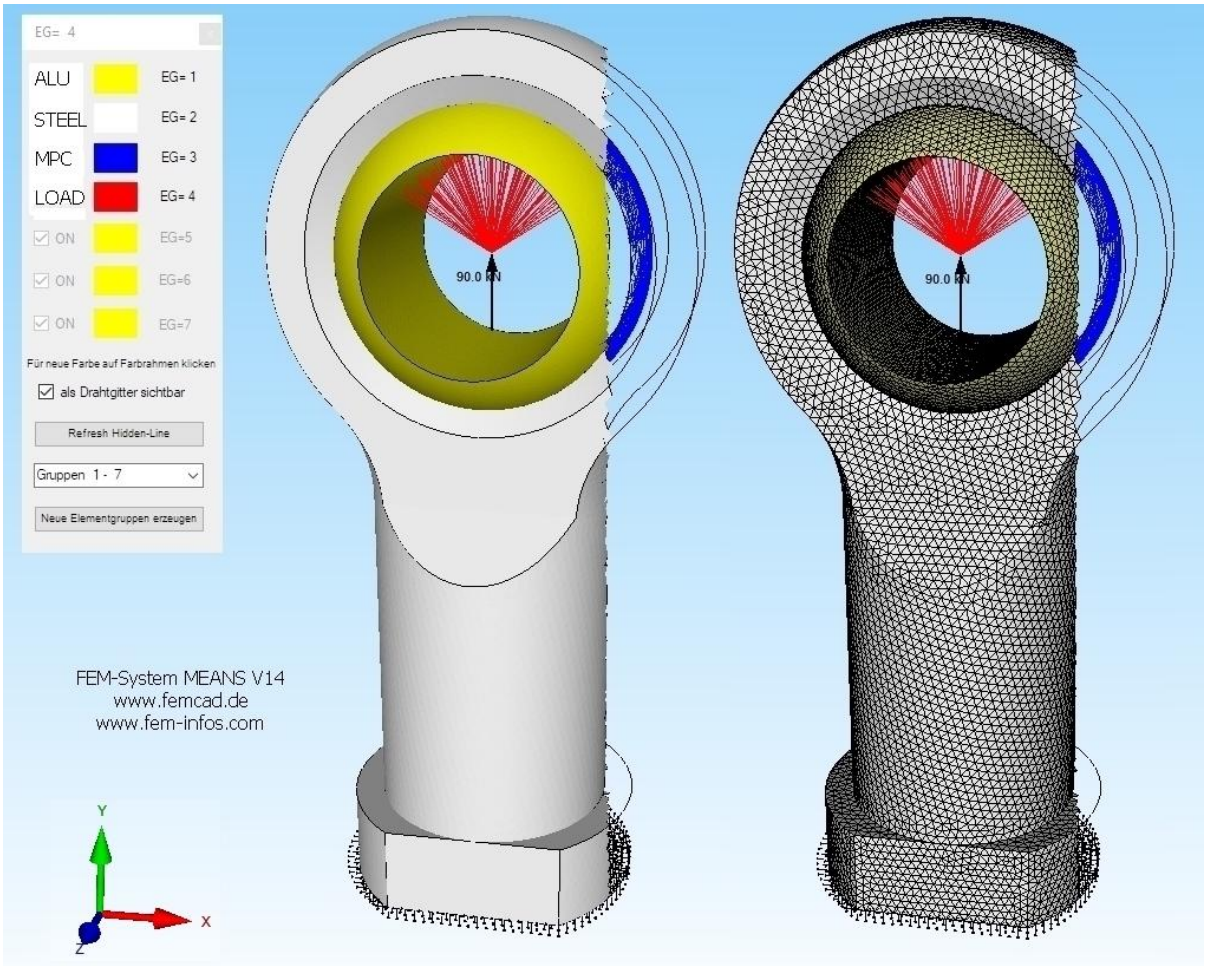
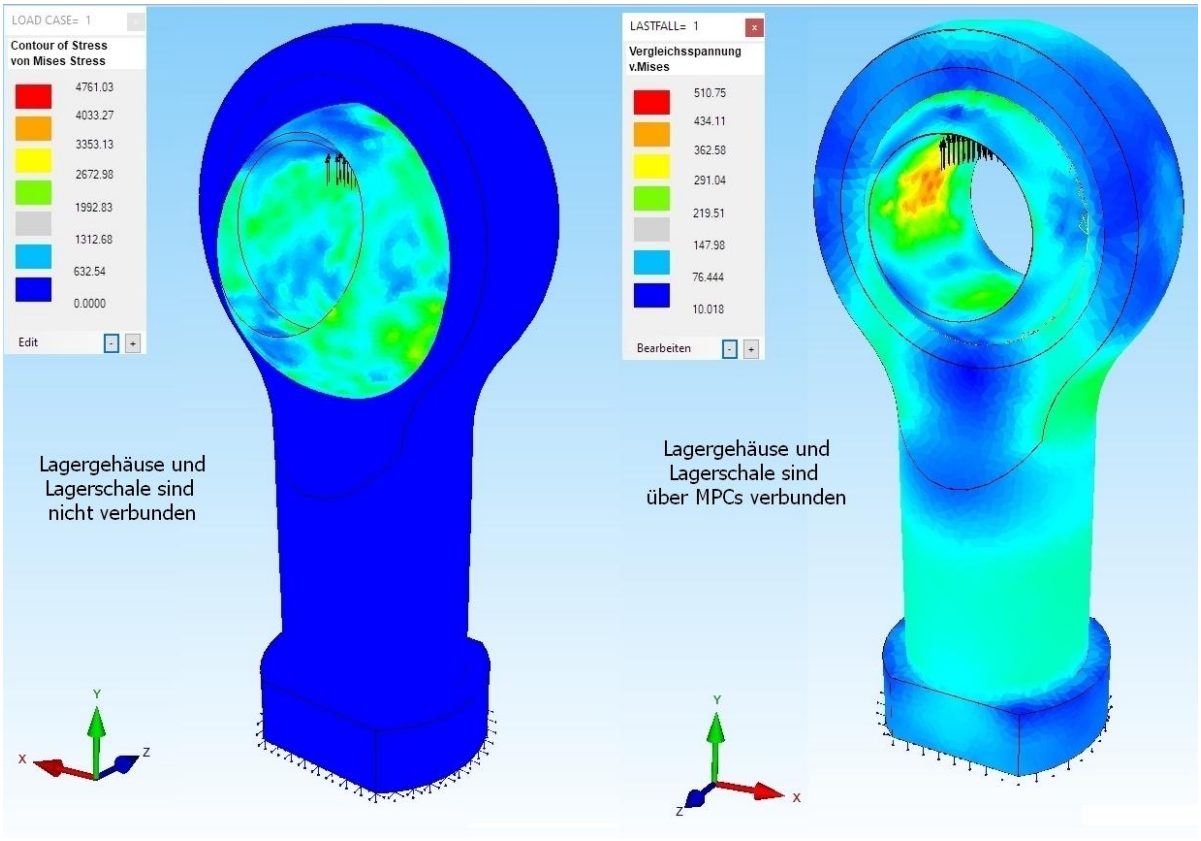
Part 39 - FEM-Analysis of a Rod End and Bearing Shell using MPC-Elements

The rod end cannot be calculated accurately enough using a "normal" FEM analysis. Therefore, the use of rigid beam elements, also known as MPC elements (multi-point constraint elements), is required:

