datas“ and input E-Modulus and the Poisson Ratio for Rubber and Steel.

EG1 - Rubber

The screenshot shows the 'Edit Material Datas' dialog box for Rubber material. The table below contains the material data:

Name	Material Datas
Youngs modulus	4.5
Poisson Ratio	.45
Density	1.11E-05
Heat Coefficient	0
*	

Below the table, the 'Element Group' is set to 1 and the 'Element' is PEG6. The 'Isotrop' radio button is selected, and the 'Anisotrop' radio button is unselected. The 'Material Data Base' and 'Copy Material Data' buttons are visible, along with an 'OK' button.

EG2 and EG3 - Steel

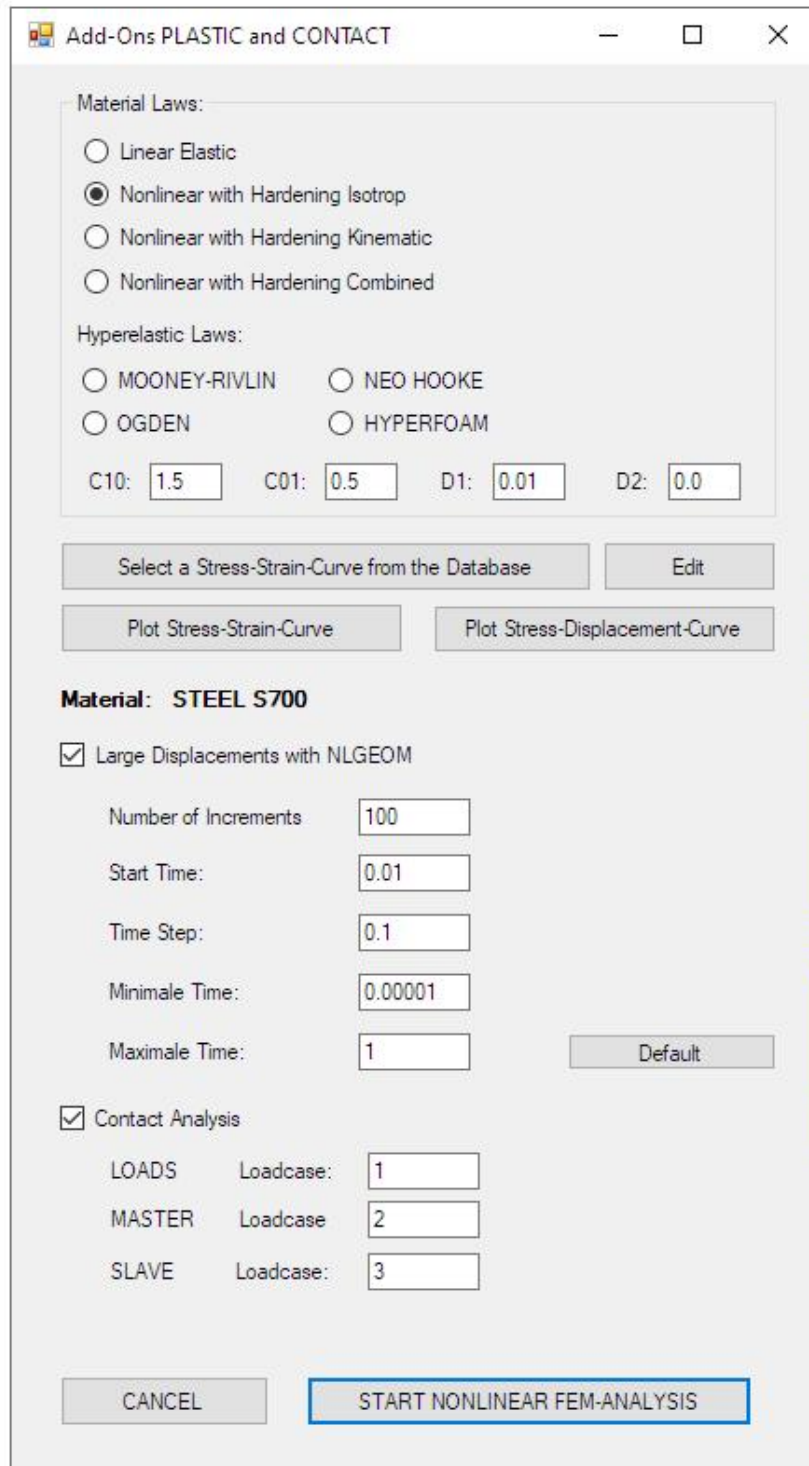
The screenshot shows the 'Edit Material Datas' dialog box for Steel material. The table below contains the material data:

Name	Material Datas
Youngs modulus	210000
Poisson Ratio	.3
Density	7.8E-06
Heat Coefficient	1.2E-05
*	

Below the table, the 'Element Group' is set to 3 and the 'Element' is PEG6. The 'Isotrop' radio button is selected, and the 'Anisotrop' radio button is unselected. The 'Material Data Base' and 'Copy Material Data' buttons are visible, along with an 'OK' button.

