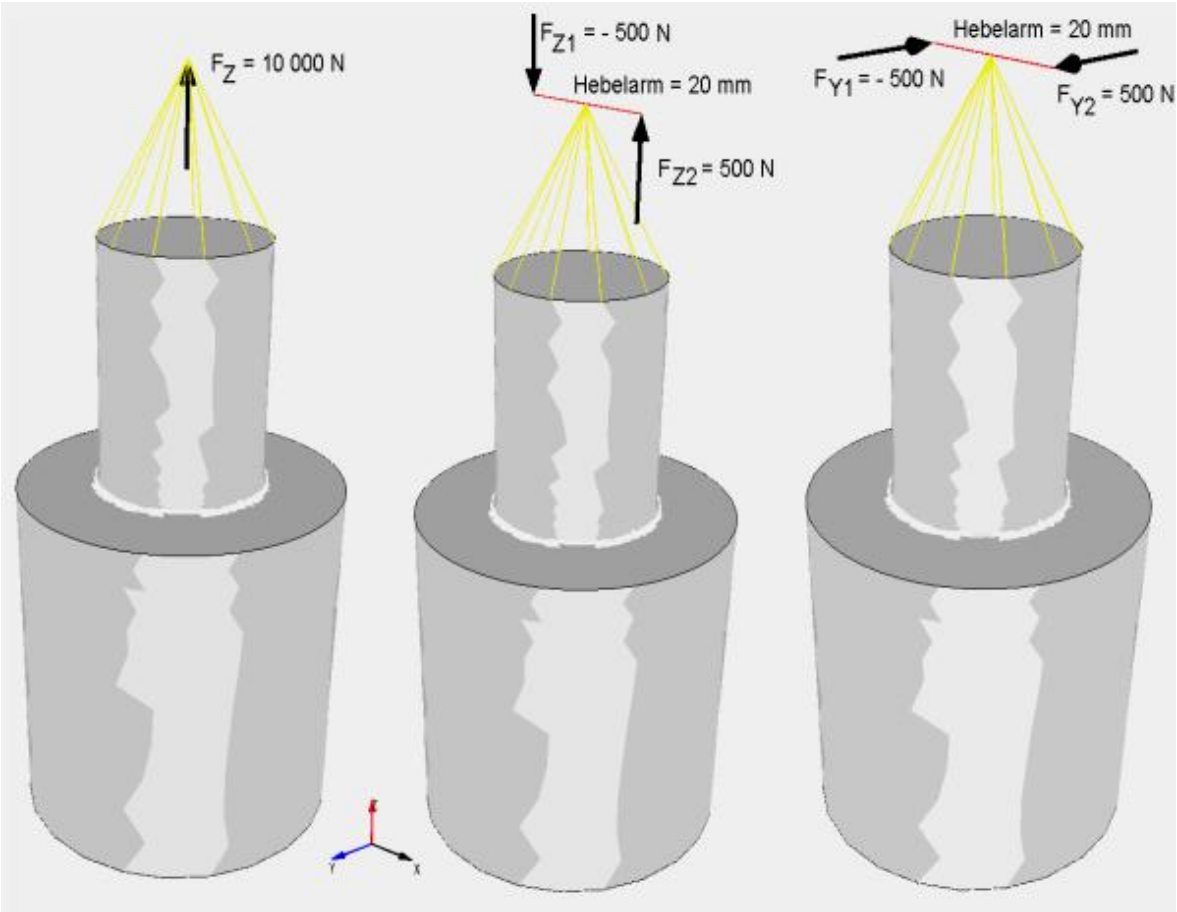
FEM-System

MEANS V12

Bending and torsional moments on a shaft

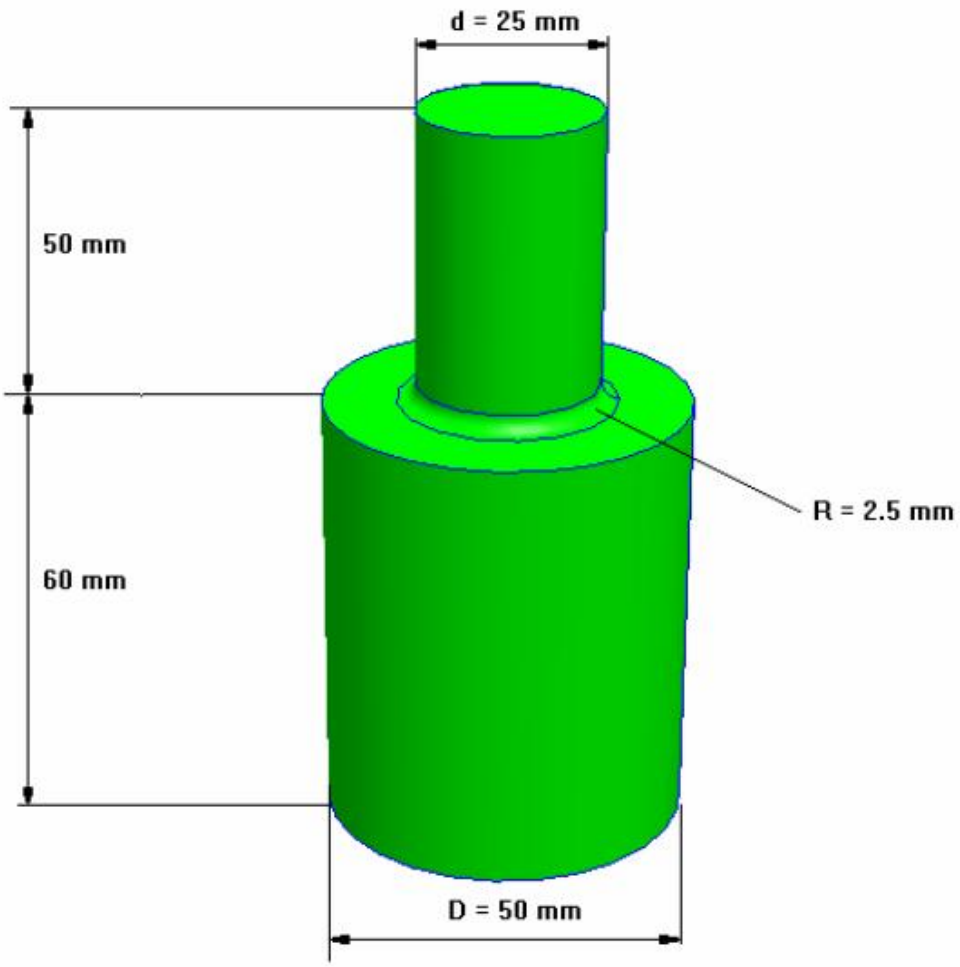


www.femcad.de

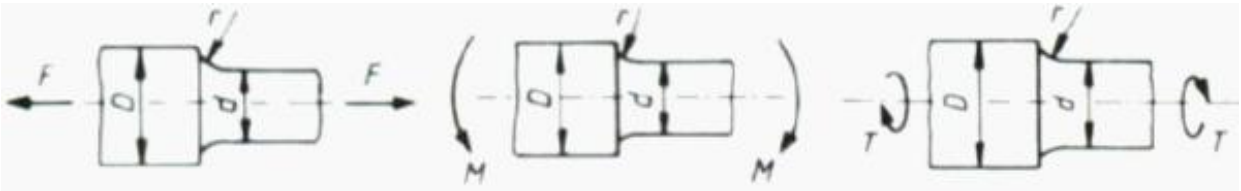
www.fem-infos.com

Part 10: Bending and torsional moments an a shaft with MEANS V12

The shaft consists of 2 shaft sections with D = 50 mm and d = 25 mm and a shaft radius of R = 2.5 mm.



The shaft is loaded with a tensile load of 10 000 N, a bending moment of 10 000 Nmm and a torsional moment of 10 000 Nmm. How big are the deformations and stresses.

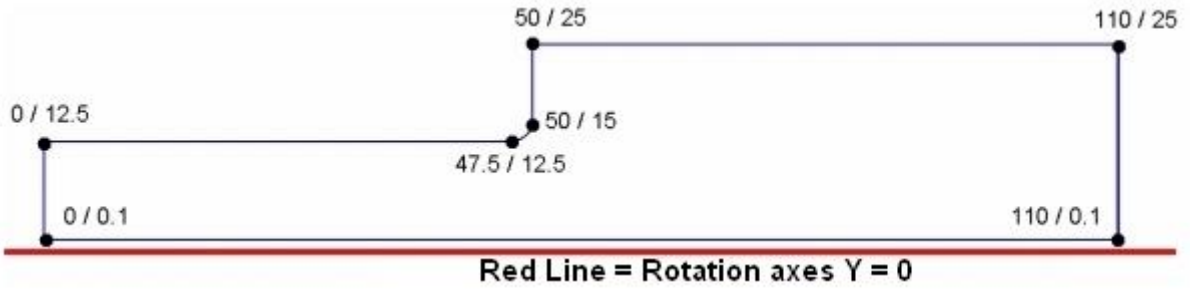


Tensile load = 10000 N Bending moment = 10000 Nmm Torsion moment = 10000 Nmm

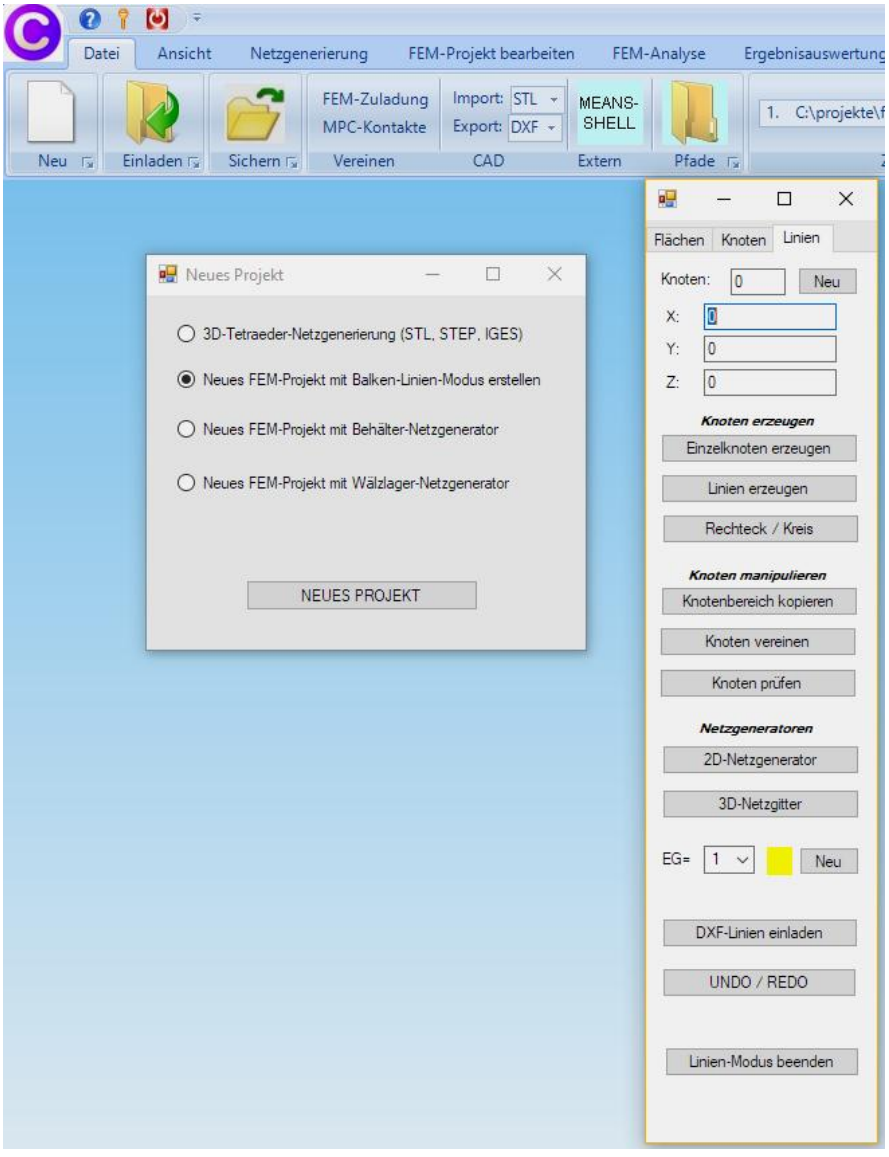
Create FEM model

The shaft can be generated with MEANS V12 in line mode as in a CAD.

First of all, a half-shaft cross-section is used to generate a 2D mesh with triangles or quadrilaterals. From this, a pentahedron or hexahedral element mesh can be generated with the 3D rotation generator.