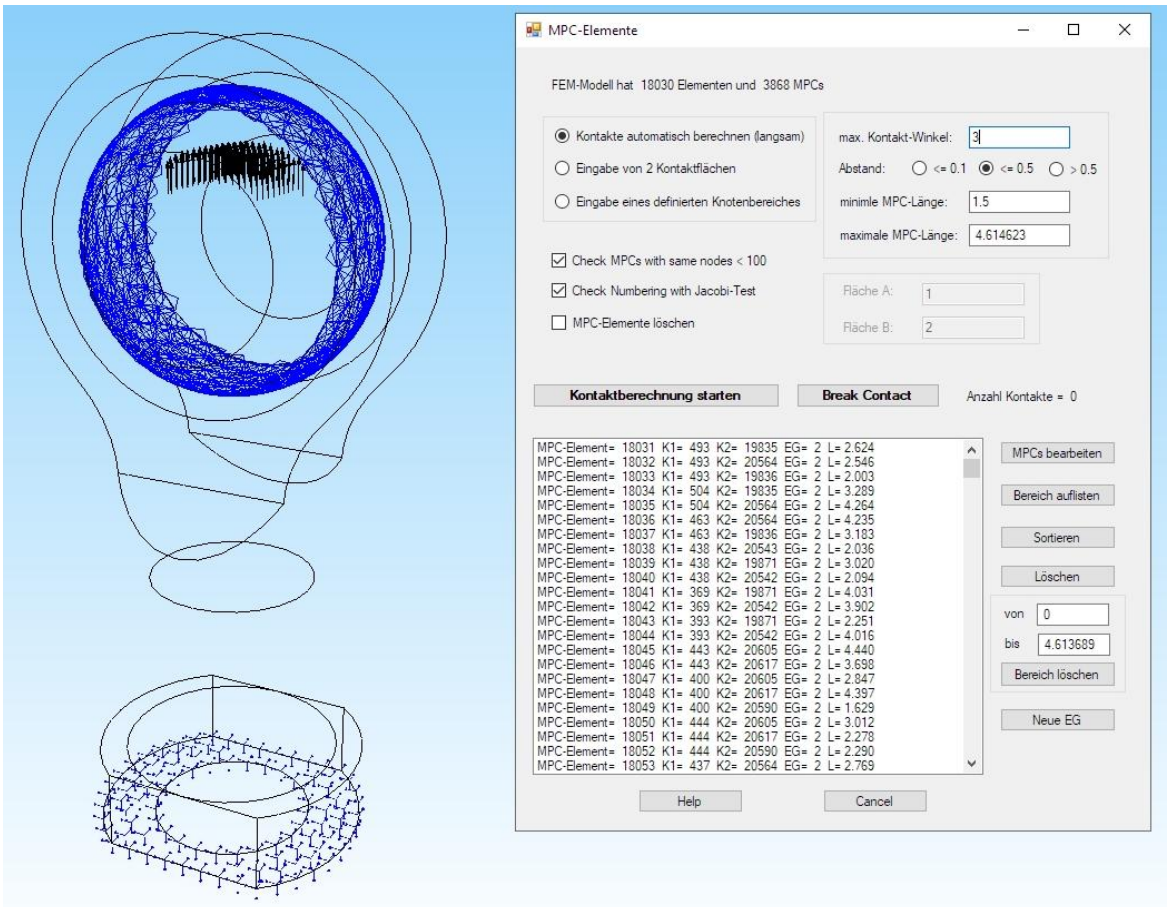
- To transfer the tensile load from the shaft to the bearing shell, a reference point in the center of the shaft must be connected to a node area of the bearing shell using MPC elements.
- To firmly connect the bearing housing to the bearing shell, surfaces of the bearing housing can be connected to surfaces of the bearing shell using MPC elements.
- In addition, the bearing housing and bearing shell must be meshed separately and combined using an FEM load to obtain two element groups for the elastic moduli of the steel housing and the aluminum bearing shell.



Tensile load acts outside the FEM mesh and must be transferred from the shaft center to the FEM nodes of the bearing shell using MPC elements.

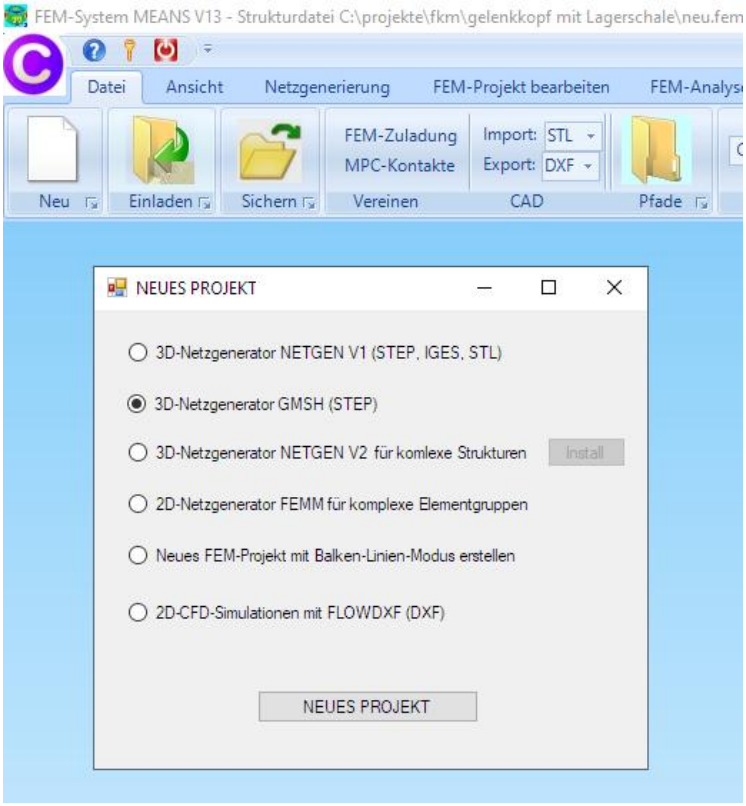


V. Mises stress of the bearing housing and bearing shell without a connection (left) and with MPCs of the bearing housing and bearing shell (right and below).

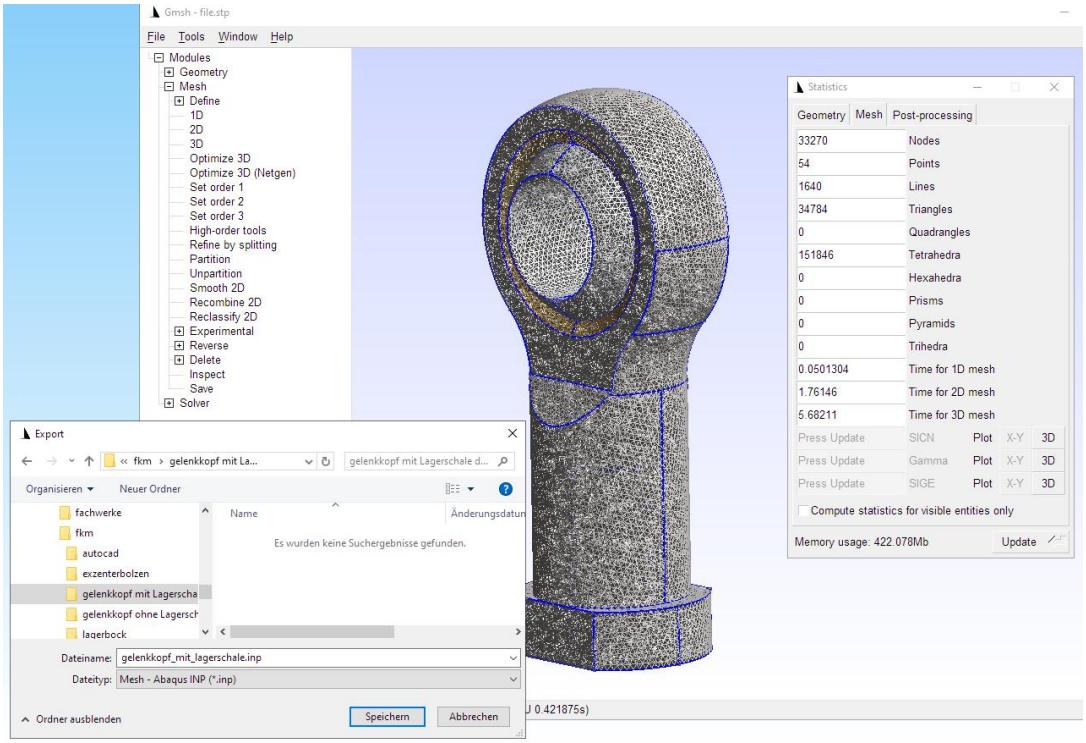


Meshing the Rod End

In MEANS V14, select the "New" tab and the "3D Mesh Generator GMSH (STEP)" menu to generate an FEM mesh with 151,846 tetrahedra and 40,139 nodes using the "3D" menu.