14.) Nonlinear FEM-Analysis

Select „FEM-Analysis“ and „Material Nonlinear“ to start an nonlinear Contact-Analysis.



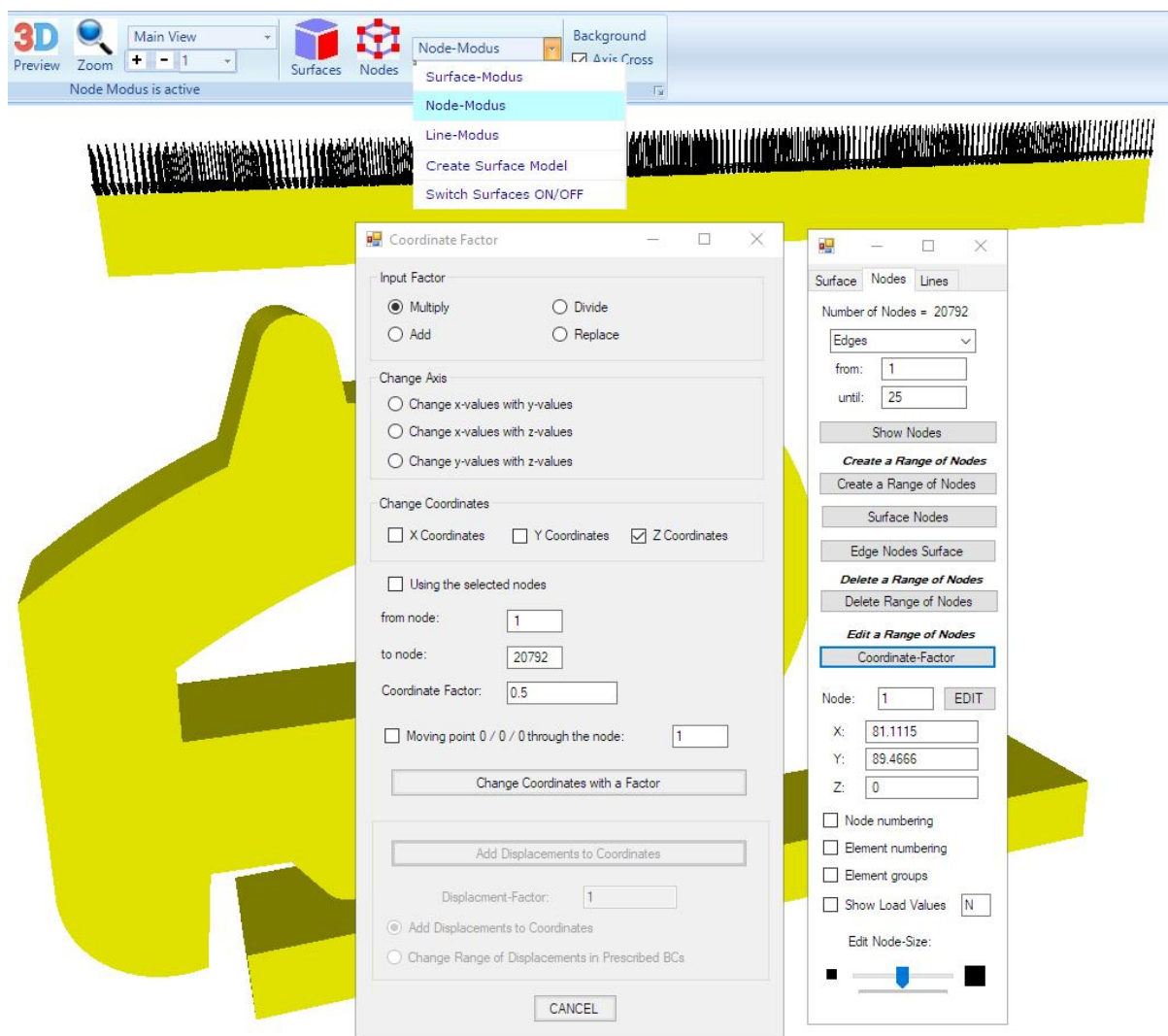
15.) Error Message

The nonlinear solver terminates the calculation with an error message.

```
C:\Program Files\FEM-System_MEANS_V14\Debug\cmd.exe