The lower line runs at Y = 0.1, which ensures that at Y = 0, the lower surfaces of the hexahedron or pentahedron do not overlap each other incorrectly. Choose tab "File" and "New" to create the model in line mode.



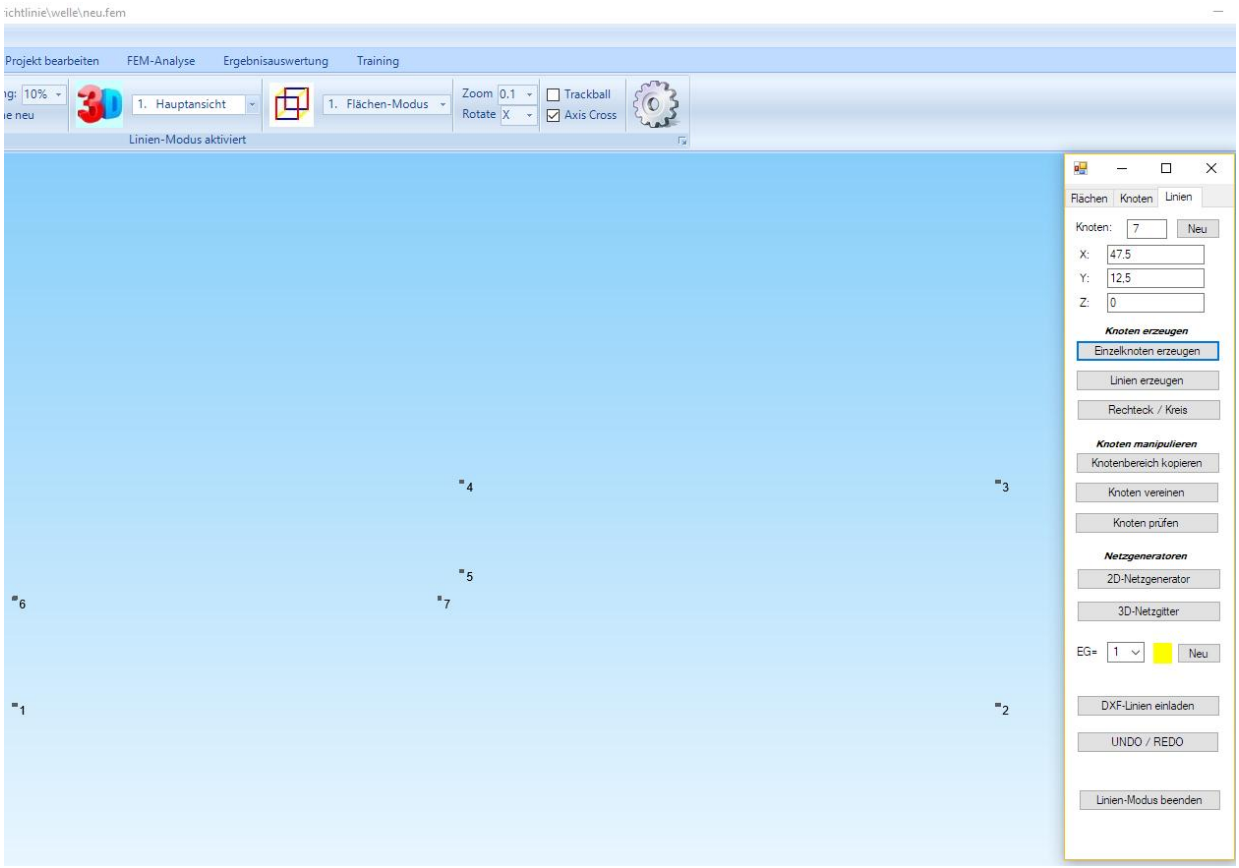
Create nodes

First, 7 nodes must be entered. On the Line Mode tab, click “New” for Node 1 with the coordinates X = 0, Y = 0.1, Z = 0. With menu "Create single node" create node 1.

Create the remaining 6 nodes in the same way:

Node 2 (110 / 0,1 / 0)
Node 3 (110/25/0)
Node 4 (50/25/0)
Node 5 (50/15/0)
Node 6 (0 / 12.5 / 0)
Node 7 (47.5 / 12.5 / 0)

Now all nodes should be visible on the screen

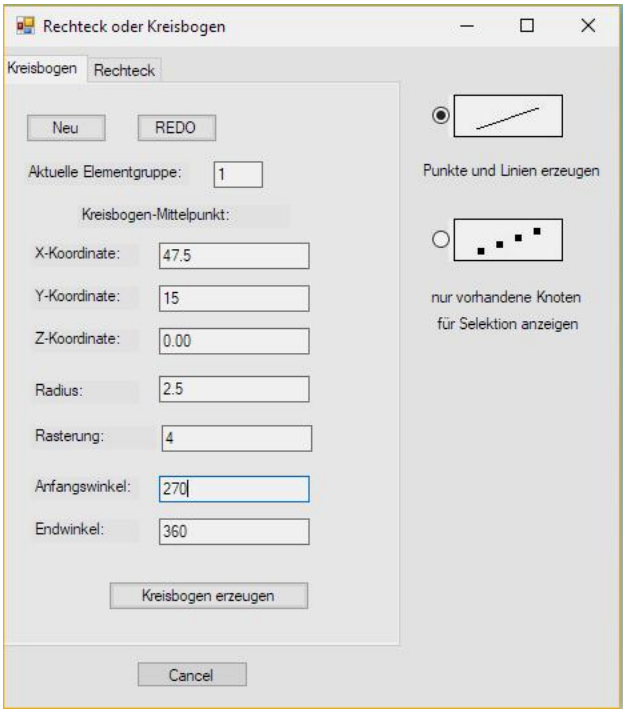


Save node model

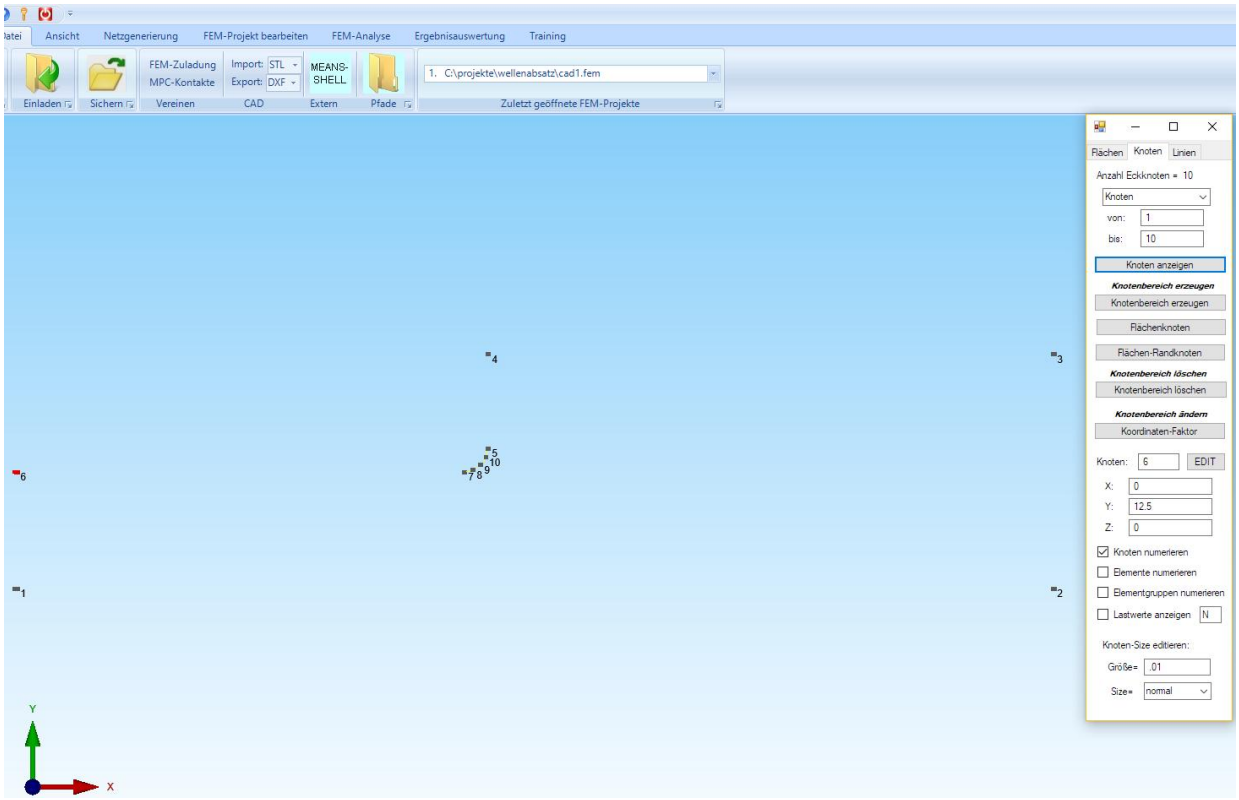
Save the model in the directory C:\ Projects\shaft\cad1.fem or any other directory so that the node model can be loaded again at any time.

Create arc

This is followed by the entry of the circular arc with the radius 2.5 mm, select the menu "Rectangle / circle" and enter the center point 47.5 / 15/0, the radius = 2.5, the screening = 4 and the start angle 270 and end angle 360 and select "Create arc" menu.



There are now 10 nodes and 4 lines:

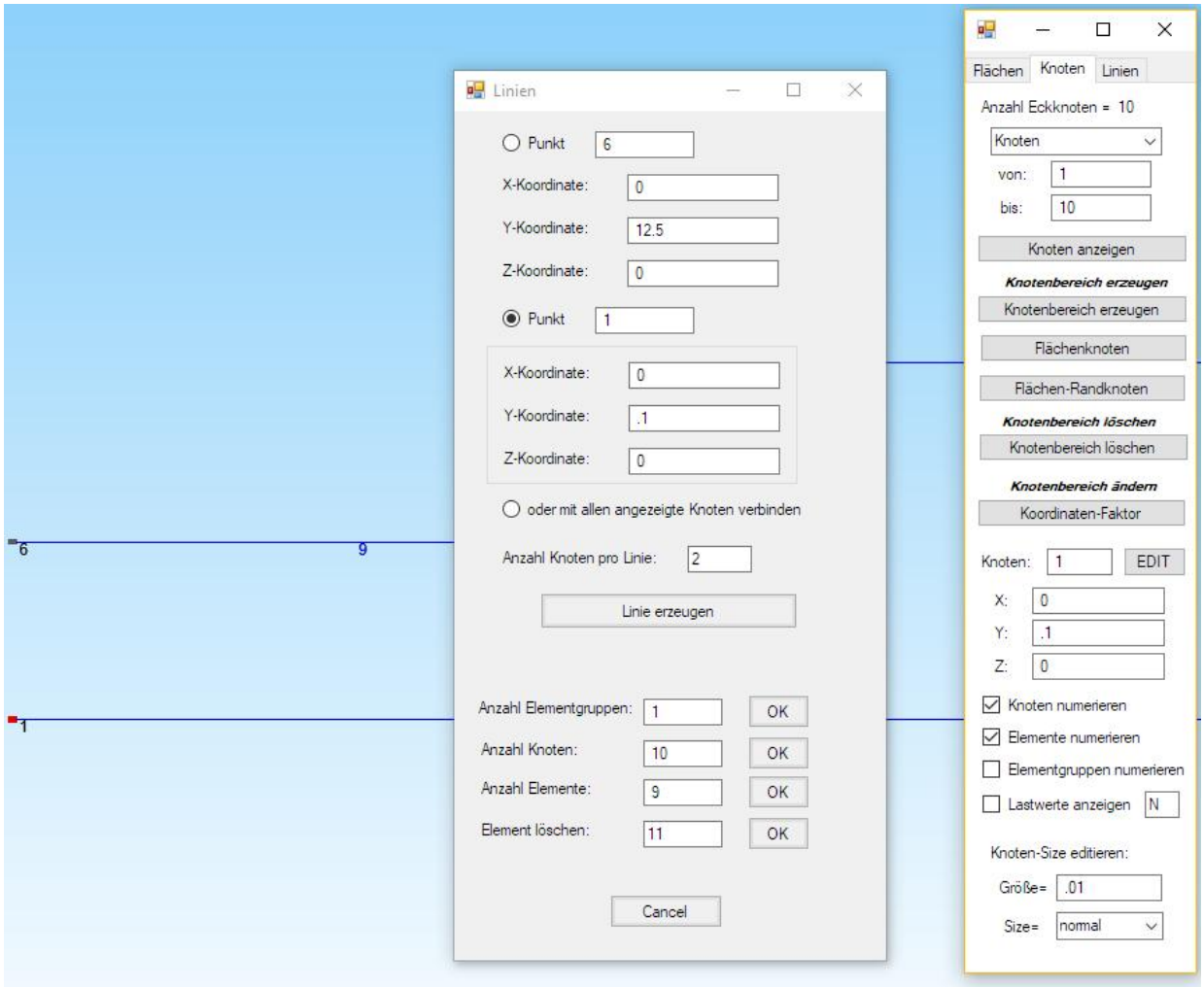


I

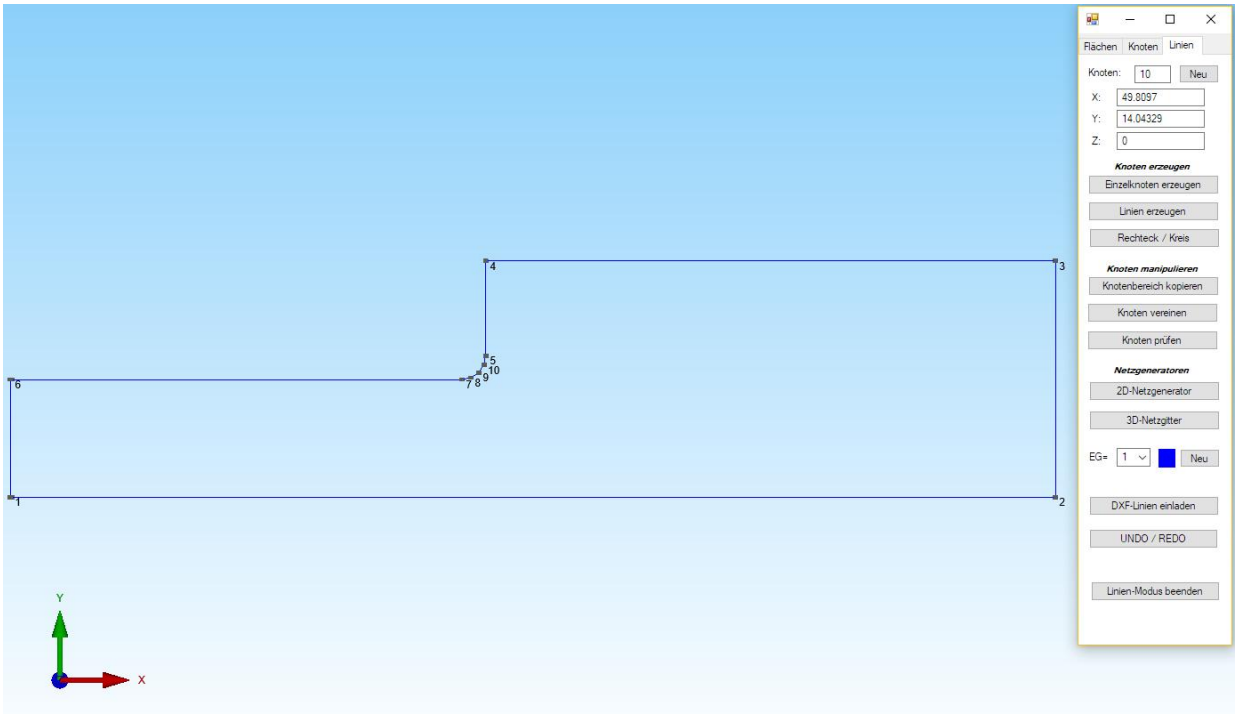
In line mode with menu "Create lines" you have to create 6 lines:
Line 1: connect node 1 to node 2
Line 2: connect node 2 to node 3
Line 3: Connect node 3 to node 4
Line 4: Connect node 4 to node 5
Line 5: Connect node 6 to node 7
Line 6: Connect node 1 to node 6

Enter line 6

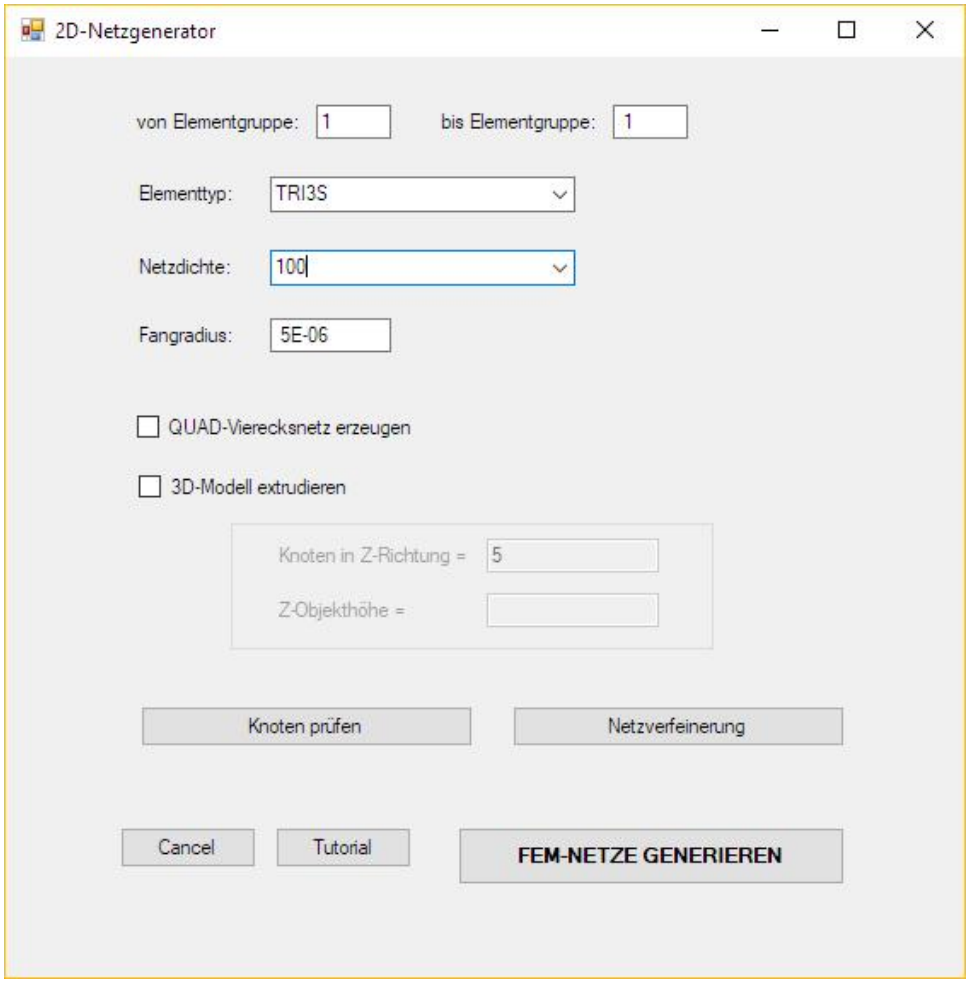
Activate the upper option "Point" and double-click on the node 6 on the model so that its coordinates are displayed in the Dialogbox, then activate the lower option "Point" and click on node 1. Finally, with the menu "Line generate" create line 1-6. It may also be useful to enter the node and element numbering.



The border of the half 2D shaft now consists of 10 nodes and 11 lines and a triangle mesh can be created.

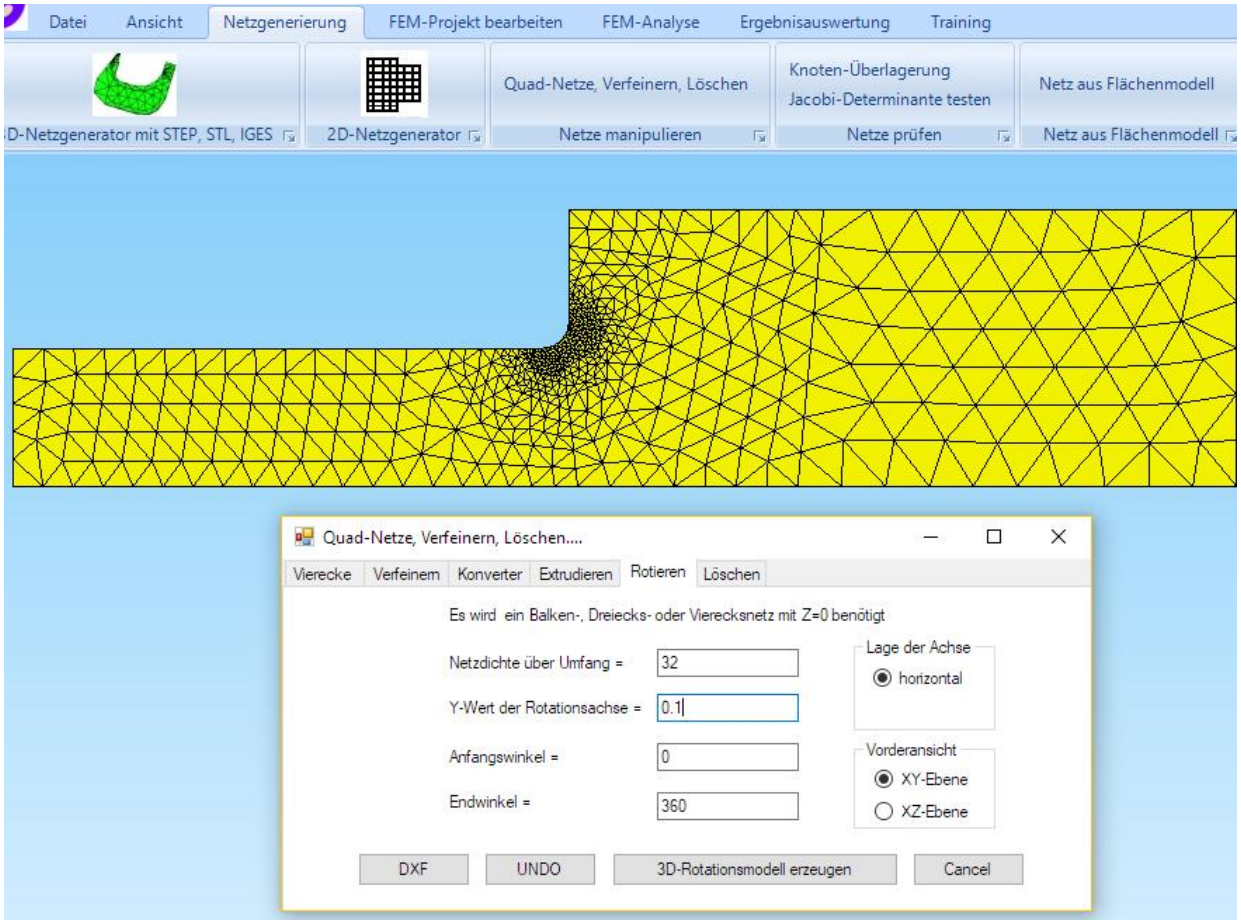


Select either the "Mesh Generation" tab or the "2D Mesh Generator" line mode menu and generate with a mesh density = 100 a not too fine triangle mesh of 1028 TRI3S elements and 574 nodes for the next step.