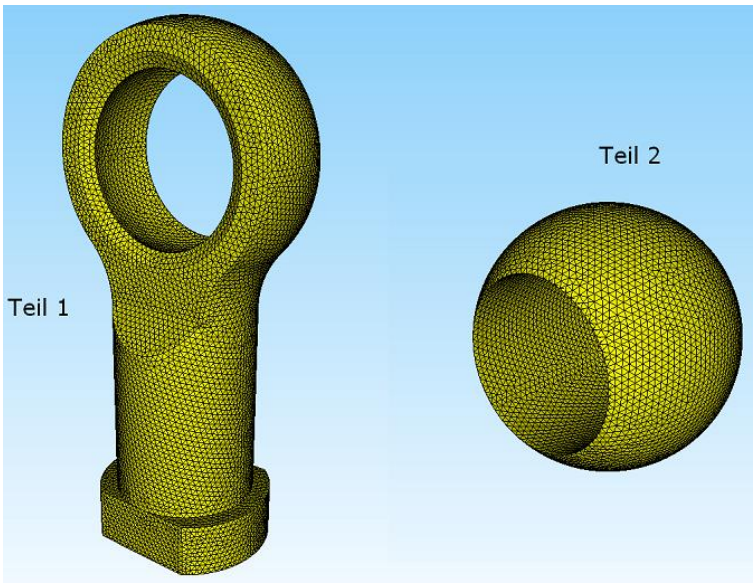


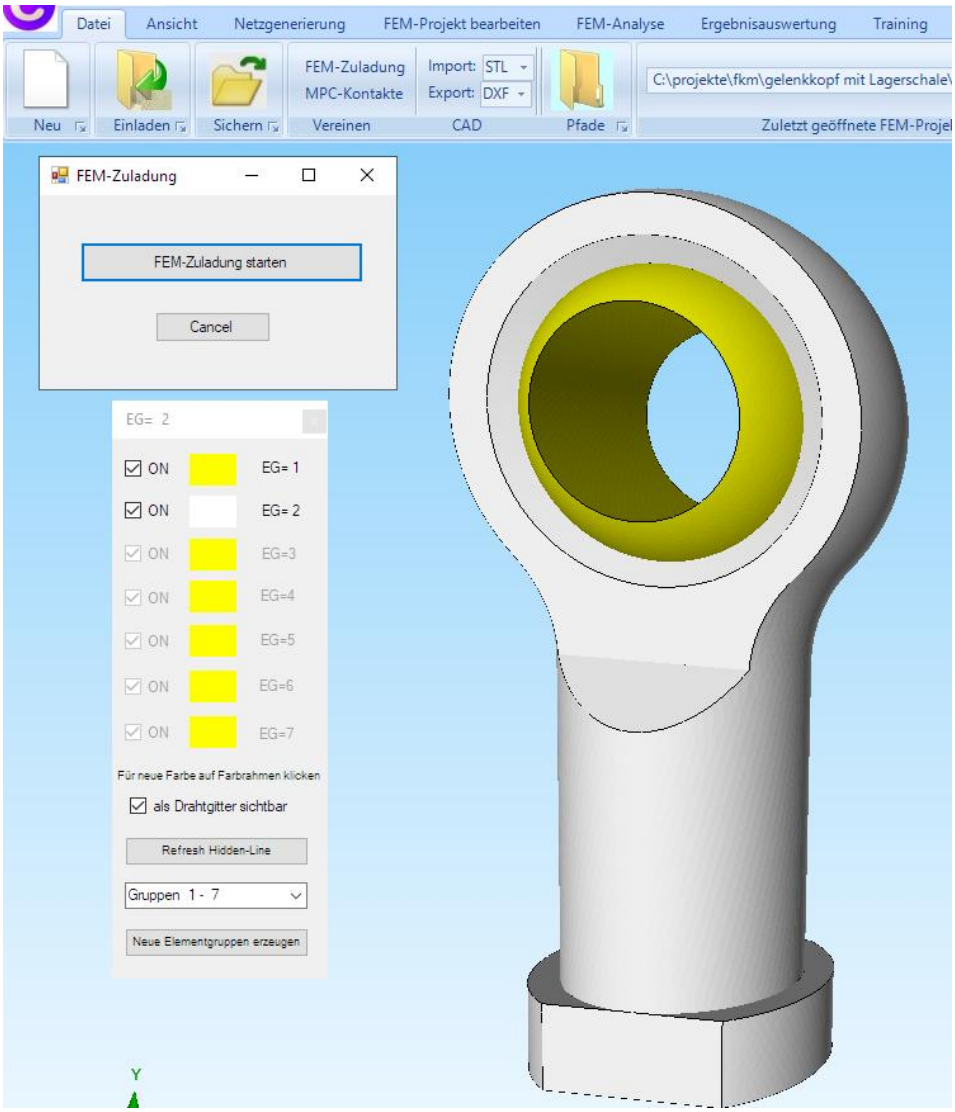
The FEM mesh must be exported in "Abaqus-INP" format, which can then be imported into MEANS V14.



Different Young's Moduli

If two different Young's Moduli are to be used for calculations, the bearing housing and bearing shell must first be meshed separately with GMSH and then combined using an FEM load.



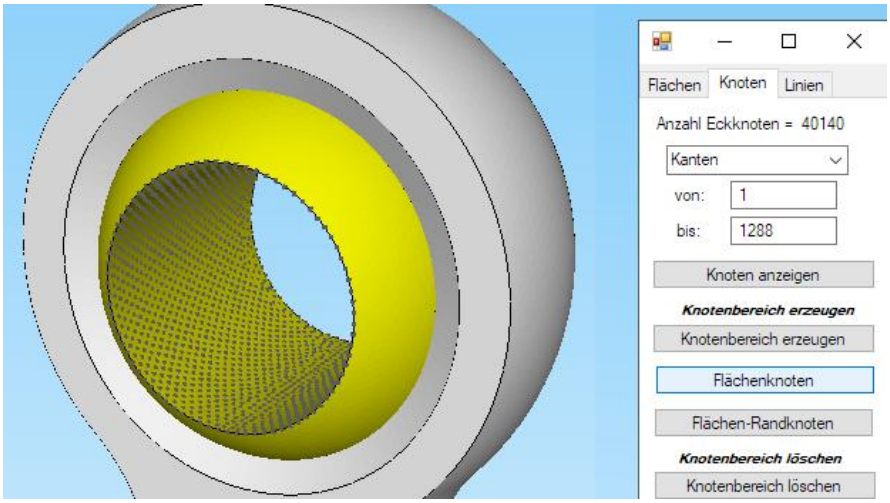


Creating a Tensile Load with a Node Area

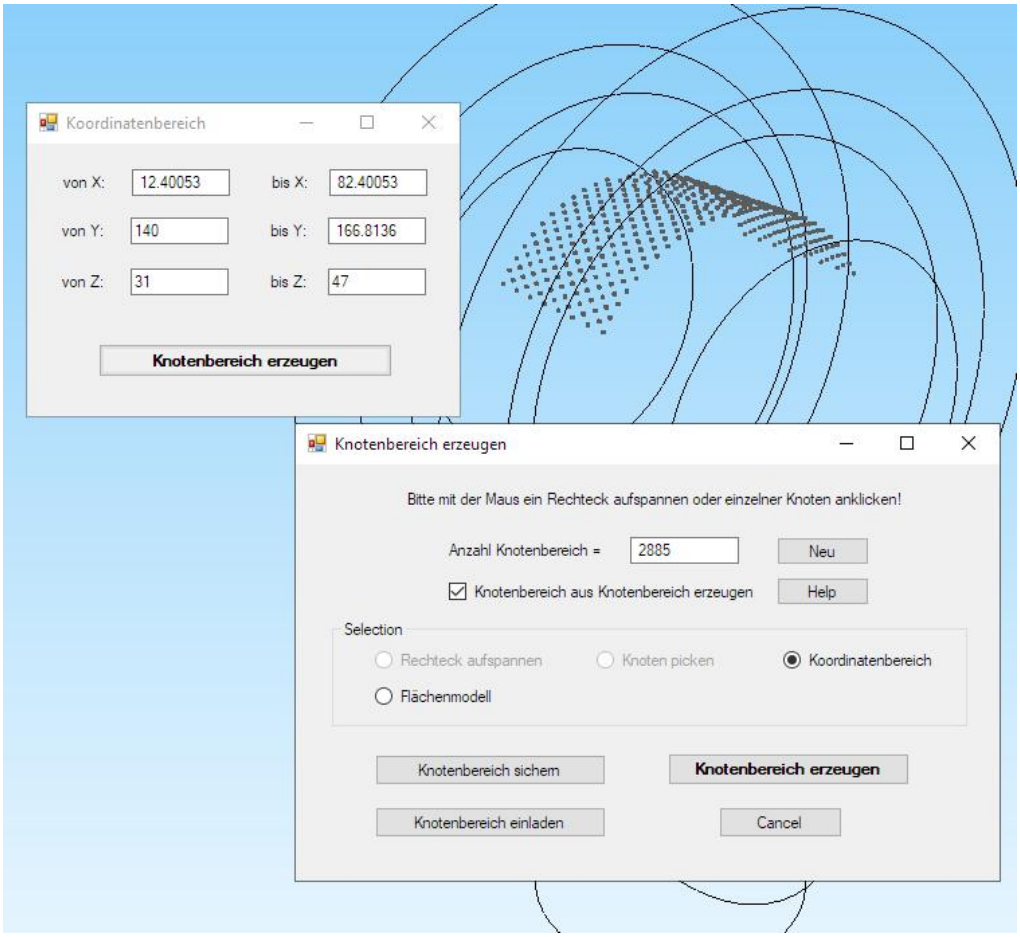
Since the shaft is not meshed, the tensile load must be defined with a reference point in the center of the shaft and connected to a nodal area of the bearing shell using MPC elements.