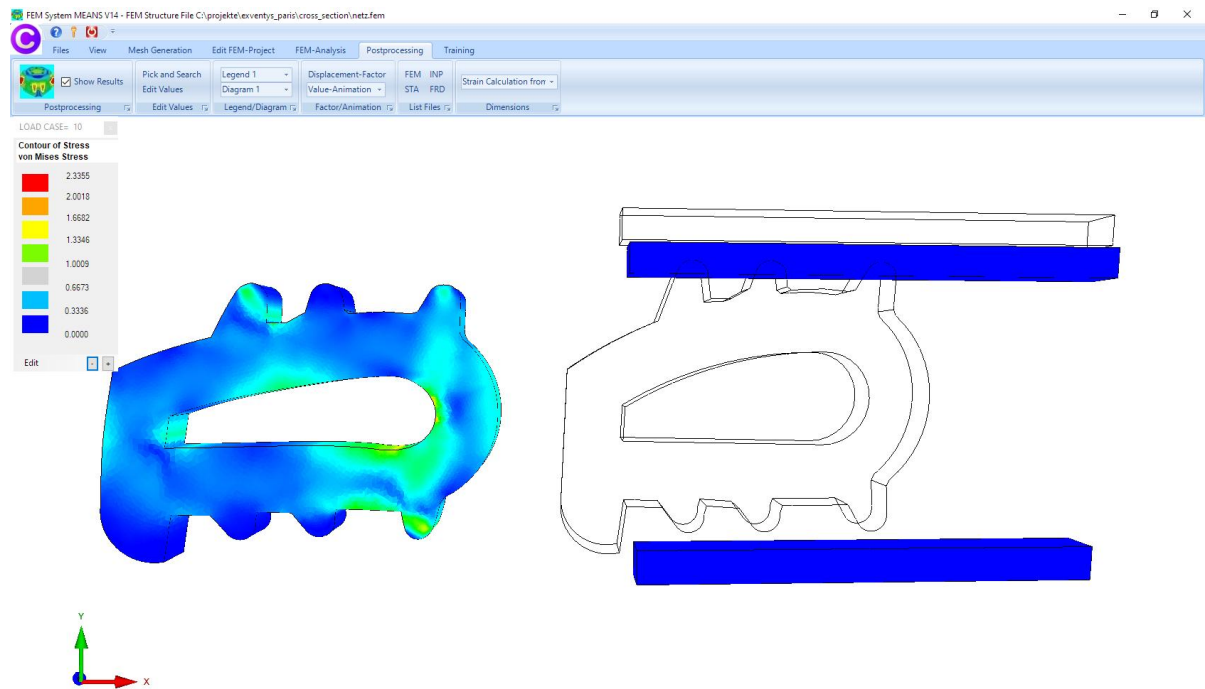
*ERROR in e_c3d: nonpositive jacobian
determinant in element      19243
```

So we must shorten the z depth from 25 mm to 5 mm with the Node Modus menu „Coordinate Factor“ which has no effect on the results and restart the nonlinear calculation.