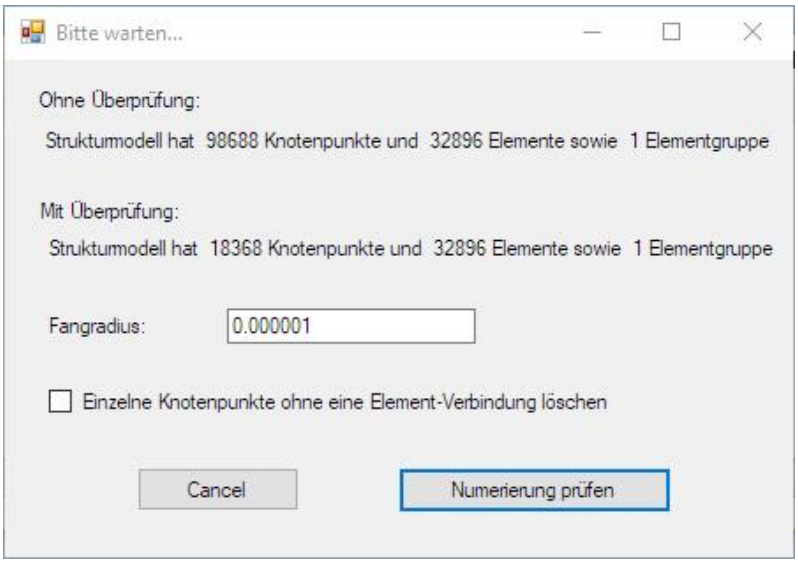


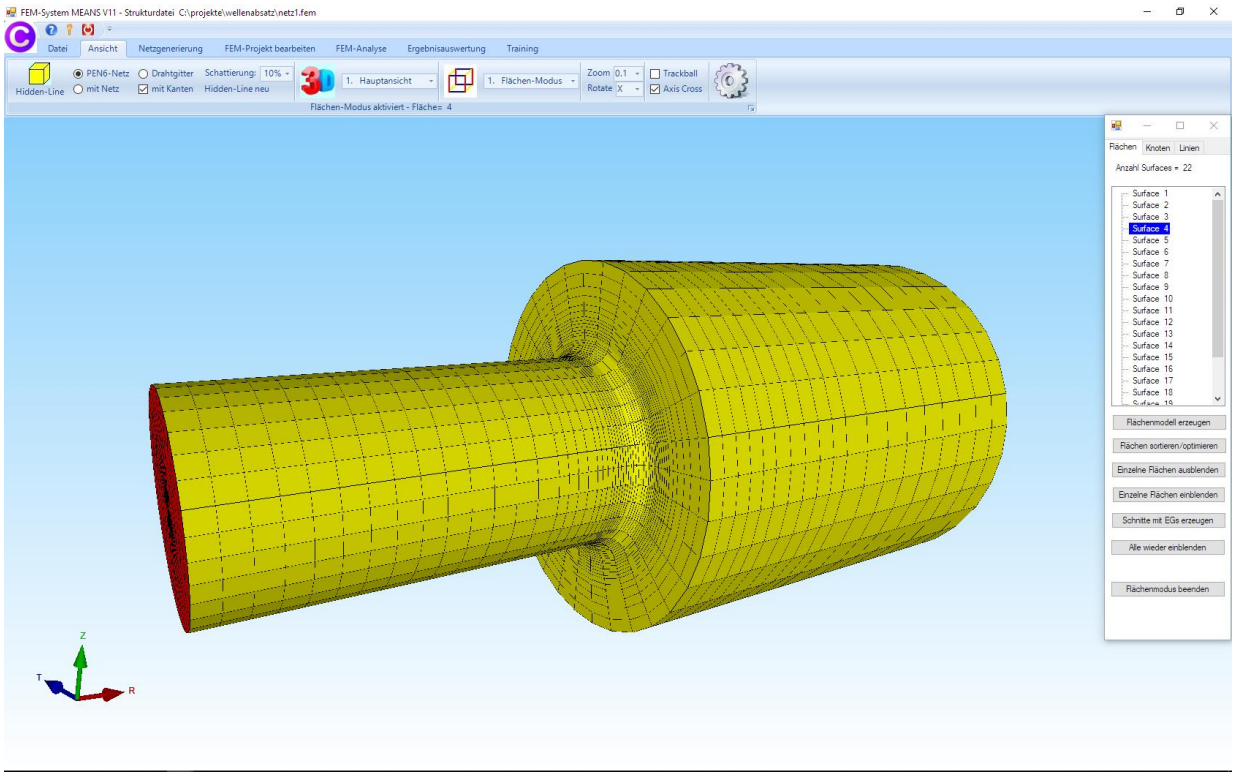
Create 3D volume mesh

This 2D mesh now serves as a template to generate a rotation mesh of pentahedron elements. Select the "Mesh Generation" tab and the "Quad Mesh, Refine, Delete" menu and in the next dialog box the "Rotate" tab to generate a rotation mesh.

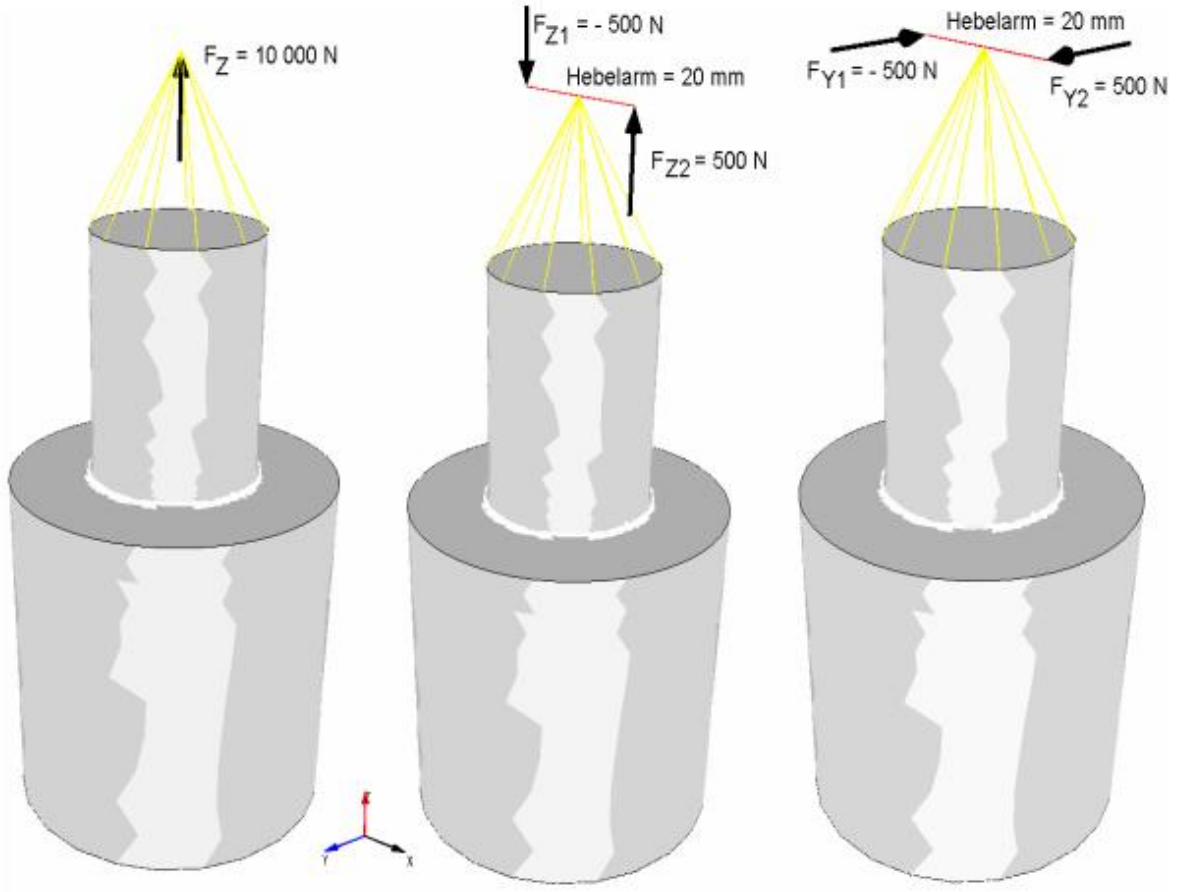


Select a mesh density of 32 and set the Y-value of the rotation axis to 0.0 to create a model of 32896 PEN6 elements and 18368 nodes. This is followed by a review with a new hidden line and surface model.





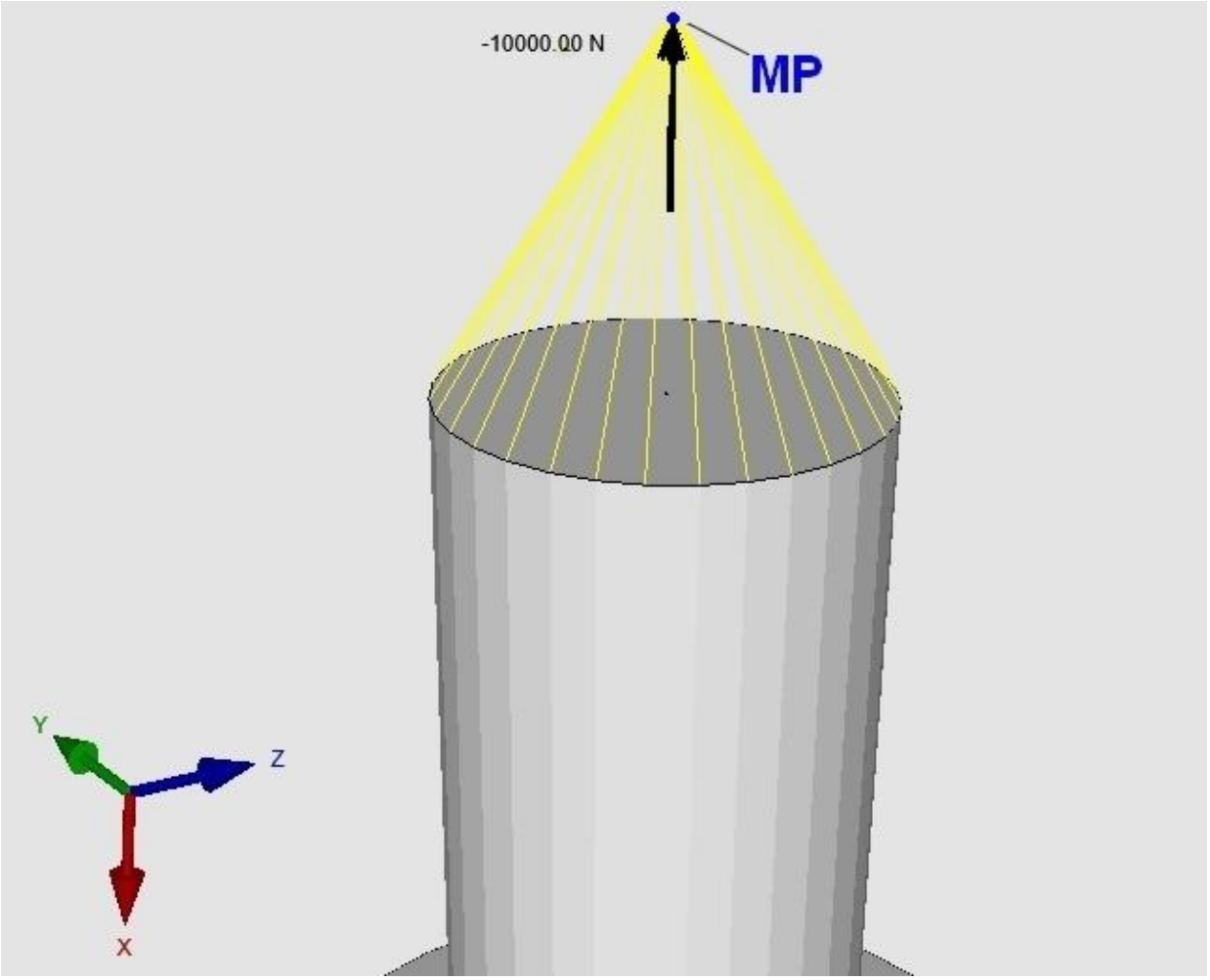
Now enter a tensile force, a bending moment and a torsional moment using MPC beam elements (yellow lines) in the line mode.