Define and shorten the range of nodes

In Node Mode and the "Surface Nodes" menu, create a range of nodes area of the Surface 6. Switch to "Wireframe" to shorten the Z-width to "31 mm to 47 mm" and

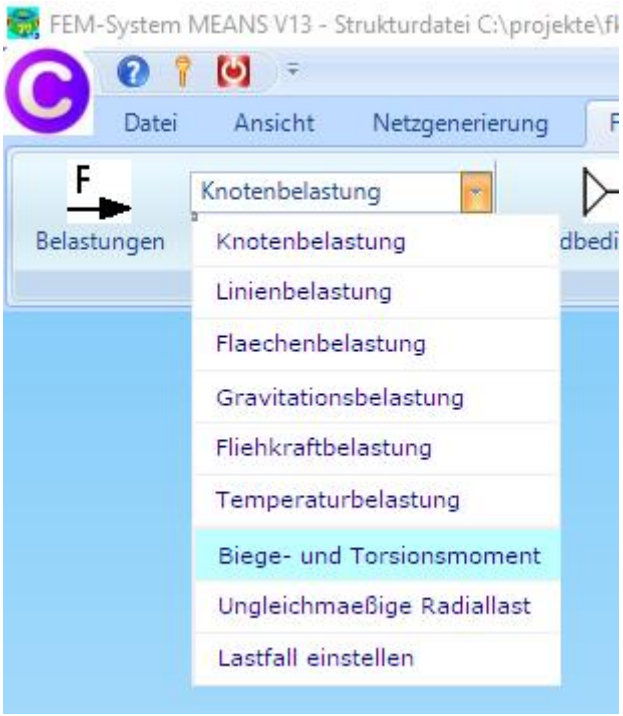


the Y-height to "140 mm to 166 mm" using a second range of nodes.

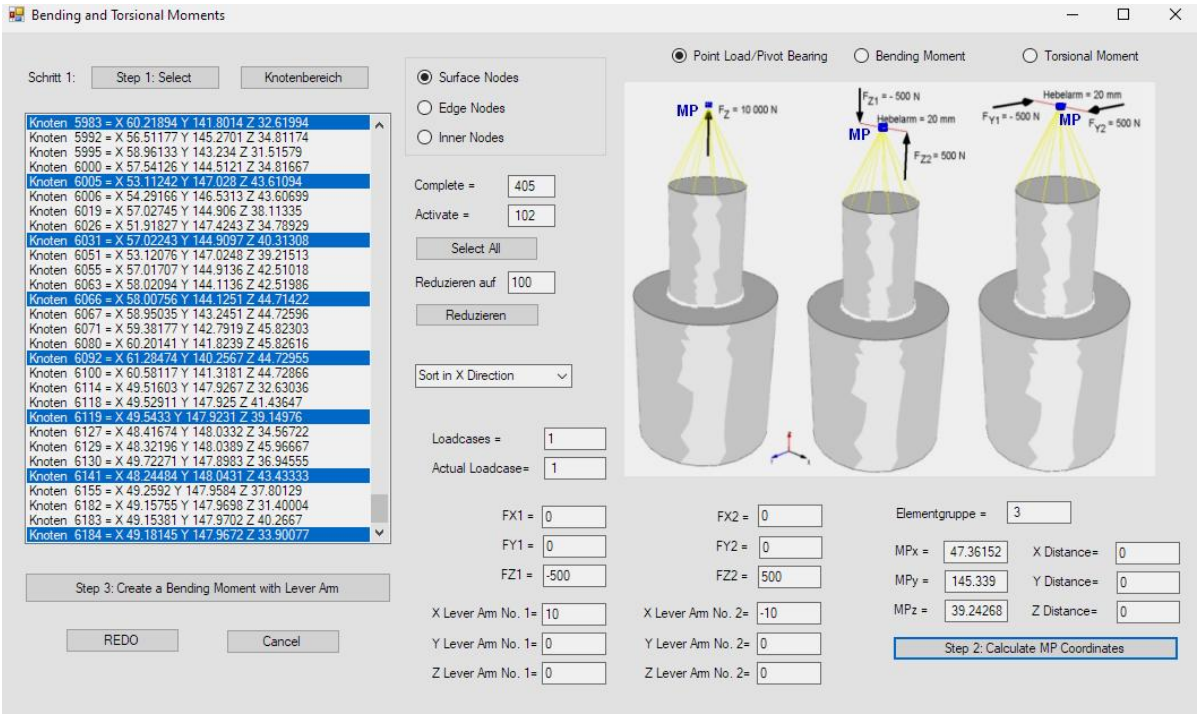


Create MPC Elements

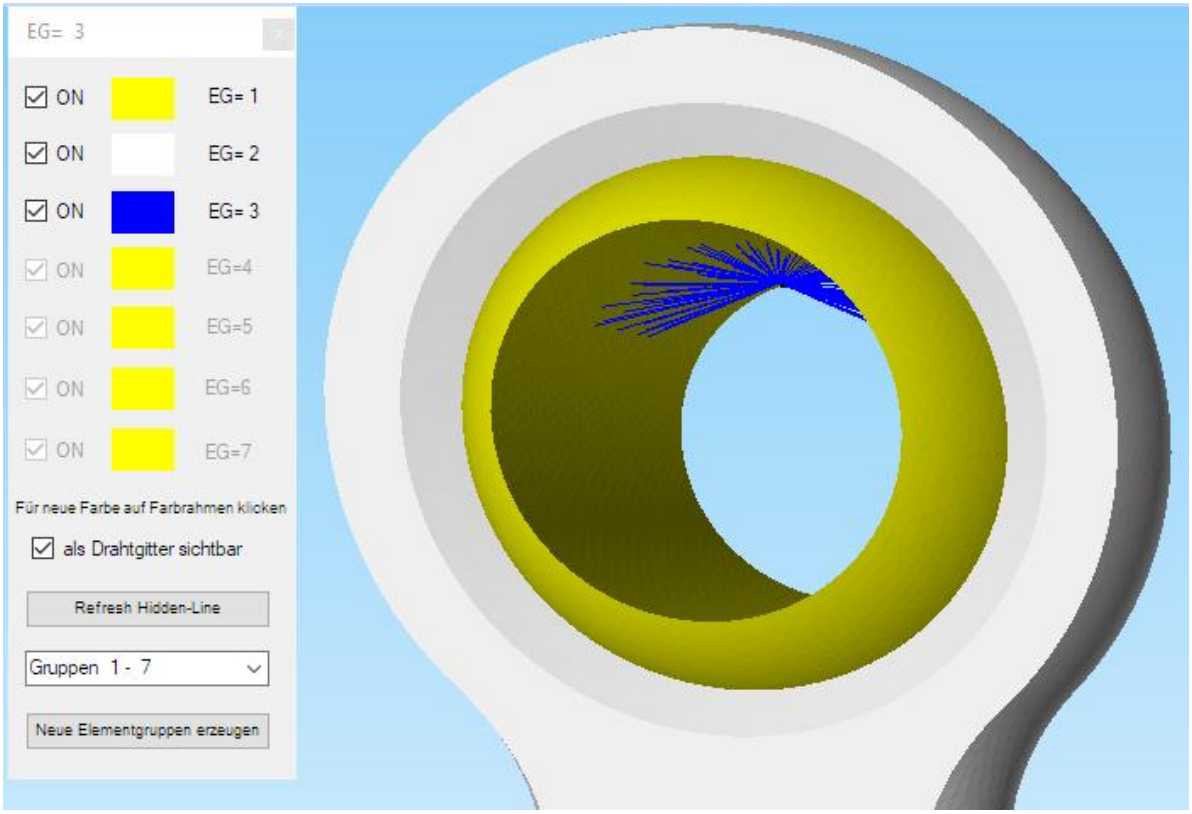
Now select the "Edit FEM Project" tab and the "Bending and Torsional Moment" menu to connect the nodal area to a reference point.