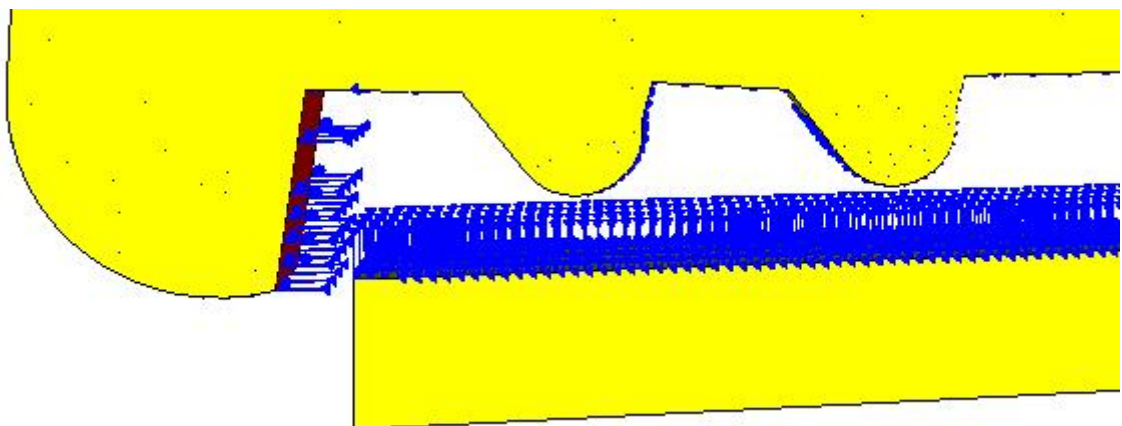


16.) Solver is failed again

The solver terminates the calculation again because the rubber is not held in the x direction.




Select the „Boundary Condition“ and click on Surface 8 to create the prescribed displacement of -0.5 mm in x direction on the Rubber and restart the calculation.

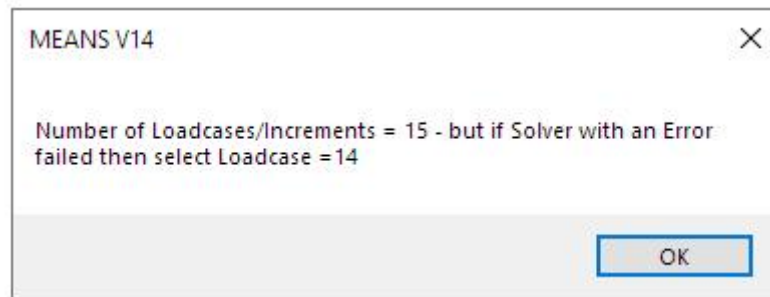


17.) Postprocessing

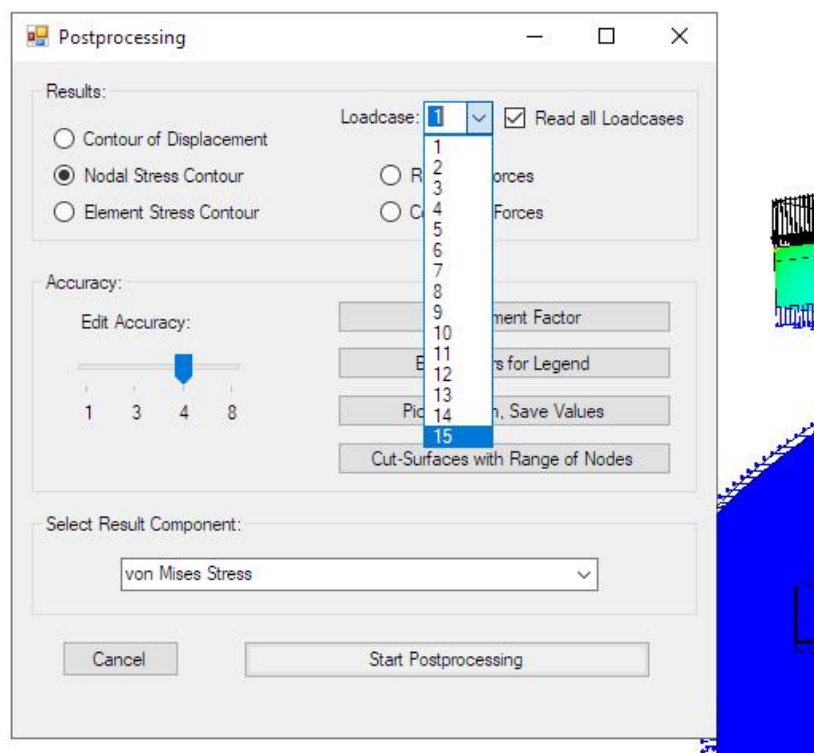
The nonlinear FEM-Calculation time is only a few minutes and now you can evaluate the large Displacements, vMises Stresses, Principal Stresses S1 or S3 or the Contact Pressure CPRESS.



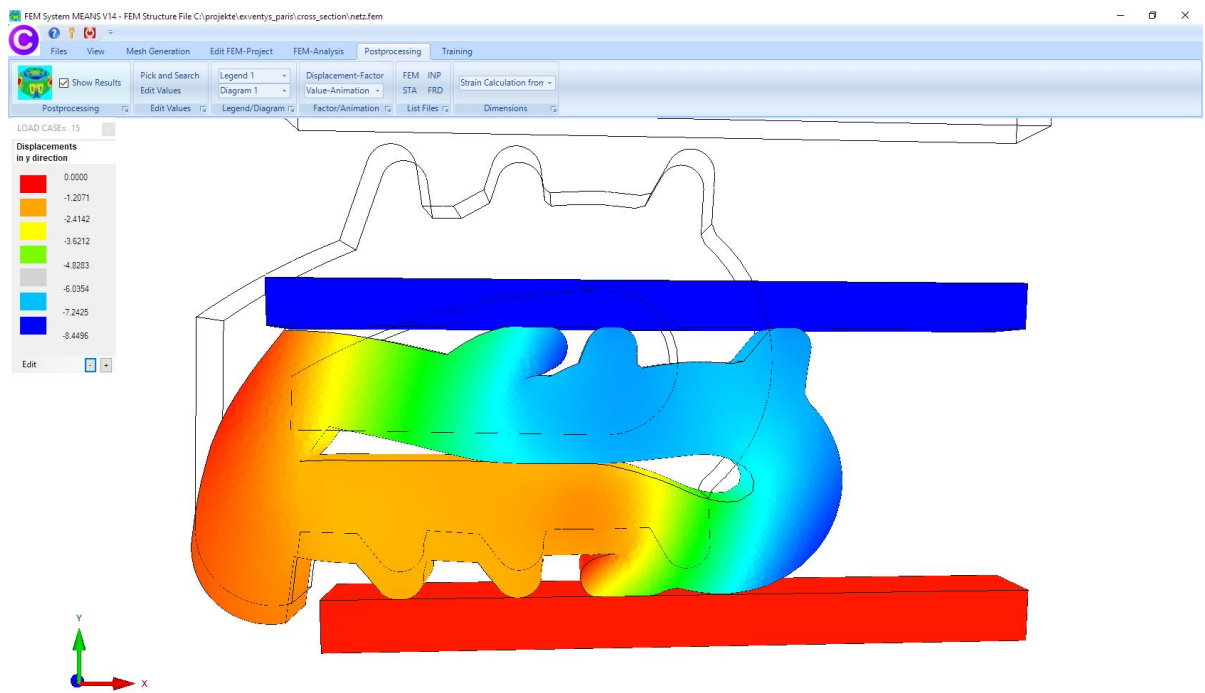
First select Register „Postprocessing“ and click on the  to show the Menubox and activate „Read all Loadcases“ and choose „Start Postprocessing“ to determine the number of load cases that could be calculated.



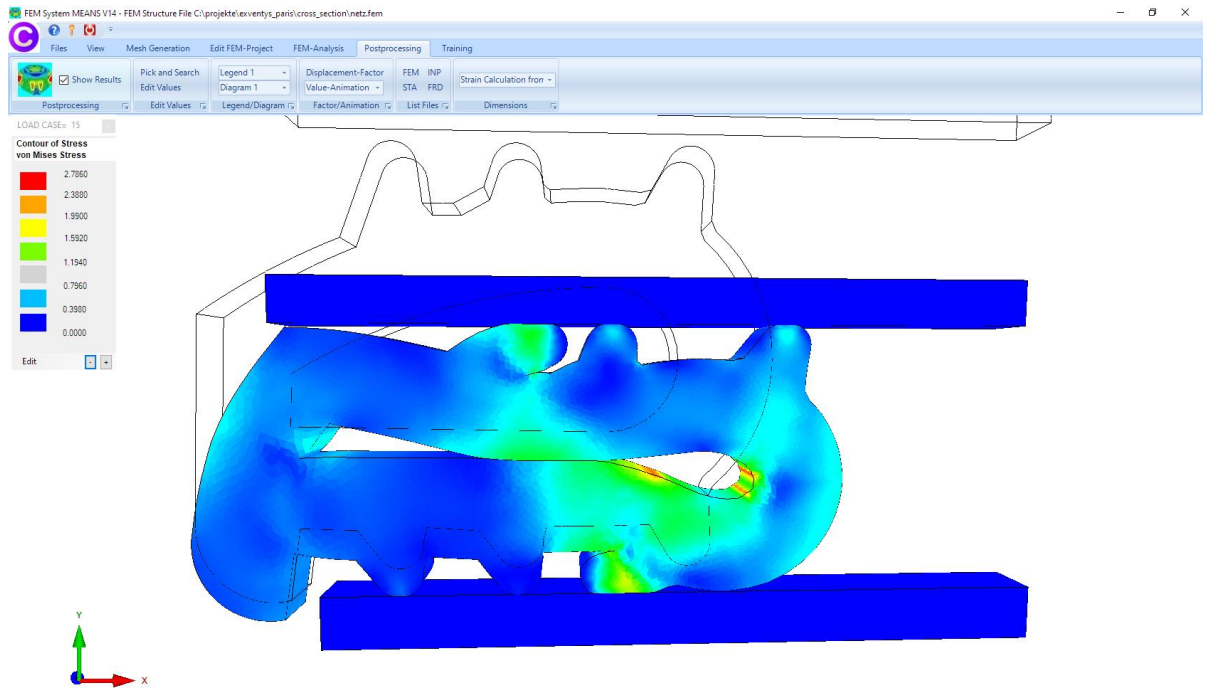
Then the number of load cases is displayed, and you can begin the postprocessing.

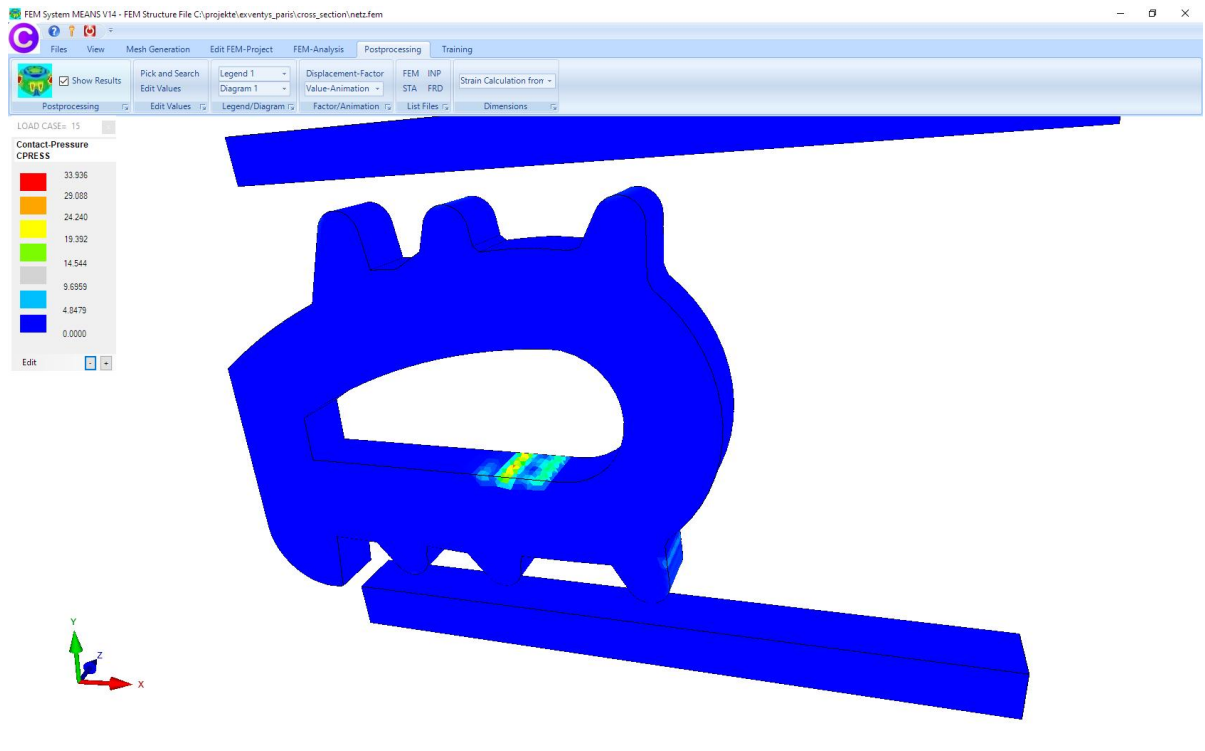


Large Displacement in y direction = - 8.44 mm



vMises Stress = 2.7 MPa



Contact Pressure CPRESS = 33 MPa**18.) Nonlinear Animation**

Select Register „Postprocessing“ and menu „Nonlinear Animation“ to animate the loadcases from 1 to 15. Please note: the larger the model, the slower the animation will be.