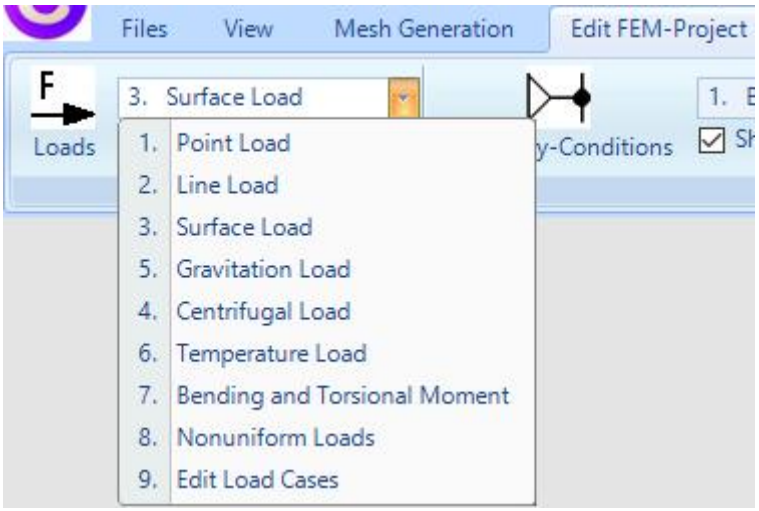
Tensile Load 10000 N Bending moment 10 000 Nmm Torsional moment 10000 Nmm

Create a Axial Load

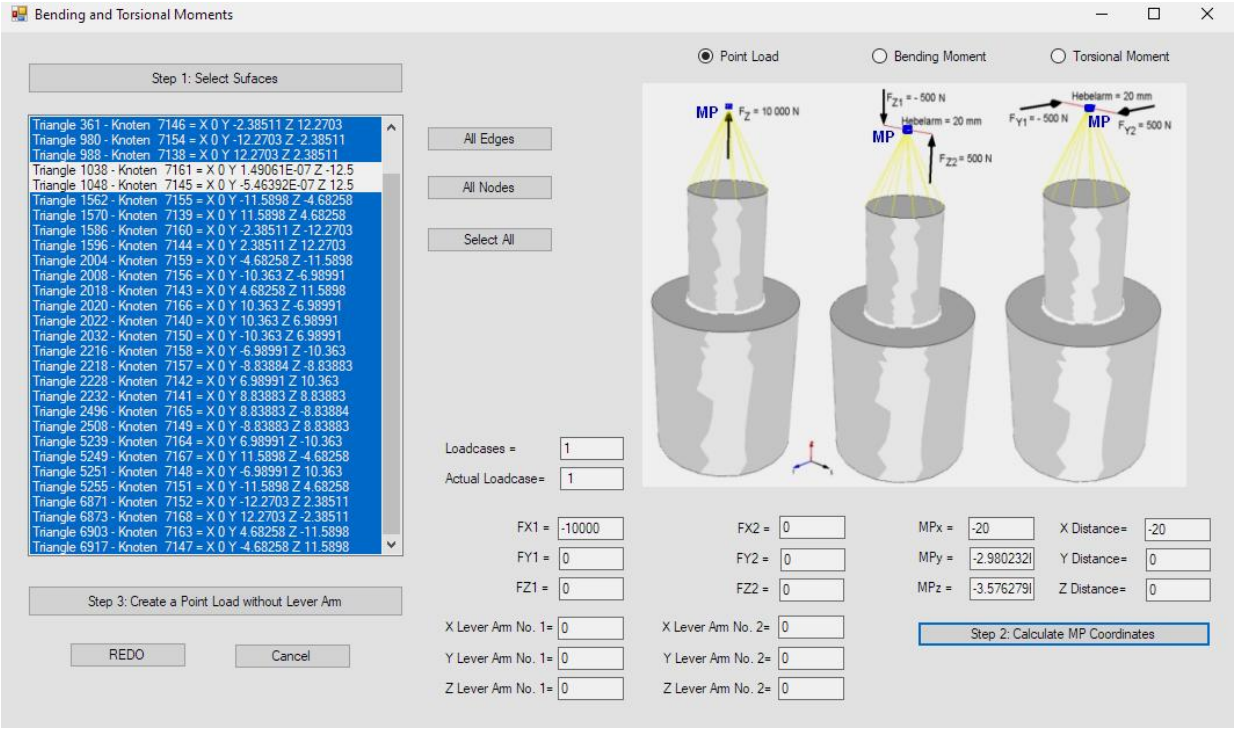
Select the "Edit FEM project" and "Bending and torsional moment" tabs and define an Axial Point Load in the X direction of -10000 N that is not loaded to an FEM node but to an outside node point MP(-20, 0, 0).



Select the "Edit FEM Project" tab and select the "Bending and Torsional Moment" load type from the drop-down menu.



In the new dialog box, first select the "Step 1: Select Surfaces" menu and click on surface 4 to list all nodes of this surface.



Define nodal load

Activate "Point Load" and enter only FX1= -10000 N. Since neither a force couple nor a lever arm occur, all other fields must be zero.

Calculate MP node

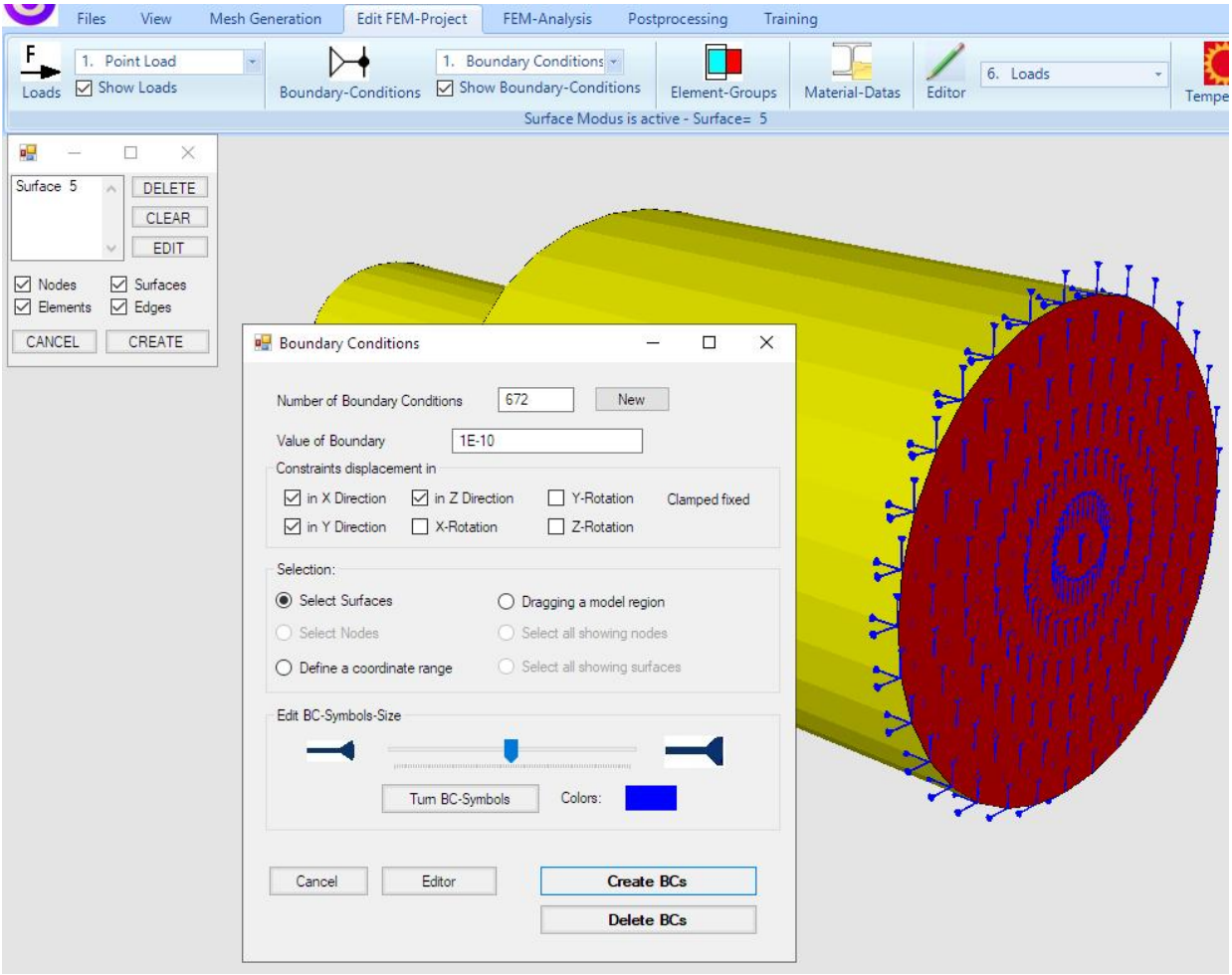
Choose menu "Select All" to select all nodes and deactivate the two center points of the surface with the STRG key. Enter the MP distance in the X direction = -20 mm and calculate the MP coordinates using the "Step 2: Calculate MP with distance" menu.

Generate a Point Load without a lever arm

Now select "Step 3: Create a Point Load without lever arm" to create the Axial Load in the X-direction outside the FEM mesh.

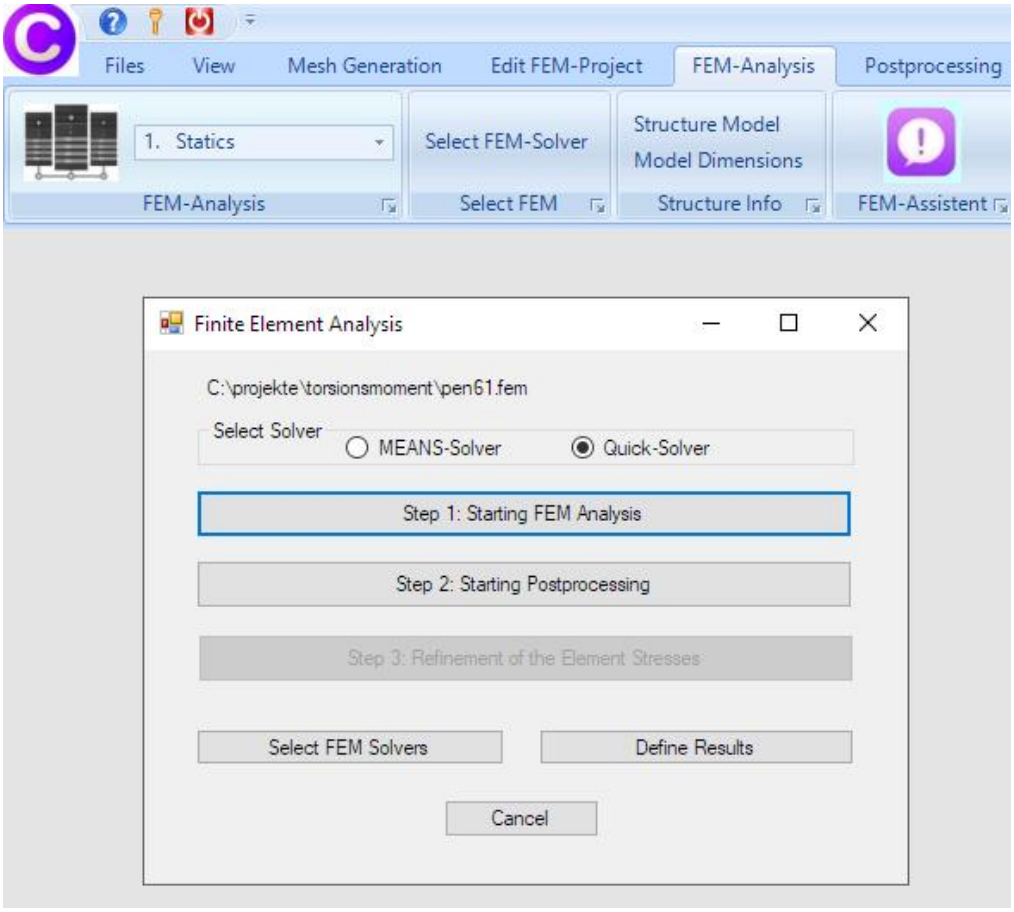
Create the Boundary Conditions

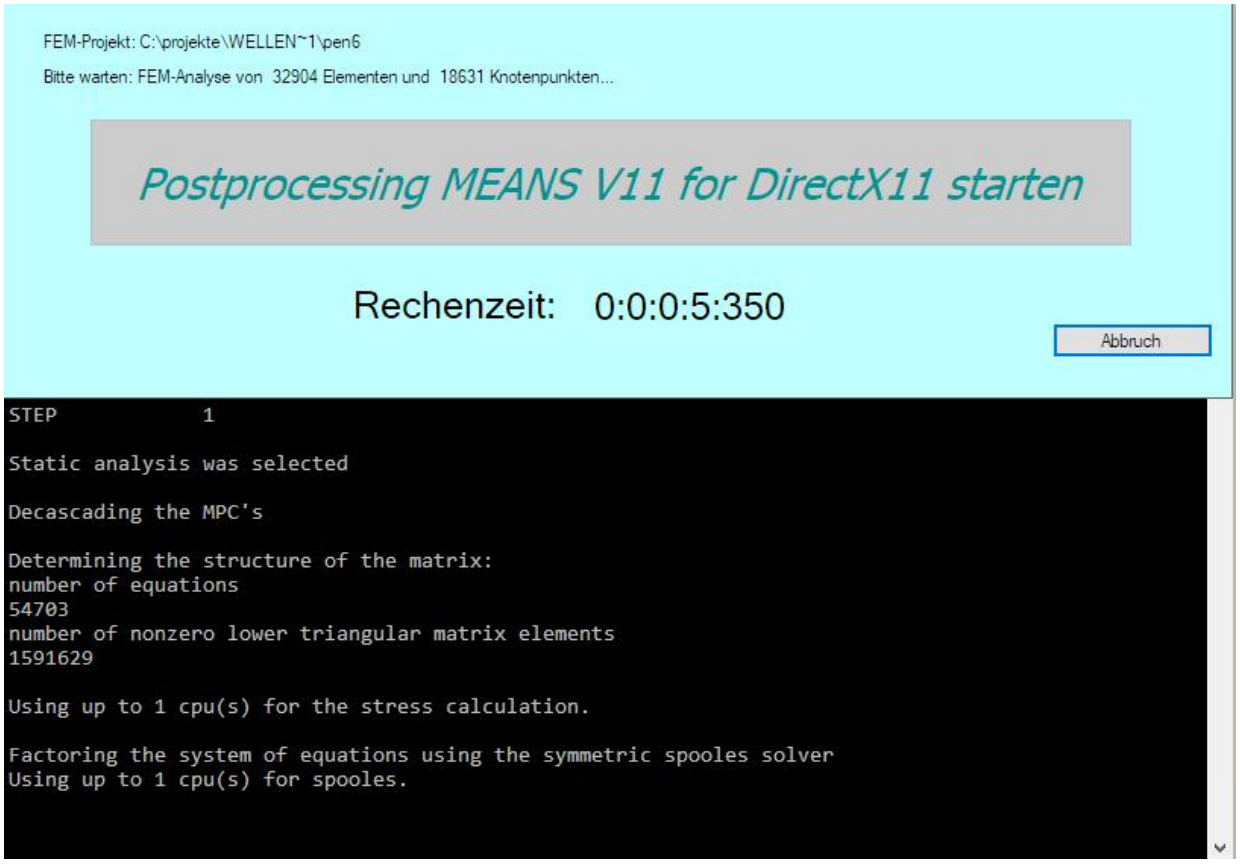
First, the lower surface is clamped with Surface 5 in the x, y, and z directions.
To do this, select the “Edit FEM project” tab and “Boundary conditions” and choose the X-, Y- and Z Direction and “Select Surfaces” as well as “Create RBs” in the RB dialog box and click on the Surface 5. This is displayed in the select box, create the boundary conditions there with “Create”.



FEM-Analysis

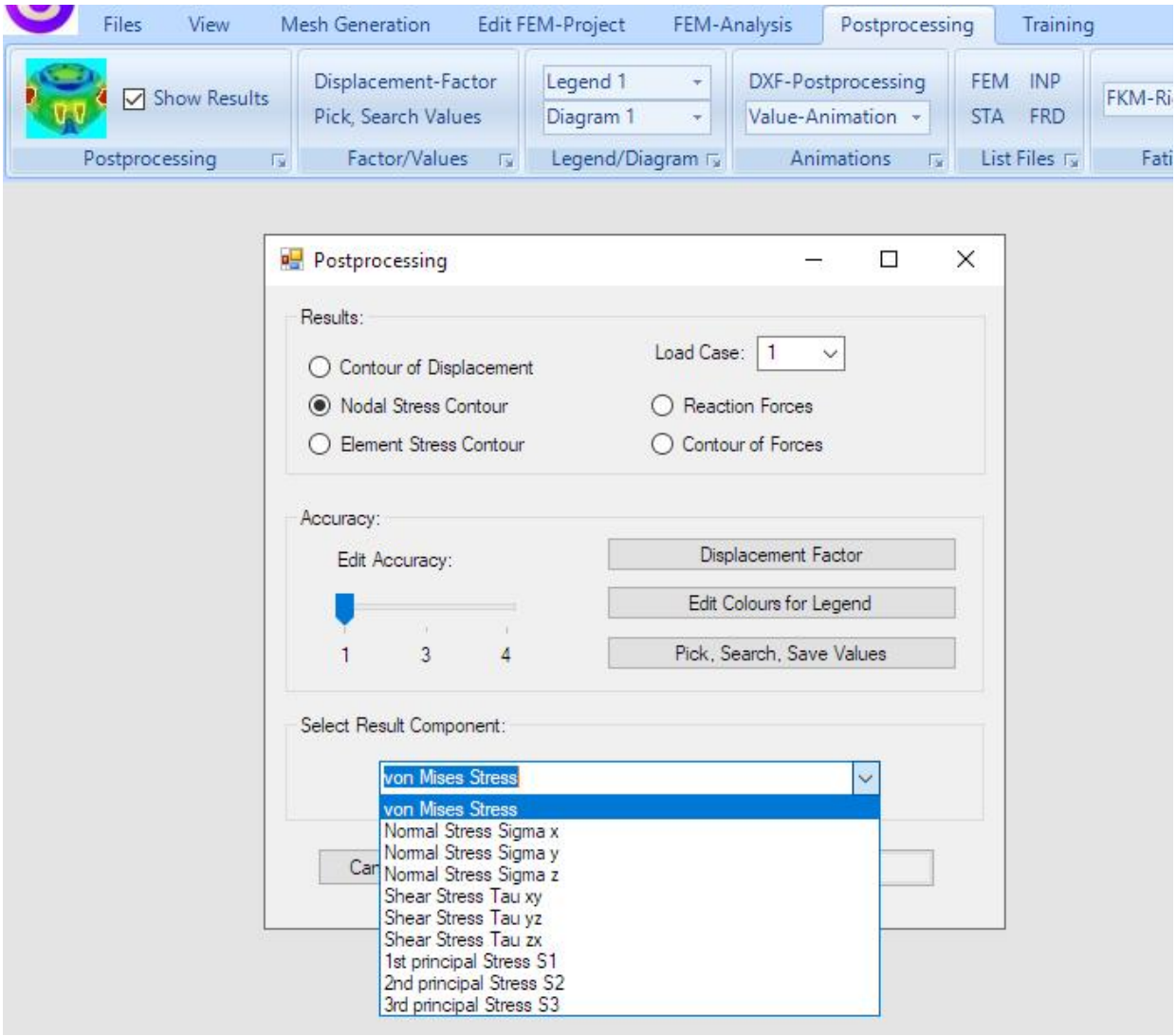
Select the “FEM Analysis” tab and “Statics” to calculate the displacements and nodal stresses with the quick solver.
In the quick solver dialogbox, select “C3D8 (8-node linear isoparametric element)” for a quick FEM Analysis and the button “Start FEM solver with INP interface”.
After closing MEANS V12, the FEM analysis is started in a Windows window and ends after a few seconds with a beep.
Then the button "Start Postprocessing MEANS V12 for DirectX11" can be clicked to display the FEM model again in MEANS V12.


Files
View
Mesh Generation
Edit FEM-Project
FEM-Analysis
Postprocessing
1. Statics
FEM-Analysis
Select FEM-Solver
Select FEM
Structure Model
Model Dimensions
Structure Info
FEM-Assistent
Finite Element Analysis
C:\projekte\torsionsmoment\pen61.fem
Select Solver
MEANS-Solver
Quick-Solver
Step 1: Starting FEM Analysis
Step 2: Starting Postprocessing
Step 3: Refinement of the Element Stresses
Select FEM Solvers
Define Results
Cancel


FEM-Projekt: C:\projekte\WELLEN~1\pen6
Bitte warten: FEM-Analyse von 32904 Elementen und 18631 Knotenpunkten...
Postprocessing MEANS V11 for DirectX11 starten
Rechenzeit: 0:0:0:5:350
Abbruch
STEP 1
Static analysis was selected
Decascading the MPC's
Determining the structure of the matrix:
number of equations
54703
number of nonzero lower triangular matrix elements
1591629
Using up to 1 cpu(s) for the stress calculation.
Factoring the system of equations using the symmetric spooles solver
Using up to 1 cpu(s) for spooles.

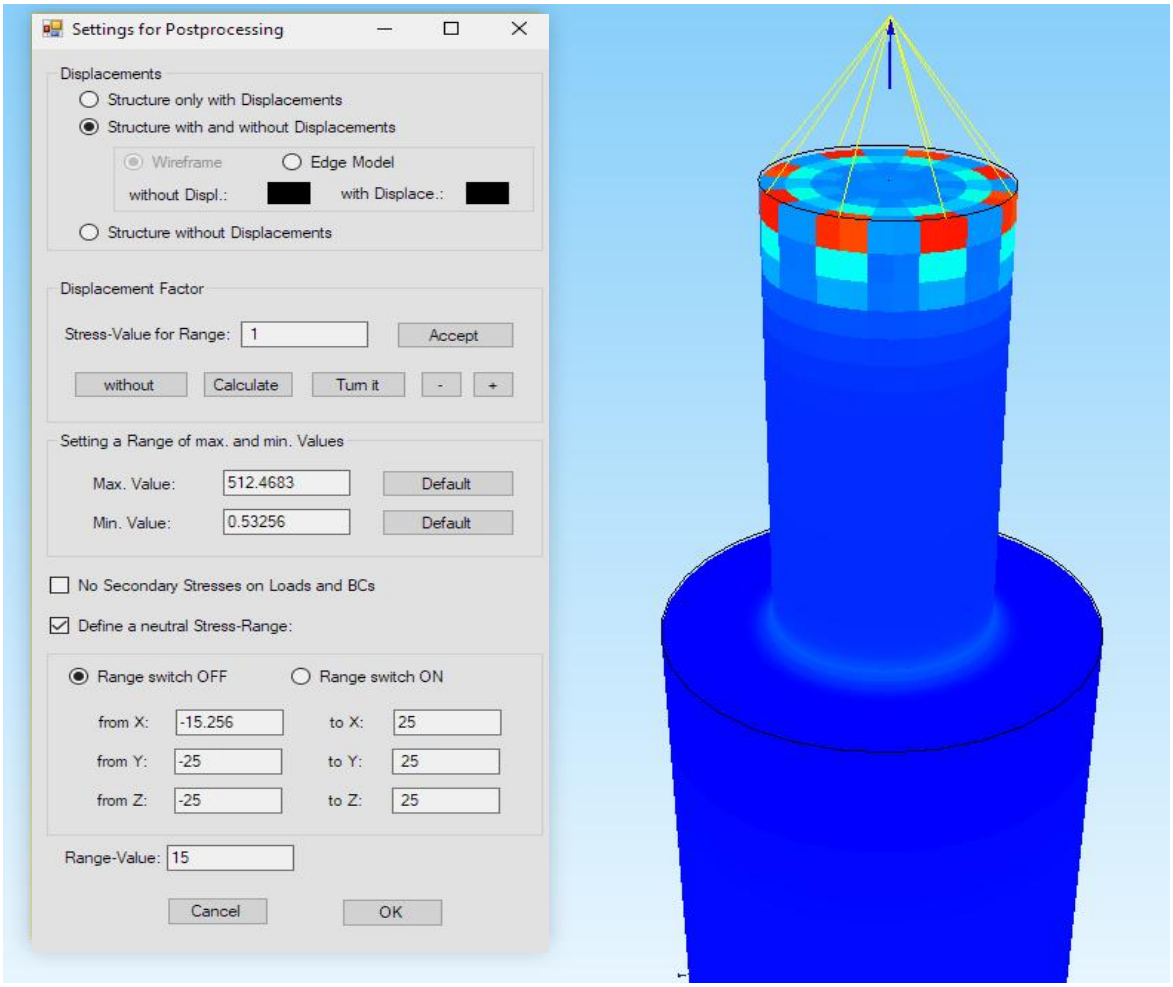
Postprocessing

Select the "Results evaluation" tab and the icon  to evaluate the displacements and stresses graphically.



Define neutral Stress Range

Select “Element Stress Contour“ to display the v.Mises Element Stresses. Since only the element stresses in the shaft shoulder are of interest, the higher stresses on the beam elements can be masked out either with a smaller maximum value or with a neutral stress area.
To do this, select “Displacement Factor” and define the following neutral stress range with an X range of -15 to +25 and a substitute value of 15:



Exact result according to Roloff-Matek:

F= 10000 [N]
D= 50 [mm]
d= 25 [mm]
r= 2,5 [mm]

$$\sigma_n = \frac{F}{\pi * \frac{d^2}{4}} \quad \left[\frac{N}{mm^2}\right]$$

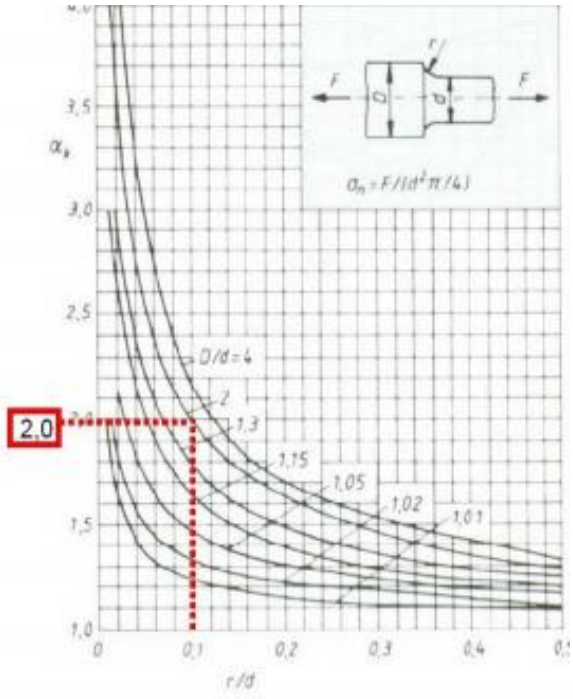
$$= \frac{10000}{\pi * \frac{25^2}{4}}$$

$$= 20{,}37 \quad \left[\frac{N}{mm^2}\right]$$

$$\sigma_{max} = \sigma_n * \alpha_K \quad \left[\frac{N}{mm^2}\right]$$

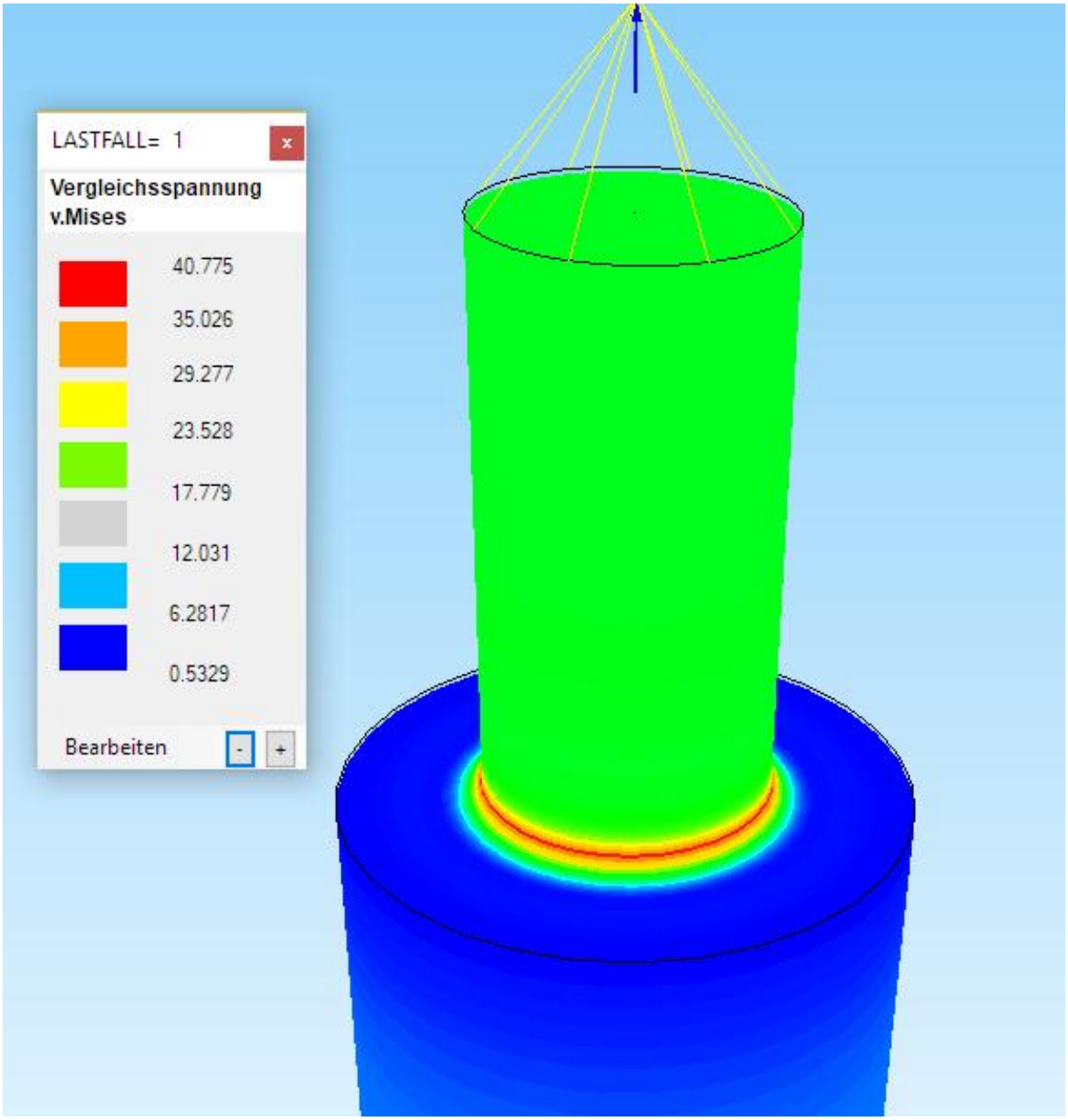
$$= 20{,}37 * 2$$

$$= 40{,}74 \quad \left[\frac{N}{mm^2}\right]$$



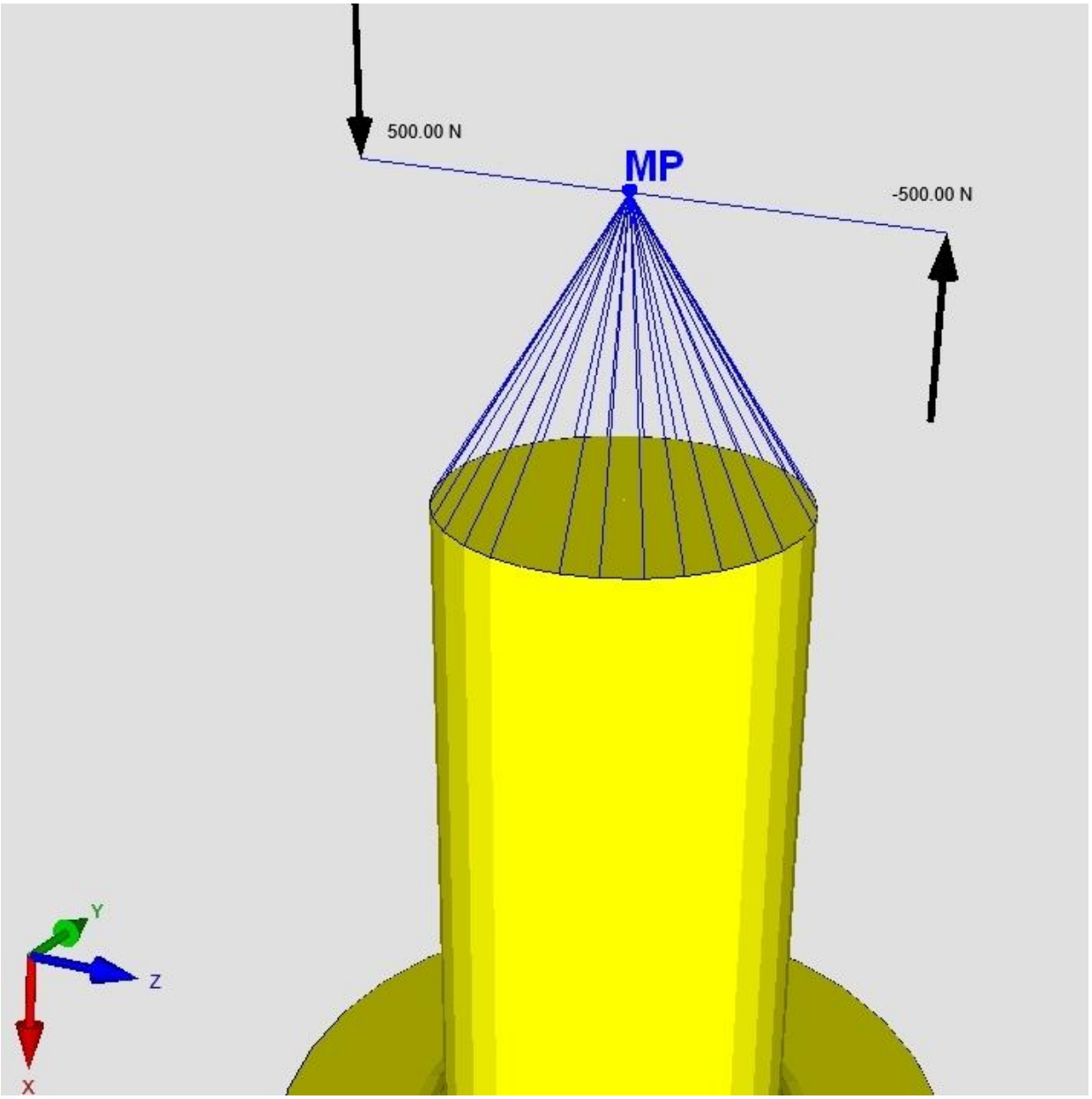
Quelle: Roloff/ Matek Maschinenelemente; 15. Auflage; August 2001; Vieweg Verlag

The maximum v.Mises Stresses is 40.77 N/mm², this value also agrees well with the exact calculation according to Roloff-Matek with 40.74 N/mm².



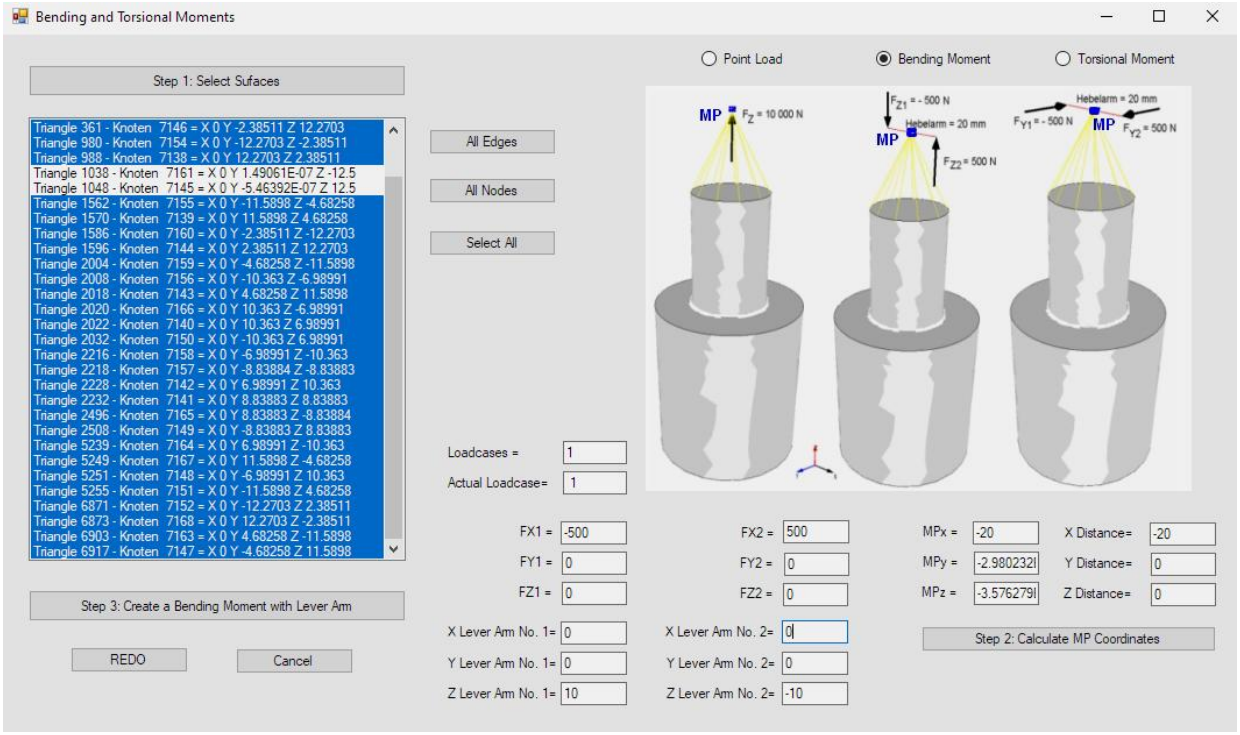
Create a Bending Moment

The bending moment of 10,000 Nmm is defined as a force couple F_{X1} = 500 N and F_{X2} = -500 N with a lever arm of 20 mm in Z direction.



Select the "Edit FEM Project" tab and select the "Bending and Torsional Moment" load type from the drop-down menu.

In the new dialog box, first select the "Step 1: Select Surfaces" menu and click on surface 4 to list all nodes of this surface.



Define Bending Moment

Activate "Bending Moment“ and enter F_{X1}= -500 N and F_{X2} = 500 N with a Lever Arm in z direction No. 1= 10 mm and No. 2 = -10 mm, all other fields must be zero.

Calculate MP node

Choose menu "Select All" to select all nodes and deactivate the two center points of the surface with the STRG key. Enter the MP distance in the X direction = -20 mm and calculate the MP coordinates using the "Step 2: Calculate MP with distance" menu.

Generate Bending Moment with lever arm

Now select "Step 3: Create a Bending Moment with lever arm" to create the Bending Moment.

Exact result according to Roloff-Matek:

M= 10000 [N*mm]
D= 50 [mm]
d= 25 [mm]
r= 2,5 [mm]

$$\sigma_n = \frac{M}{\pi * \frac{d^3}{32}} \quad \left[\frac{N}{mm^2}\right]$$

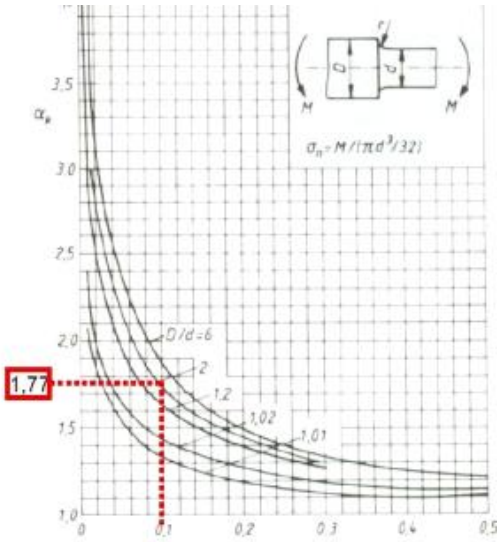
$$= \frac{10000}{\pi * \frac{25^3}{32}}$$

$$= 6{,}52 \quad \left[\frac{N}{mm^2}\right]$$

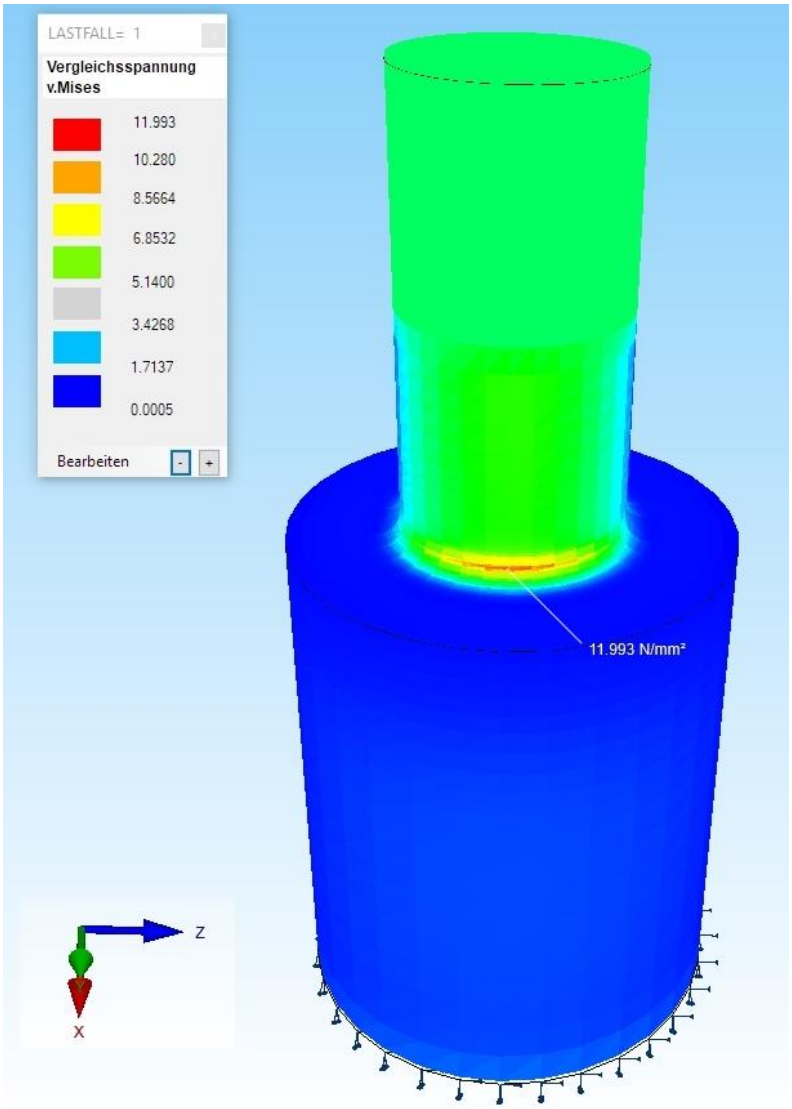
$$\sigma_{max} = \sigma_n * \alpha_K \quad \left[\frac{N}{mm^2}\right]$$

$$= 6{,}52 * 1{,}76$$

$$= 11{,}47 \quad \left[\frac{N}{mm^2}\right]$$

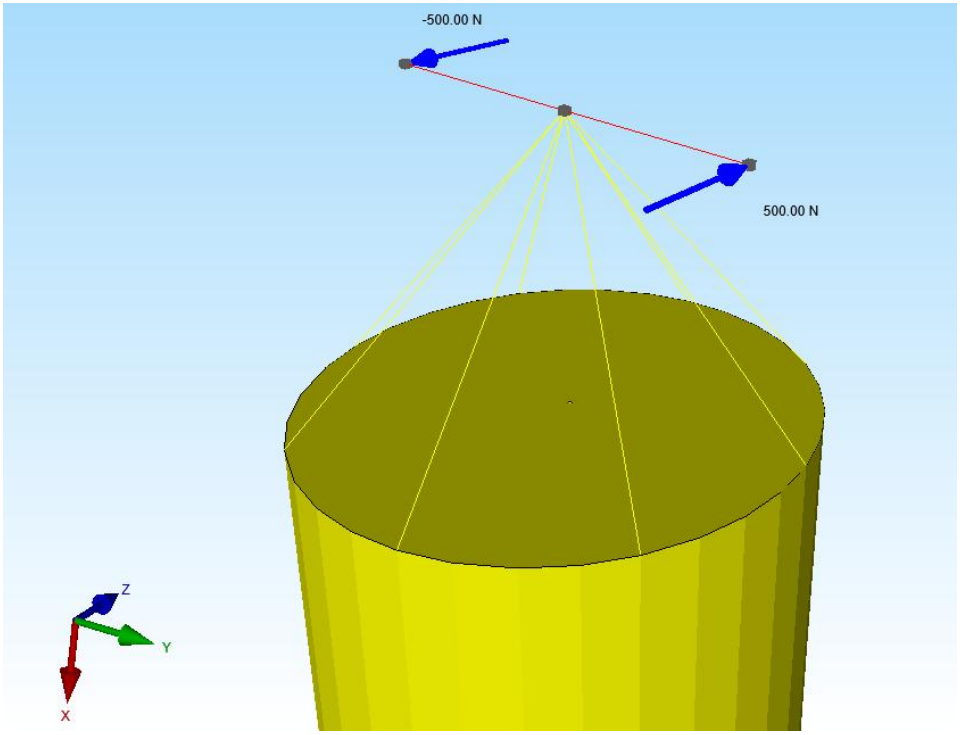


The v.Mises Stress is 11.99 N/mm², this value also agrees well with the exact calculation according to Roloff-Matek with 11.47 N/mm².

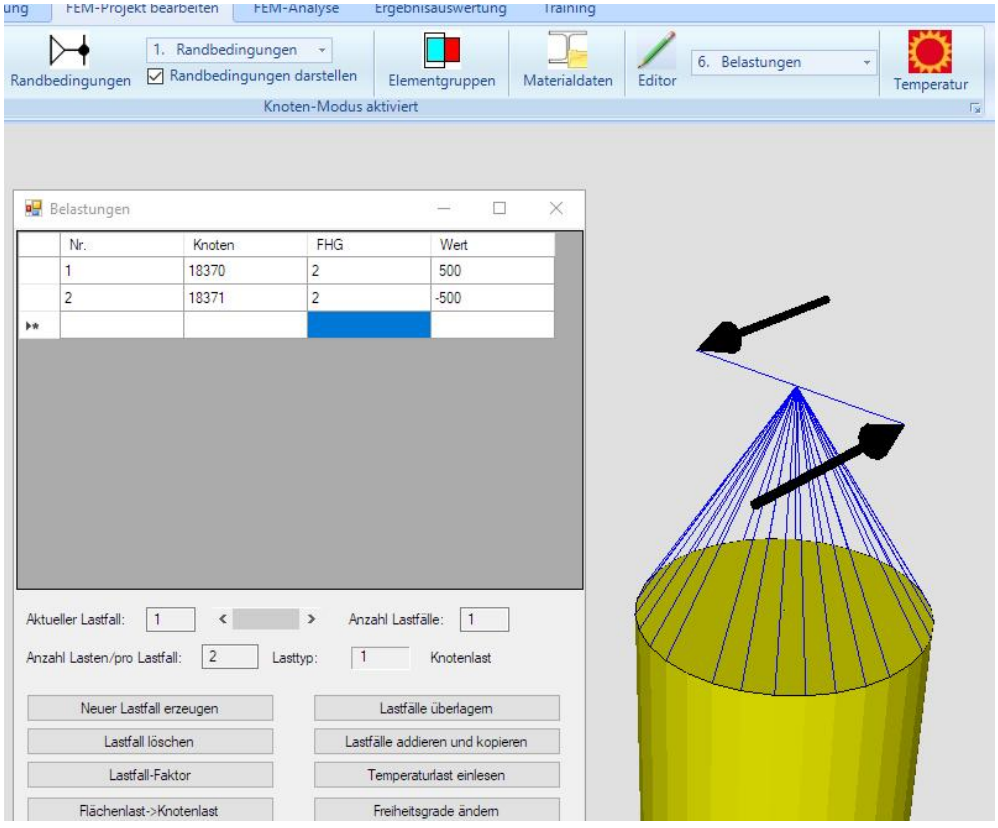


Create a Torsional Moment

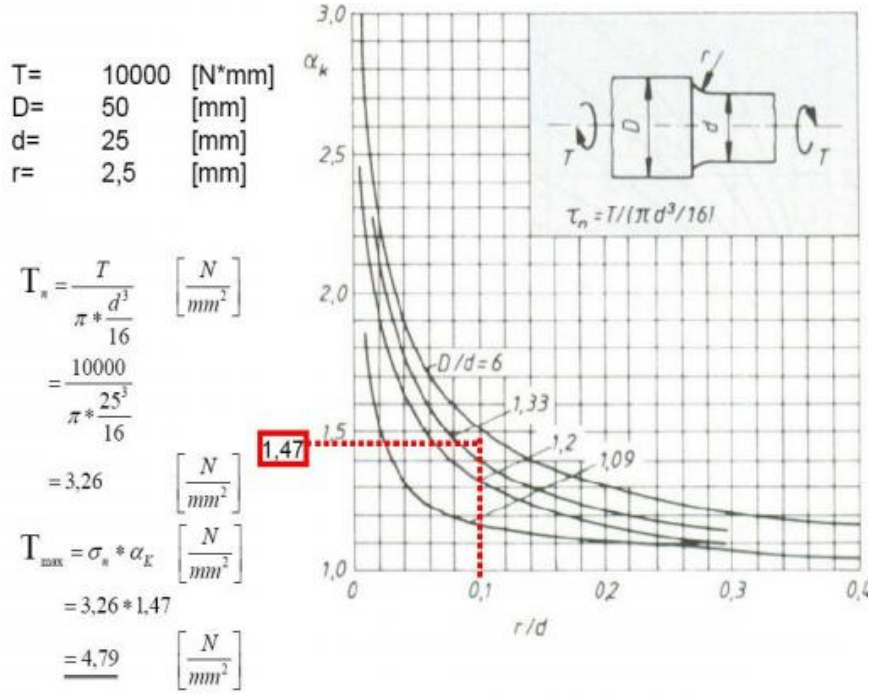
The torsional moment of 10000 Nmm can also be replaced by a pair of forces with two equal forces FZ18370 = 500 N and FZ18371 = - 500 N and a lever arm of 20 mm.



You can easily generate this load from the previous load by changing the degree of freedom in the X-direction FHG = 1 to the Y-direction FHG = 2 with “Editor”.



Exact result according to Roloff-Matek:



The v.Mises Stresses is 5.08 N/mm², this value also agrees well with the exact calculation according to Roloff-Matek with 4.79 N/mm².