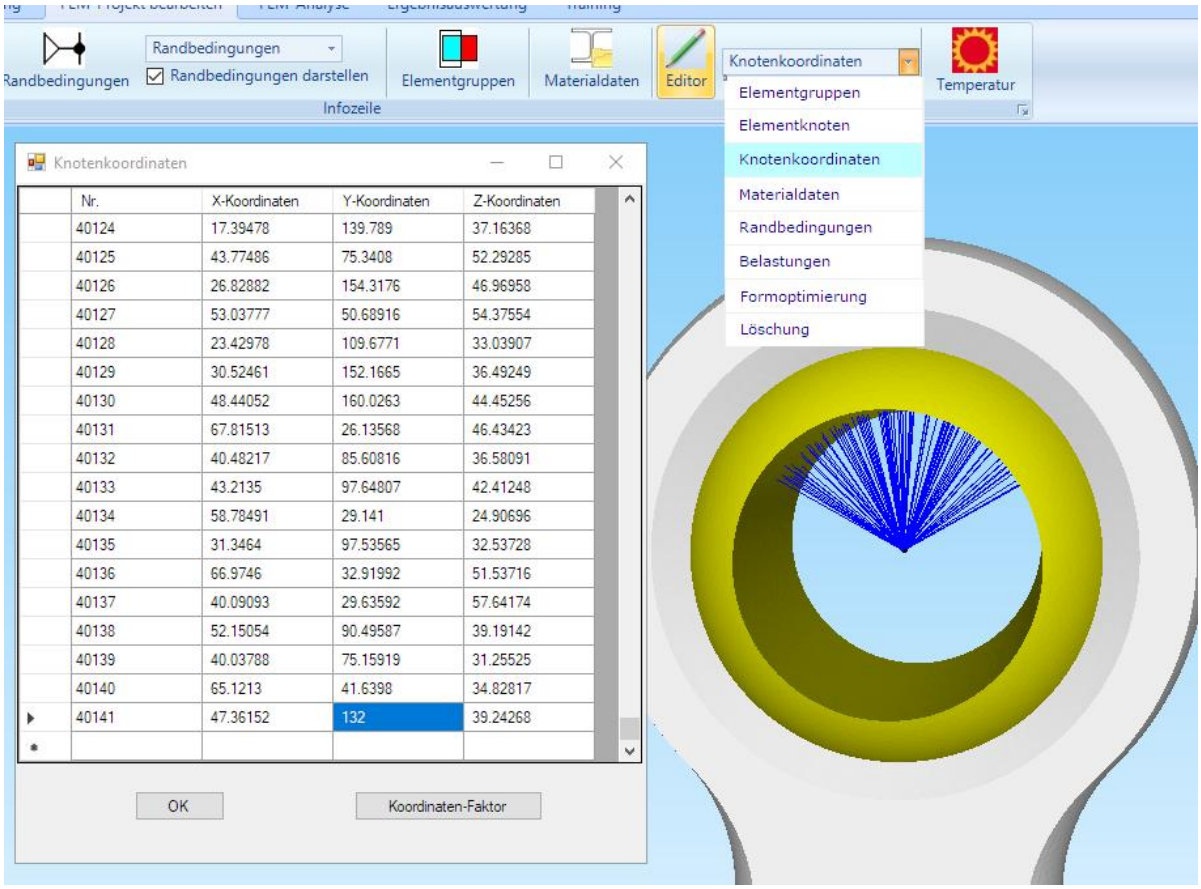
Use "Create Range of Nodes" to display the points in the dialog box and reduce the number to "100" because the Quick Solver can "only" calculate 100 beam elements with a common node. Then use "Step 2" to create the MP point and "Step 3" to create the MPC elements. Without much input, you have now generated an extended FEM model with reference point 40 140 and element group 3 consisting of beam elements.



A new element group 3 with the MPC elements is now available:



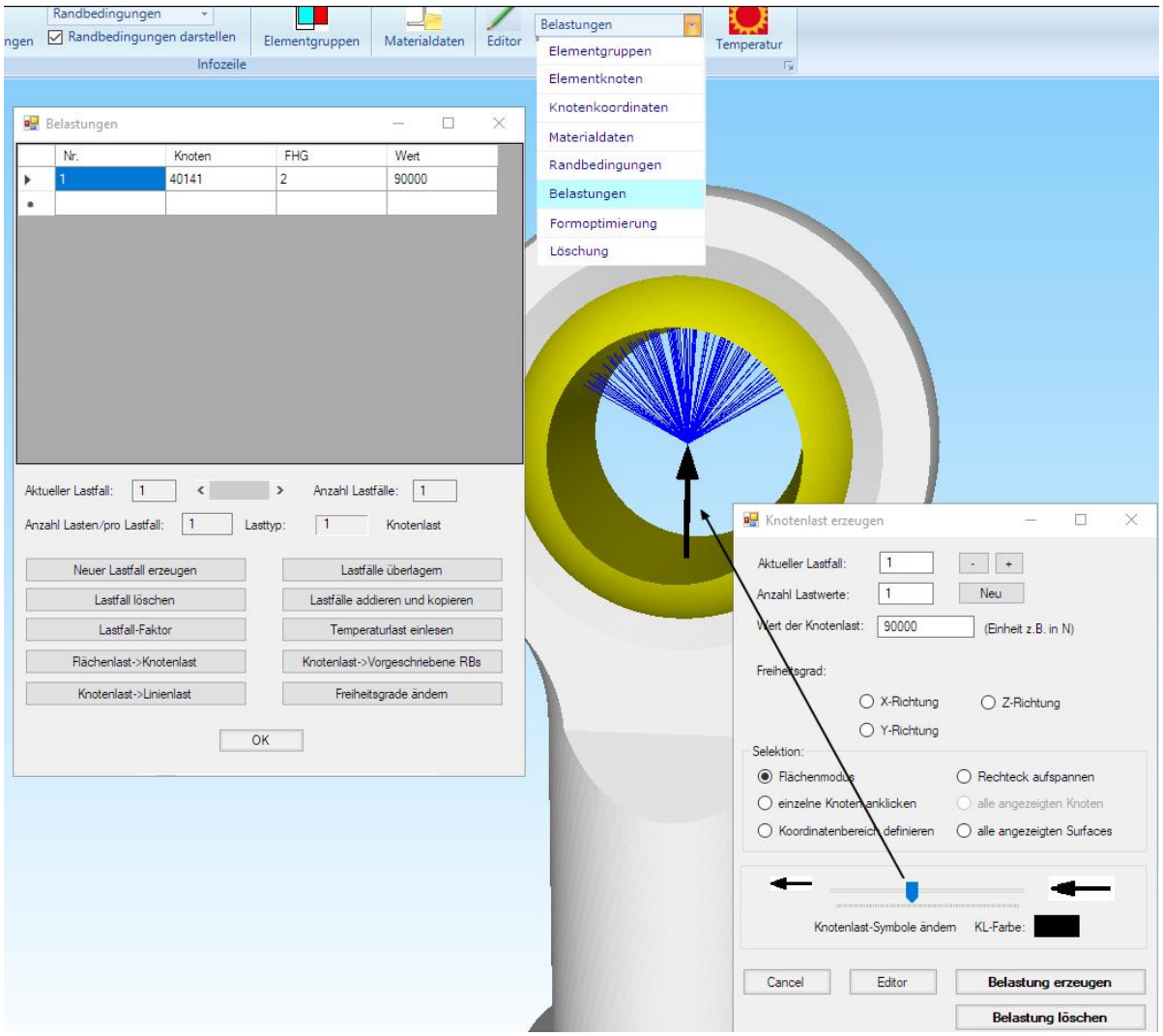
Select "Edit FEM Project" and "Editor" as well as "Nodal Coordinates" and set reference point 40141 to the shaft center with a Y value of "132" lower.



Nr.	X-Koordinaten	Y-Koordinaten	Z-Koordinaten
40124	17.39478	139.789	37.16368
40125	43.77486	75.3408	52.29285
40126	26.82882	154.3176	46.96958
40127	53.03777	50.68916	54.37554
40128	23.42978	109.6771	33.03907
40129	30.52461	152.1665	36.49249
40130	48.44052	160.0263	44.45256
40131	67.81513	26.13568	46.43423
40132	40.48217	85.60816	36.58091
40133	43.2135	97.64807	42.41248
40134	58.78491	29.141	24.90696
40135	31.3464	97.53565	32.53728
40136	66.9746	32.91992	51.53716
40137	40.09093	29.63592	57.64174
40138	52.15054	90.49587	39.19142
40139	40.03788	75.15919	31.25525
40140	65.1213	41.6398	34.82817
40141	47.36152	132	39.24268

Edit Loads

Select "Editor" and "Loads" again and enter a value of "90000" N in the Y direction at reference node 40141.
This menu can also be used to delete a load case or change the number of load cases and the number of load values by clicking on the cells and entering a new value.



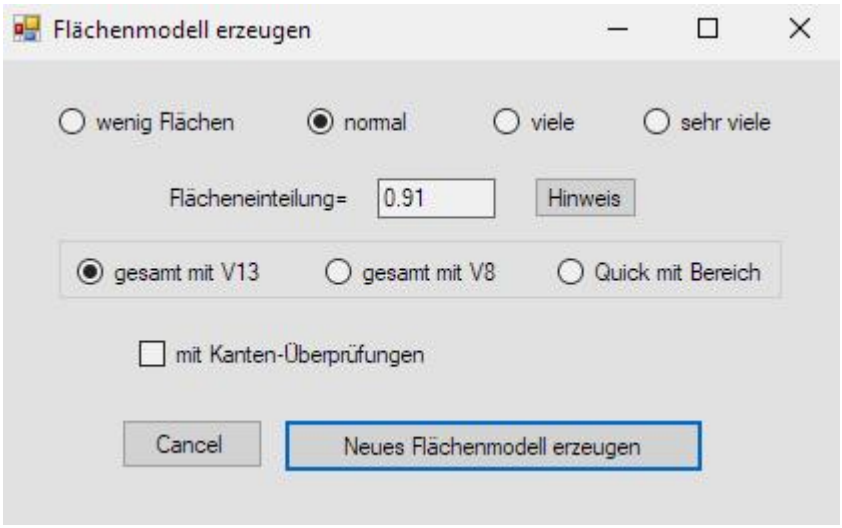
Set Arrow Size and Arrow Color

To change the arrow size, click the load icon 

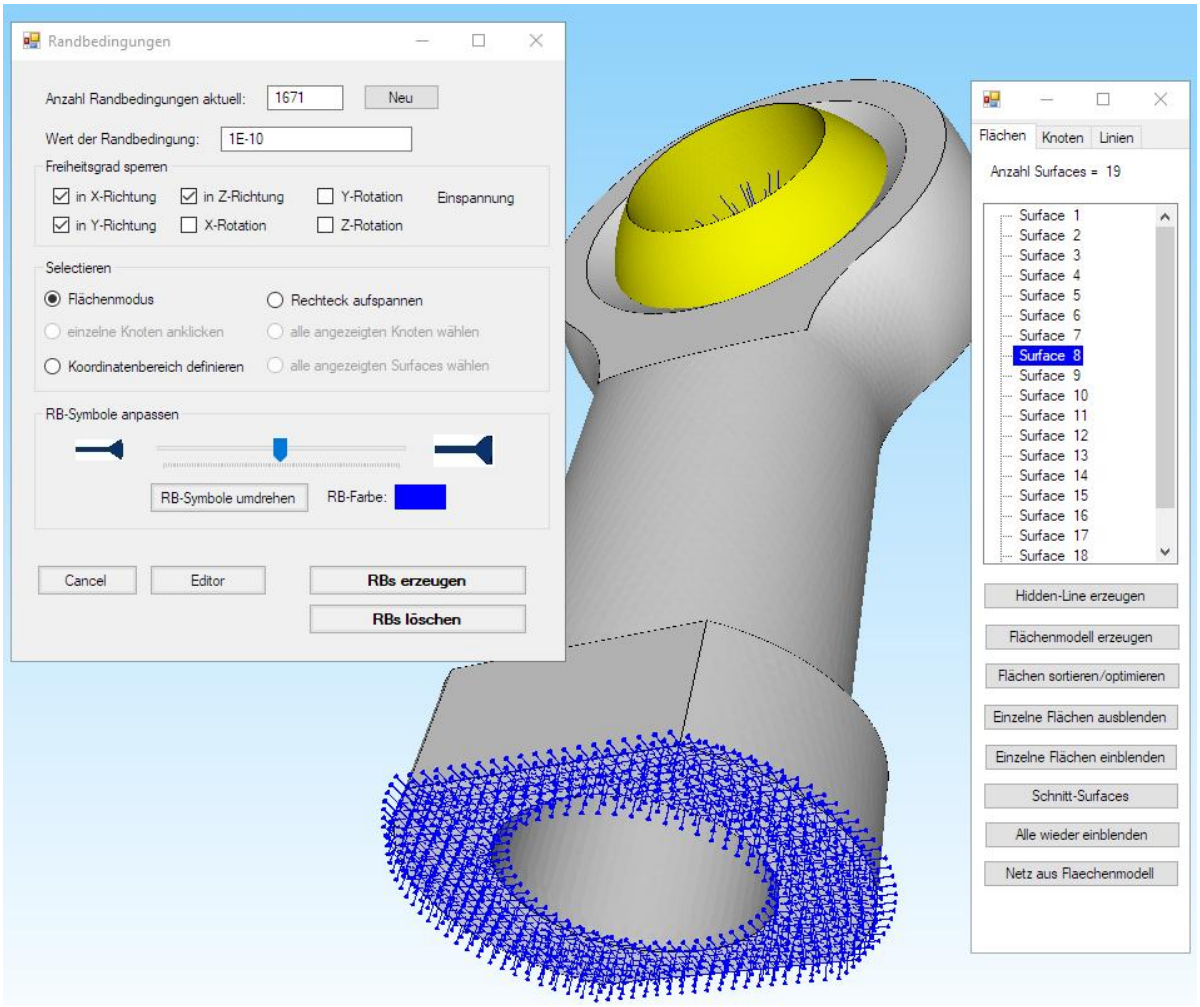
and use the slider to change the arrow size. Click the color box to change the arrow color.

Create Clamping

The rod end is clamped firmly at the bottom. Click the icon in the "Edit FEM Project" tab to clamp Surface 8 firmly in the x, y, and z directions.



However, the surface model must first be created using "Surface Mode," otherwise no surfaces can be selected.



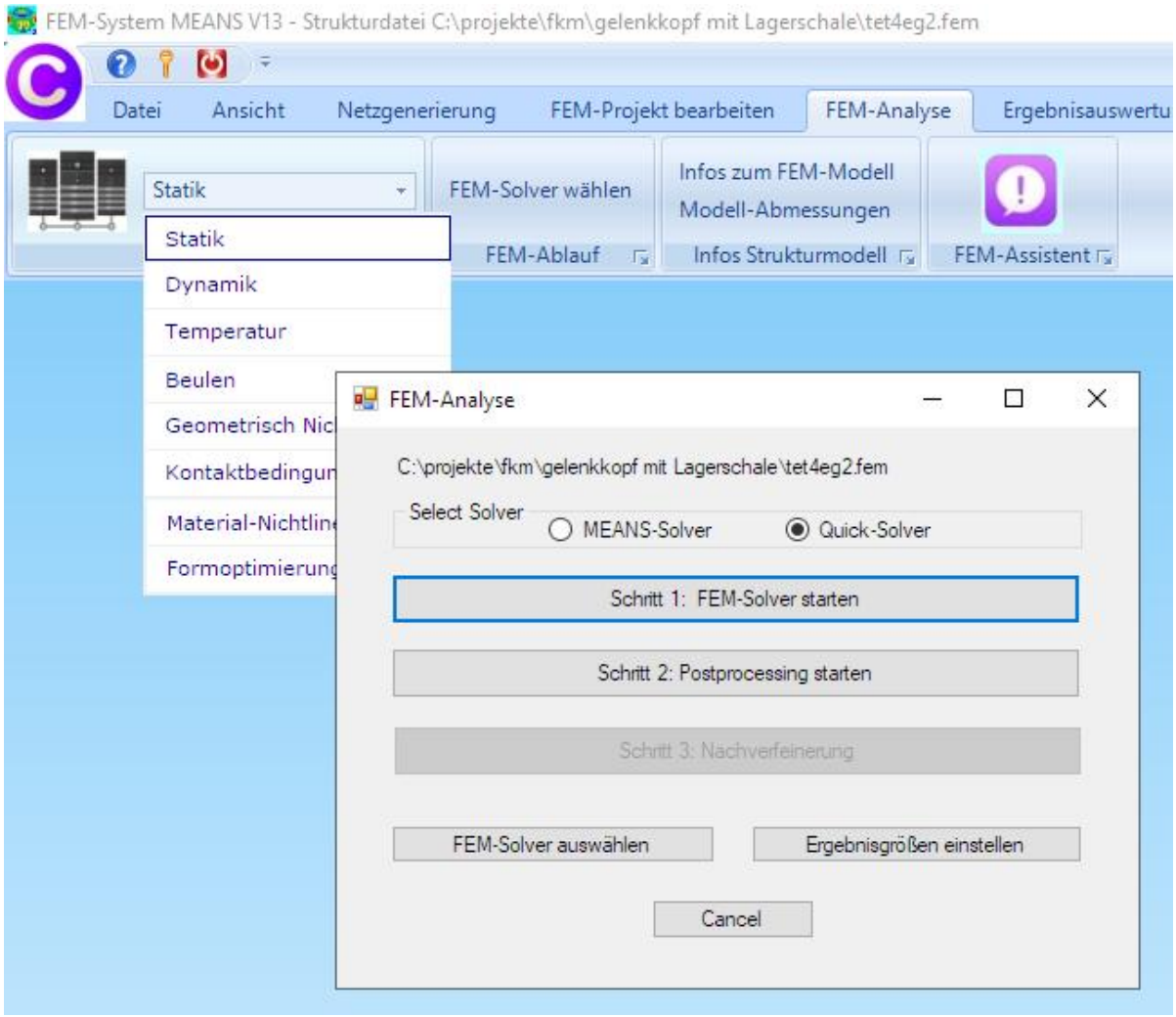
First FEM Analysis

Finally, the elastic moduli and material data must be entered via the "Edit FEM Project" tab:

Element Group 1
Elastic modulus for aluminum = 70,000 MPa
Poisson's ratio = 0.3

Element Group 2
Elastic modulus for steel = 210,000 MPa
Poisson's ratio = 0.3

For the first FEM analysis, select the "FEM Analysis" and "Statics" tabs to calculate the deformations and stresses using the Quick Solver.

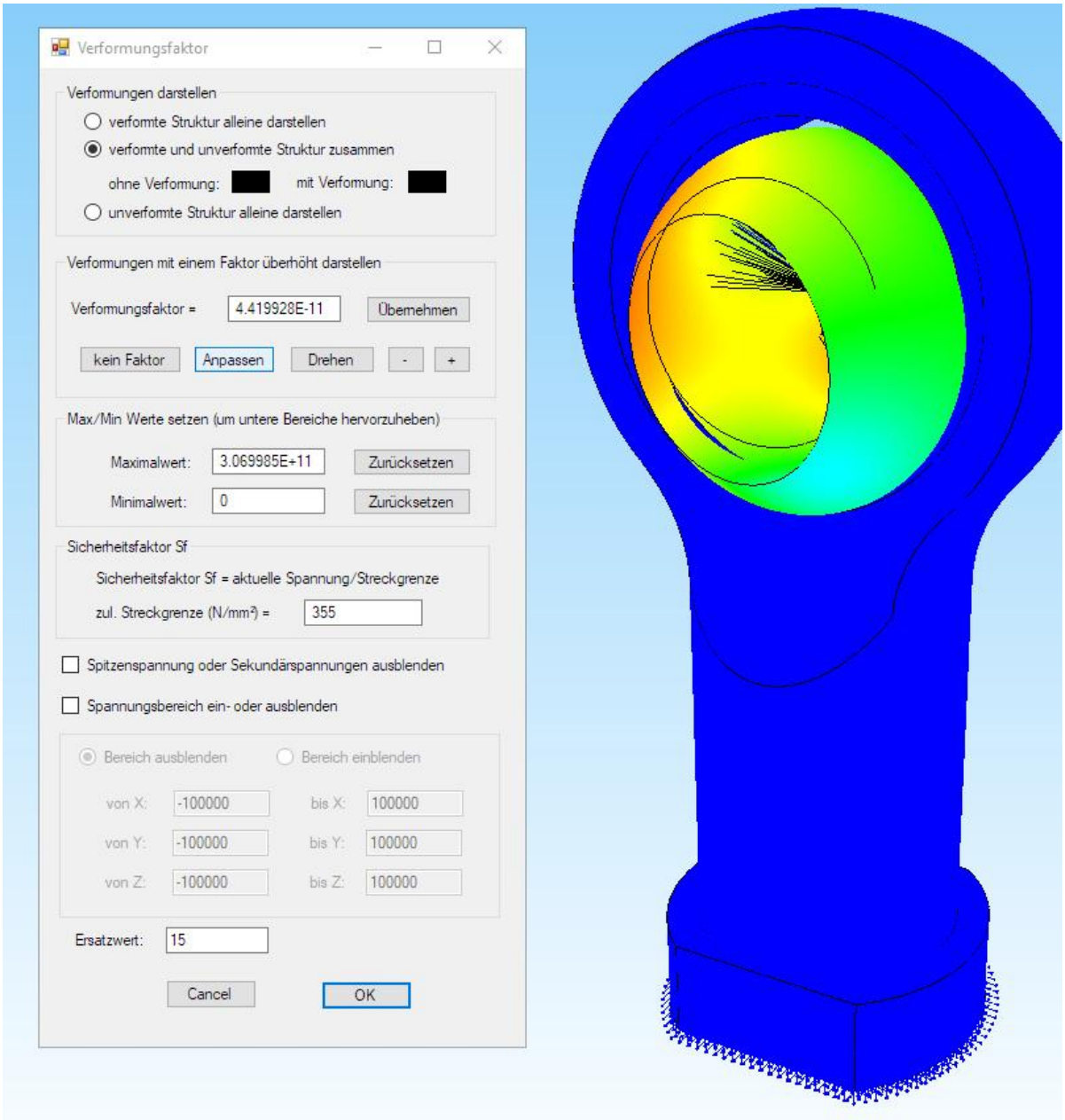


No plausible results

Due to the lack of contact with the bearing housing, the plain bearing shell has been spherically displaced from the housing and is no longer visible. A maximum deformation of 1E+11 mm is displayed.

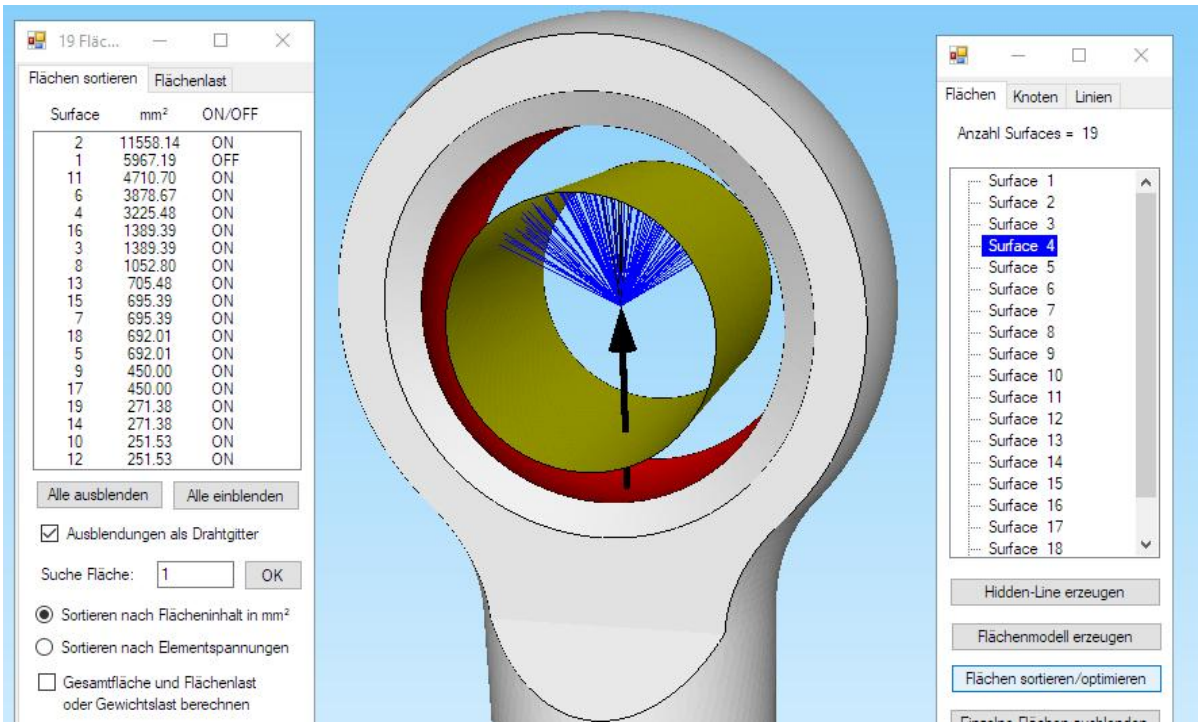
Deformation Factor

In the "Result Evaluation" tab, select the "Deformation Factor" menu and "Adjust." The bearing shell will become visible again with a very small deformation factor of 1E-11.



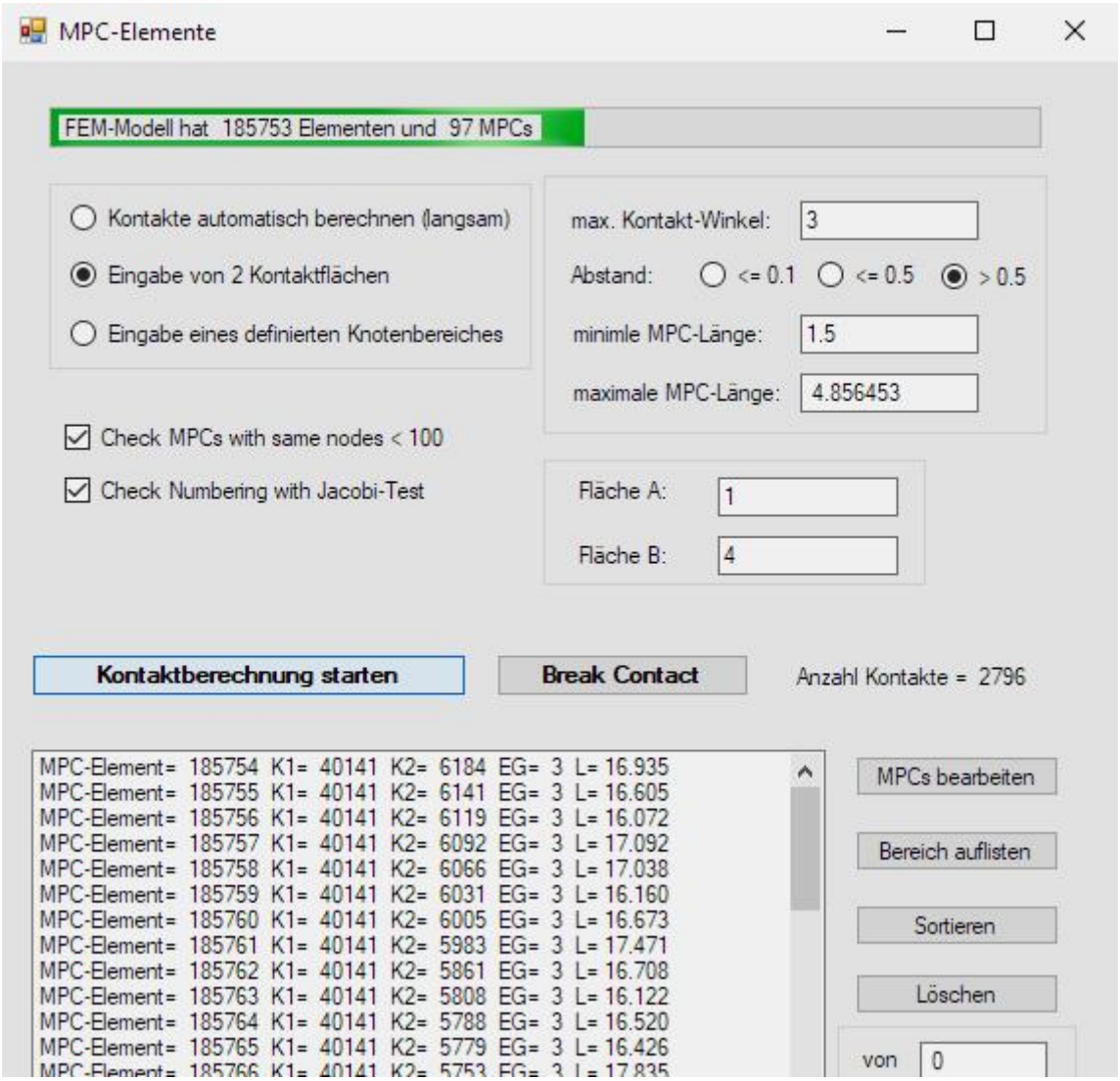
Connect the bearing housing and bearing shell

In "Surface Mode," select the "Sort/Optimize Surfaces" menu and hide the second-largest surface 1 of the bearing shell with "OFF" to fix surface 4 of the bearing housing with one click. Surfaces 1 and 4 can now be connected with "rigid" MPC elements.

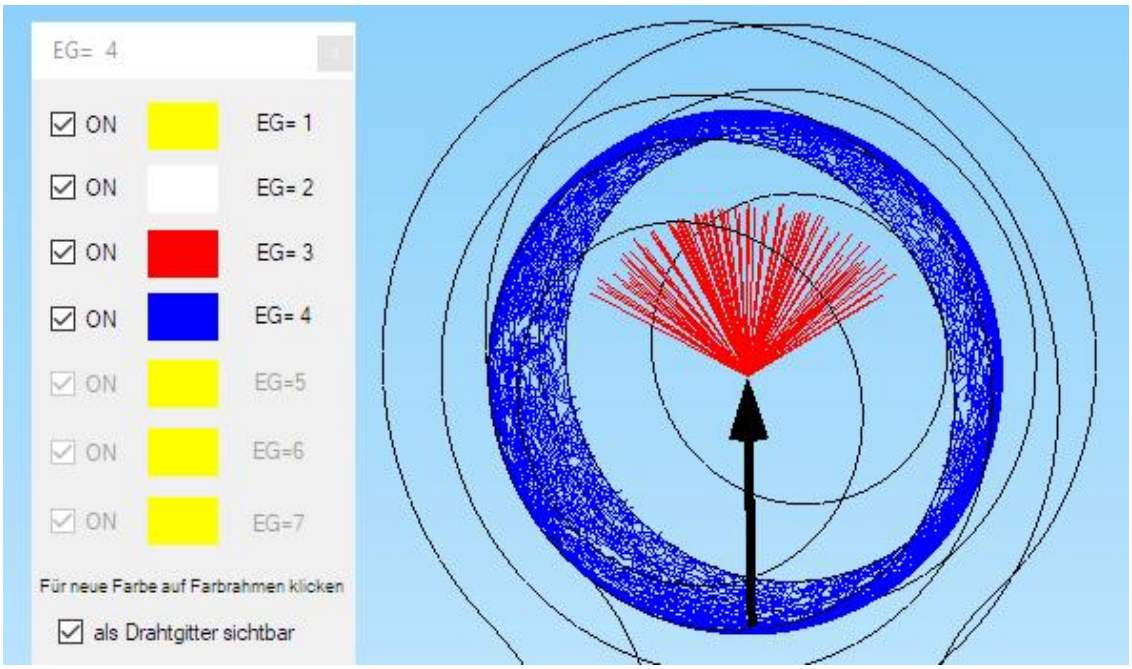


MPC Contacts

To generate the MPC contact analysis, which is a flexible alternative to surface-to-surface contact analysis, select the "File" tab, the "MPC Contacts" menu, and then select the "Input of 2 contact surfaces" option with area A = 1 and area B = 4. Furthermore, the maximum contact angle = 3 is a very important input parameter that can often only be determined after several runs.
Start the calculation with the "Start Contact Calculation" menu. You can also cancel it with the "Break Contact" menu.



FEM model with 185,753 elements, 6,345 MPC elements, and 4 element groups:



Postprocessing

After the FEM analysis, the following v.Mises stress distribution is obtained: