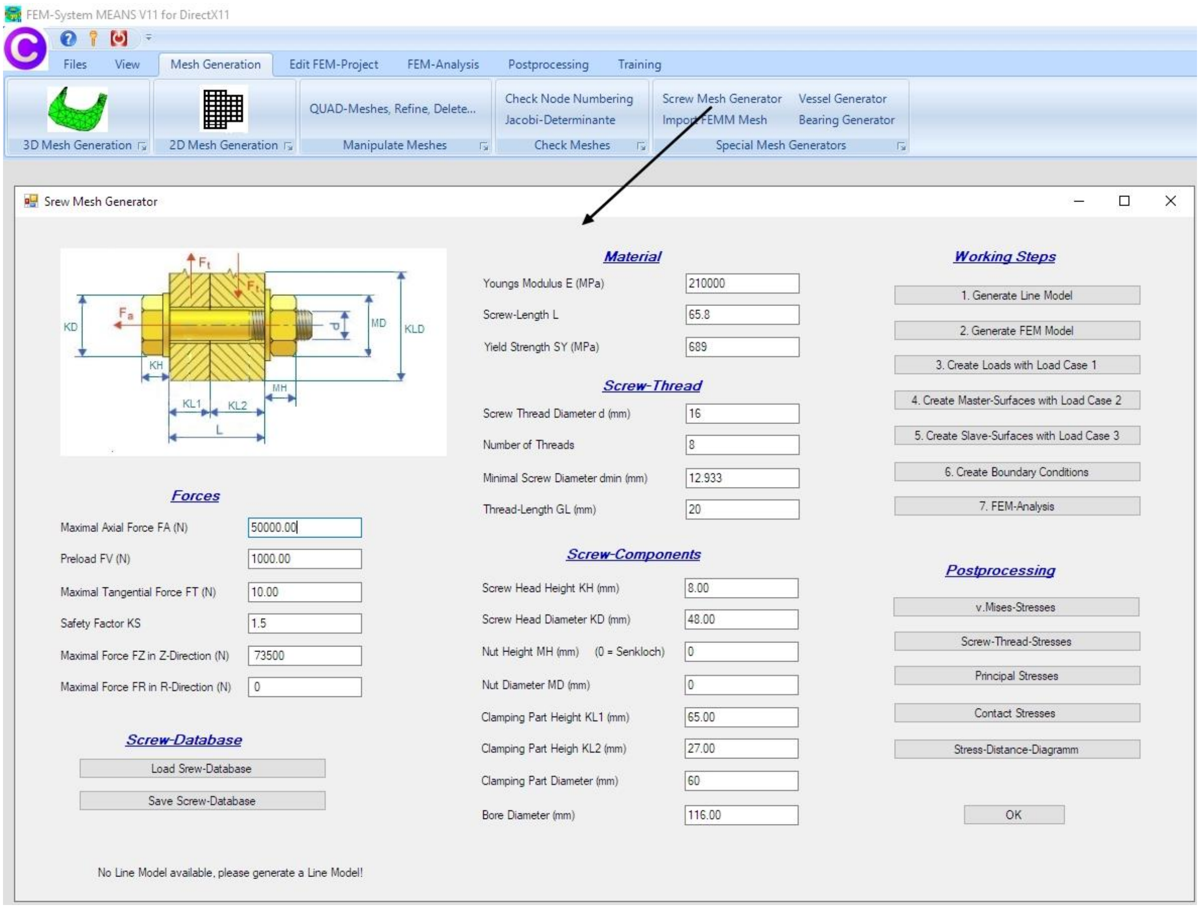
Part 17: Screw Mesh Generator of FEM-System MEANS V11

With the new screw mesh generator, by entering a few data, screw models can be meshed fully automatically and calculated using a contact analysis. Since most screw models generally have a axisymmetrically geometry and load, the very precise quatratic 2D axisymmetrical finite elements can be used, which require considerably less computing and working time compared to the 3D-Screw-Thread models.

Start the Screw Mesh Generator

The screw mesh generator is started with the register "Mesh Generation" and the "Screw Mesh Generator" menu.



Forces

Only axisymmetrical loads can be calculated, but there is the option of converting a moderate 2D rotation mesh into a 3D solid volume mesh for non-symmetrical loads.

Maximale Axial Force FA (N)

Preload FV (N)

Maximal Tangential Force FT (N)

Required Safety Factor KS

Maximal Force in the Z direction FZ = (FA - FV) * KS

Maximal Force in the R direction FR = FT * KS

Screw-Thread

Screw-Thread Diameter d (mm)

Number of Threads (0 = no thread is created)

Minimal Screw Diameter dmin (mm)

Thread-Length GL (mm)

Screw-Components

KH = Screw-Head Height

KD = Screw-Head Diameter

MH = Nut Height

MD = Nut Diameter (0 = sinke hole without nut is created)

KL1 = Clamping Part Height of upper component

KL2 = Clamping Part Height of lower component

Working Steps

Perform the following 7 working steps one after the other to generate the 2D FEM screw model fully automatically in a few seconds.

1. Generate Line Model
2. Generate FEM model
3. Create Loads with Load Case 1
4. Create Master-Surfaces with Load Case 2
5. Create Slave-Surfaces with Load Case 3
6. Create Boundary Conditions
7. FEM-Analysis

Postprocessing

After the FEM analysis, the following stresses can be evaluated and displayed in a stress-displacement diagram:

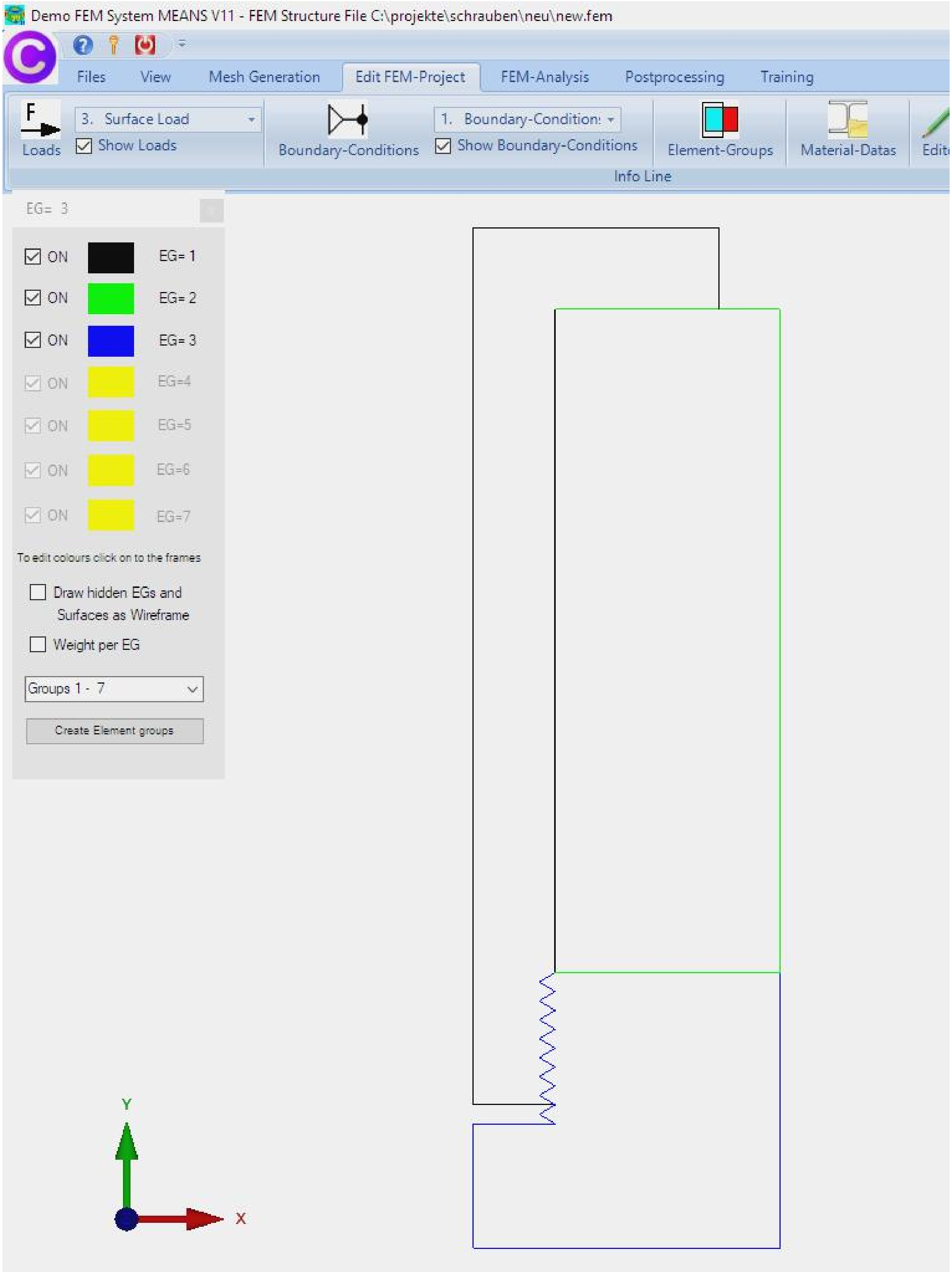
1. v.Mises Stresses
2. Screw-Threads Stresses
3. Principal Stresses
4. Contact Stresses
5. Stress-Distance-Diagram

Screw Database

All screw datas can be saved in the screw database and loaded again.

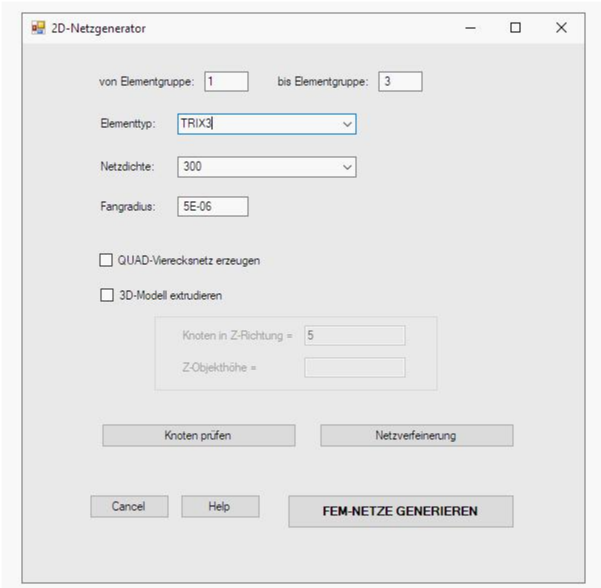
1. Create a Line Model

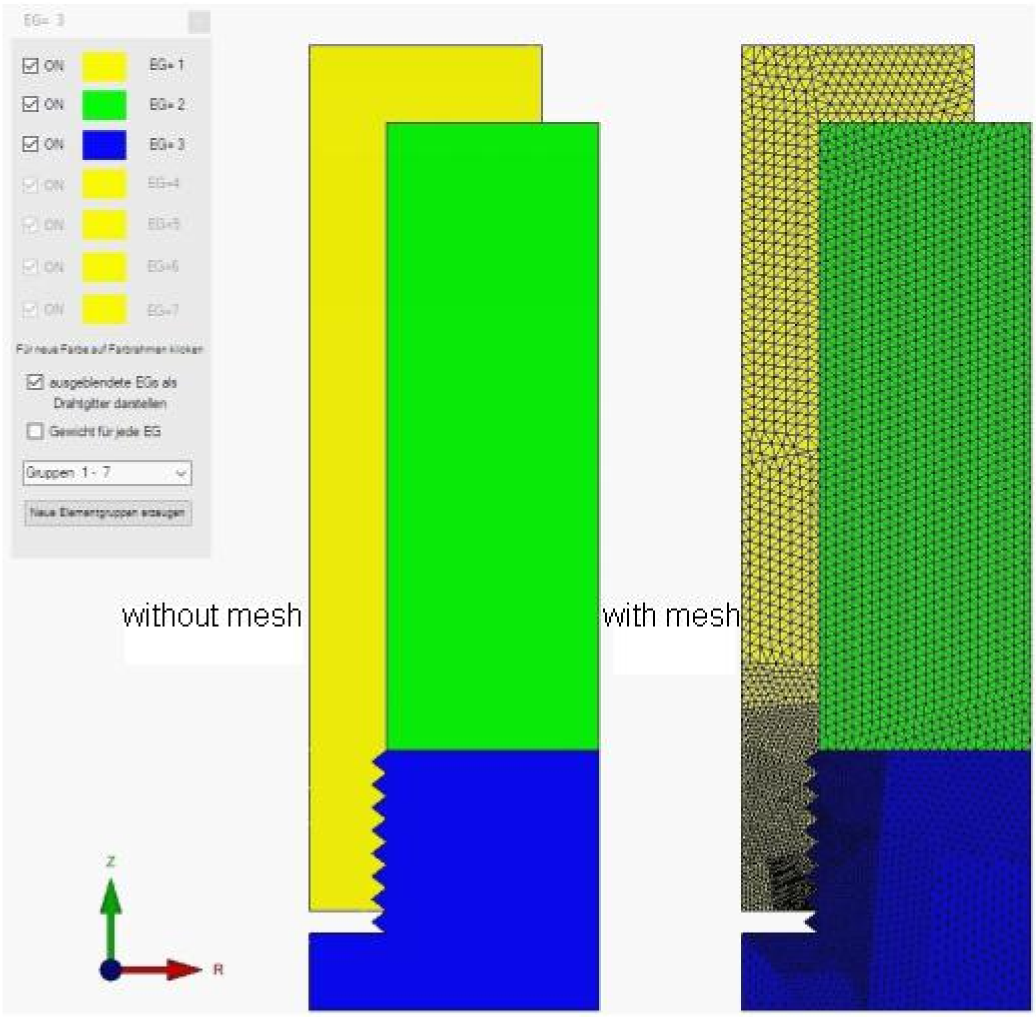
A beam line model with 3 element groups consisting of screw, thread and clamping parts is created.



2. Create a FEM Model

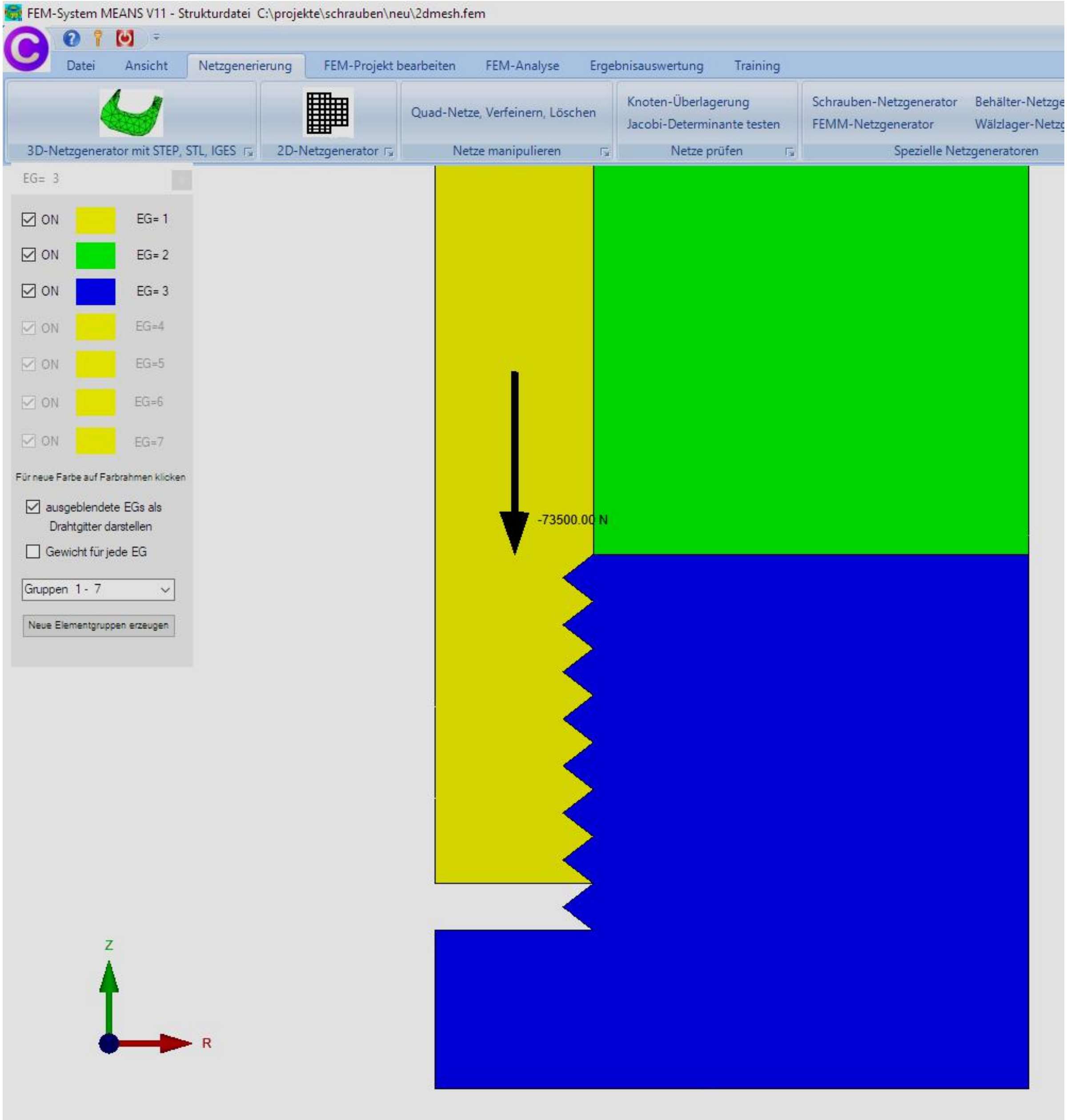
Mesh Generation follows the line model with element type “TRIX3” and the mesh density “300” for a FEM mesh with 9046 nodes, 4875 TRIX3 elements and 3 element groups. The material data are also created with the entered Youngs Modulus.





3. Generate Axial Force

The resulting axial force FZ in the Z direction is generated with the entered value of -73500 N in load case 1. The force acts directly on the thread, where clamping forces, preload and frictional forces. A tangential load in the R direction can also be calculated.



4. Master Contact Surface

The contact nodes between screw, thread and the clamping parts are determined fully automatically and summarized in load case 2.

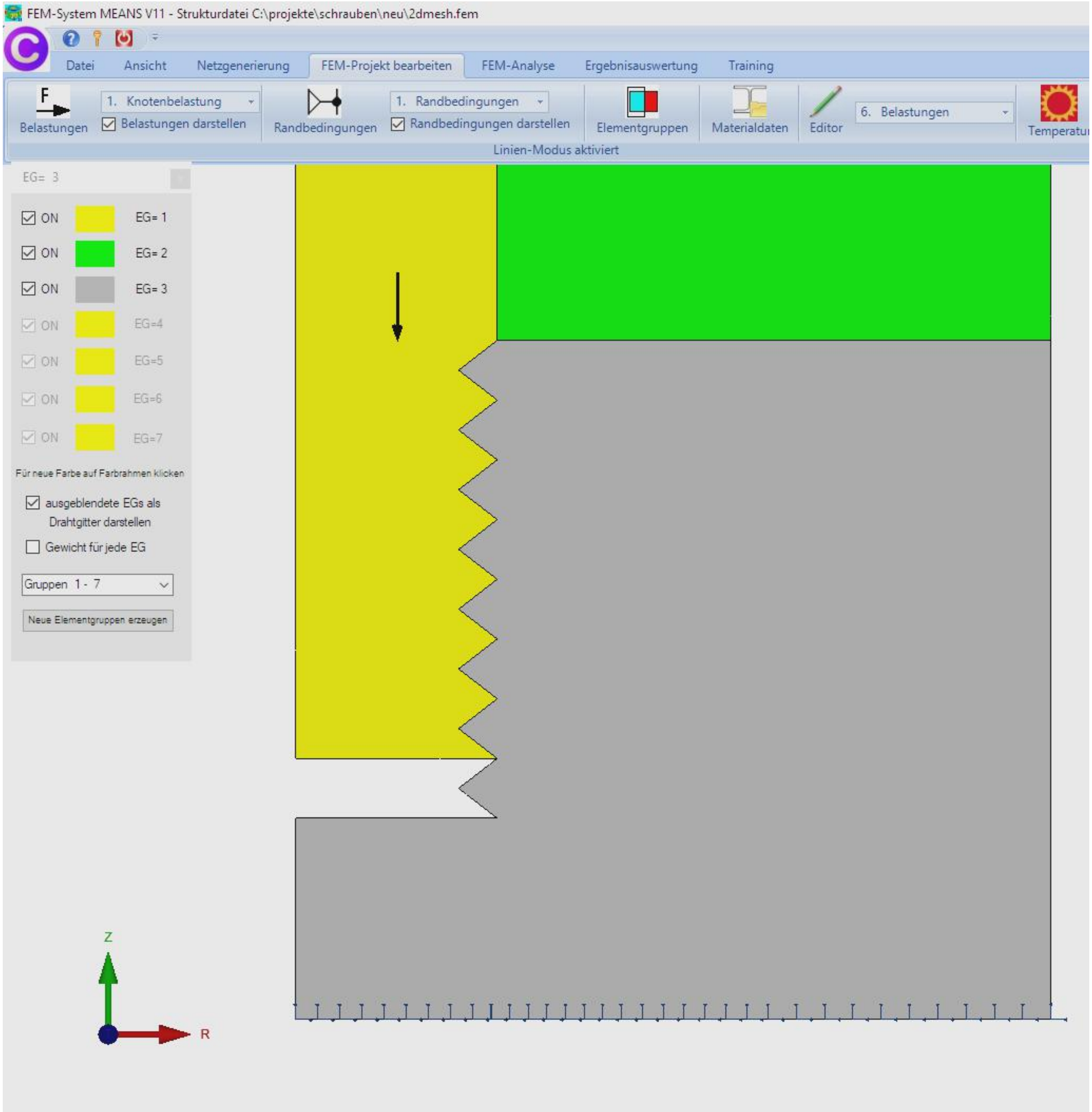
5. Slave Contact Surface

Thecontact nodes between screw, thread and the clamping parts are determined fully automatically and summarized in load case 3.

6. Generate boundary conditions

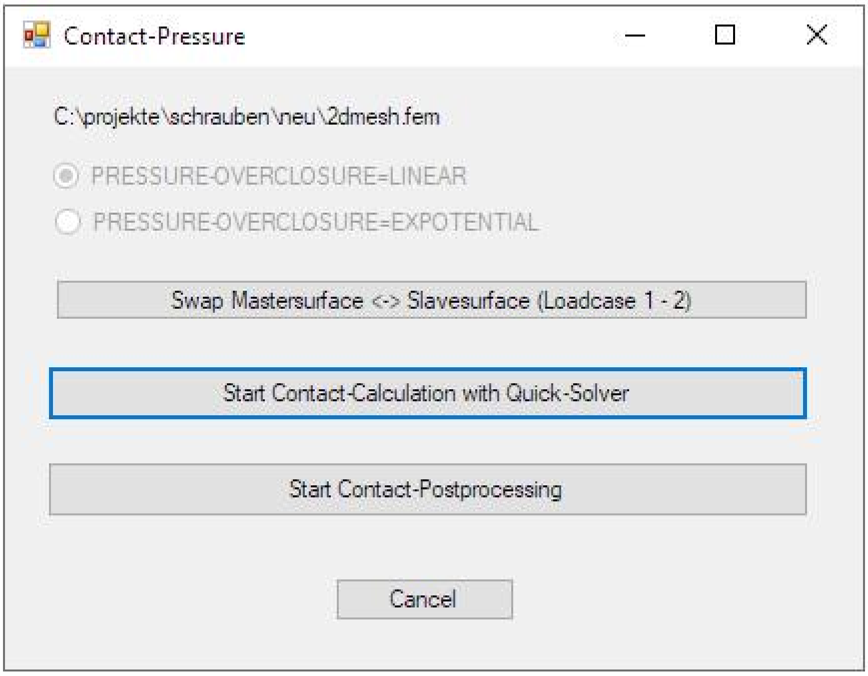
Clamping part 2 is firmly clamped at the bottom in the R and Z directions.

The boundary conditions and the loads can be edited or deleted at any time without the screw mesh generator.

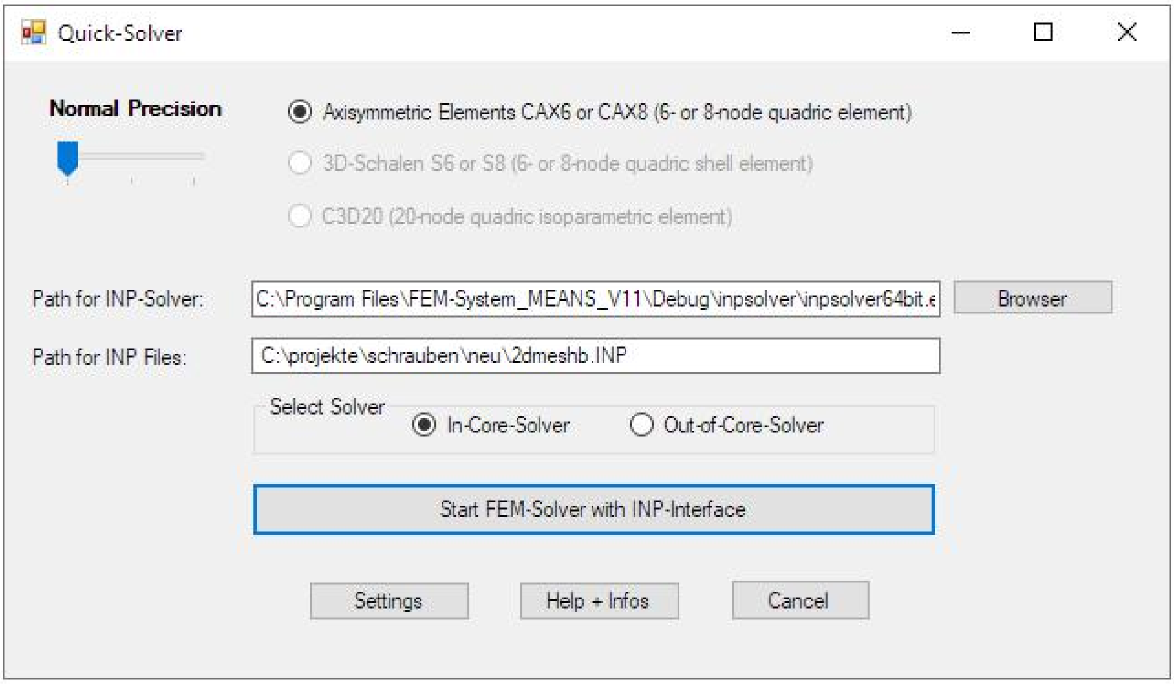


7. FEM-Analysis

Finally, the contact analysis is started in order to calculate the contact stresses. Before the calculation, there is still the possibility to change master and slave surface if a contact analysis should not converge.

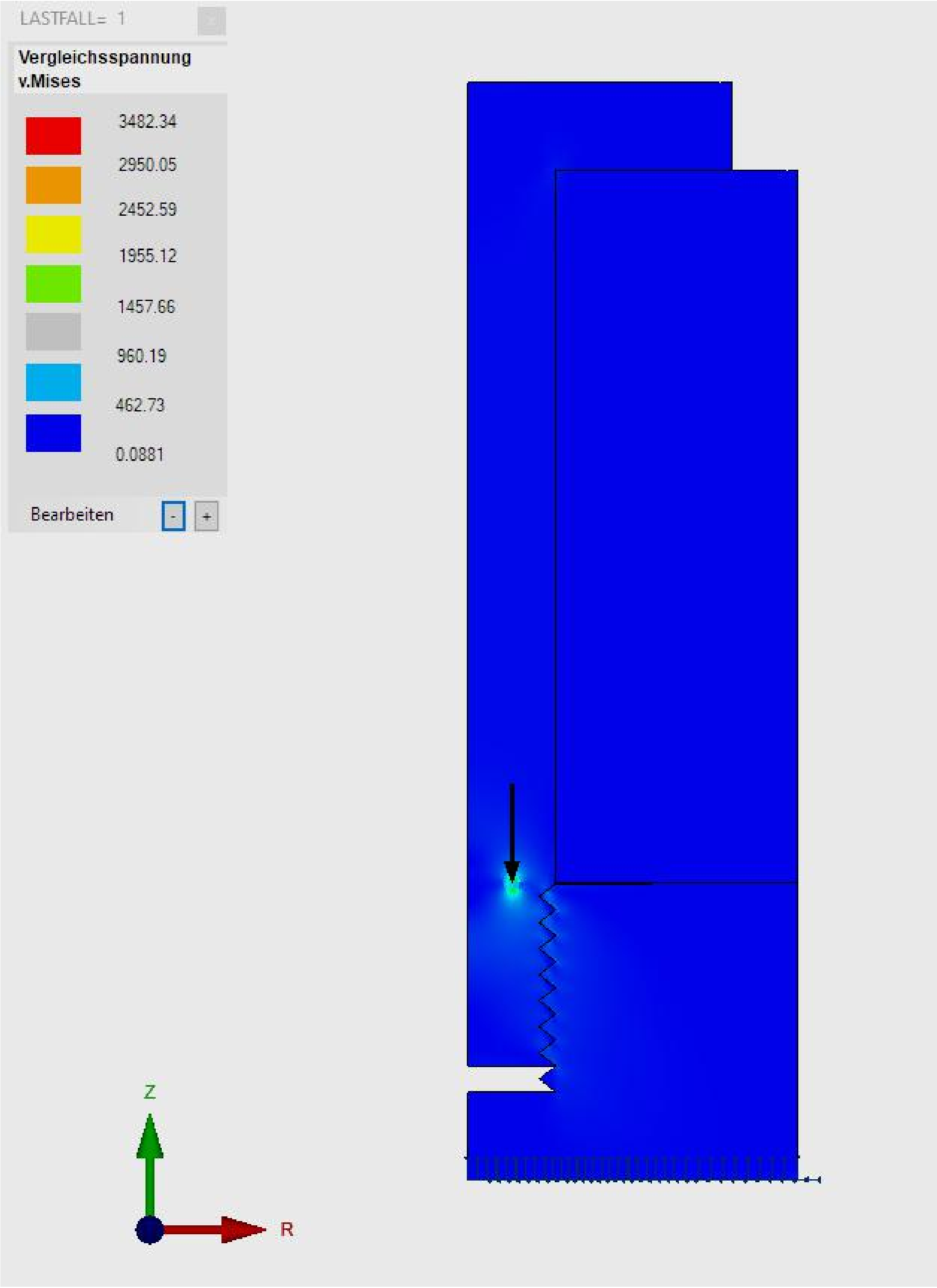


A numerically stable contact analysis converges after 8 to 15 iterations and requires only a few minutes of computing time.



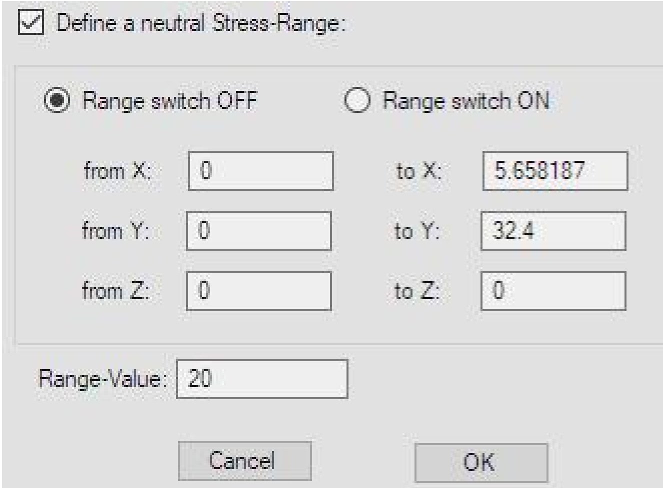
v. Mises Stresses

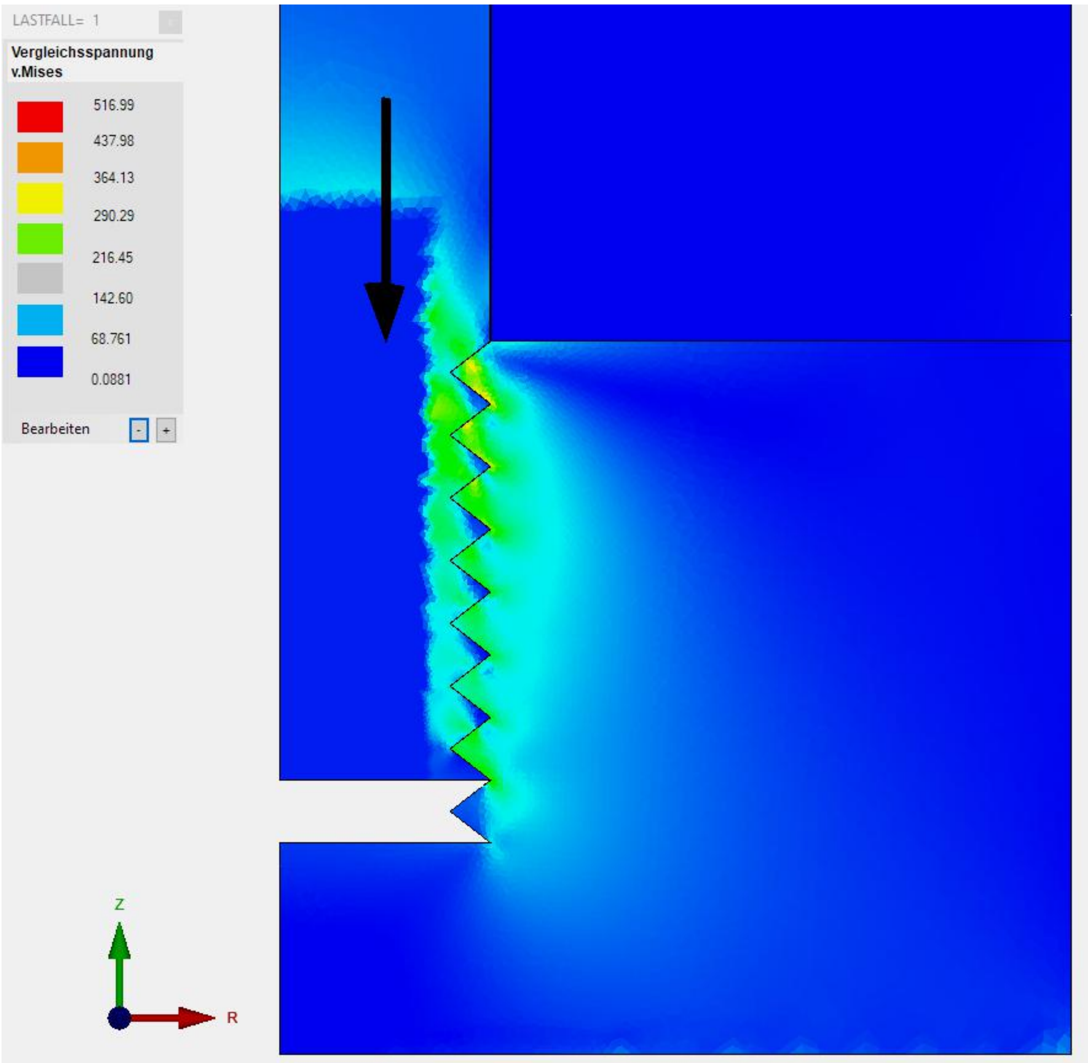
In the range of the nodes of the point load, very high secondary stresses occur, which can be hidden with the next "Screw Thread Stresses" menu.



Screw Thread Stresses

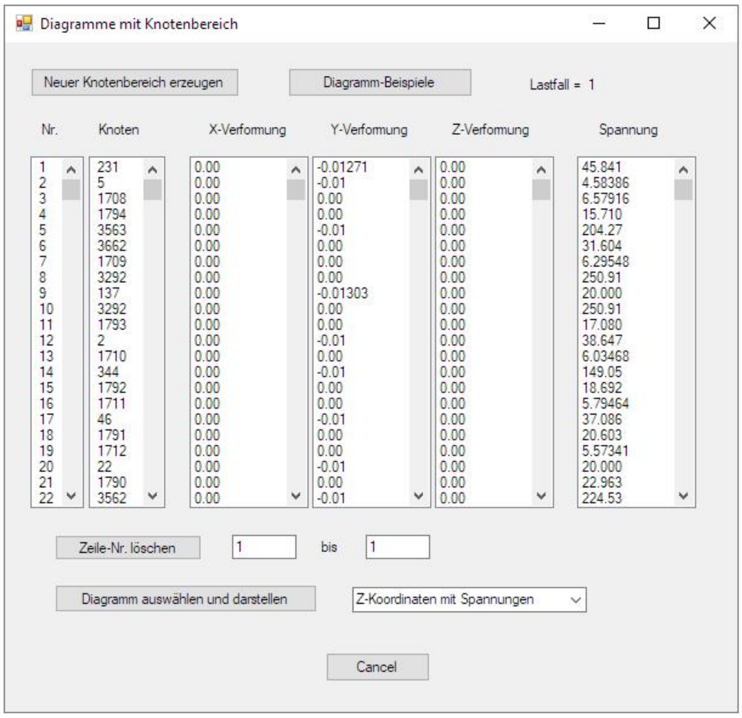
With “Screw Thread Stresses” the stresses can be hidden and the primary v.Mises stresses can be shown on the thread. The hidden R range goes from 0 mm - 5.65 mm and the Z range from 0 mm - 32.4 mm.

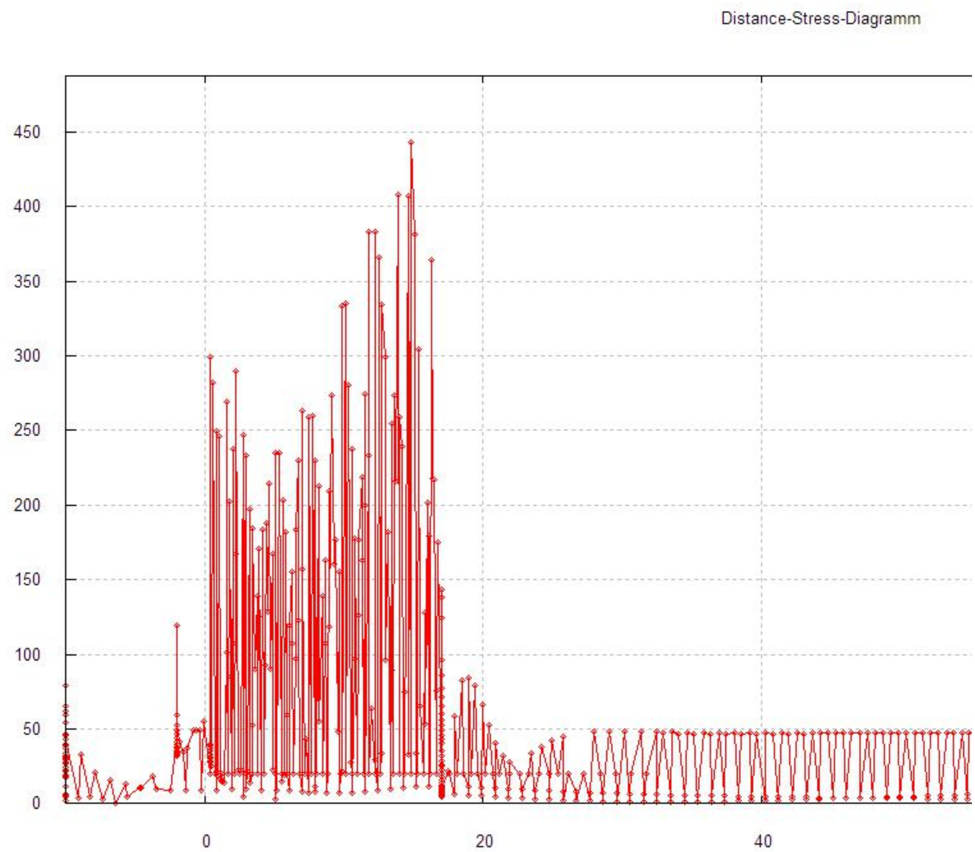




v.Mises-Stresses-Distance Diagram

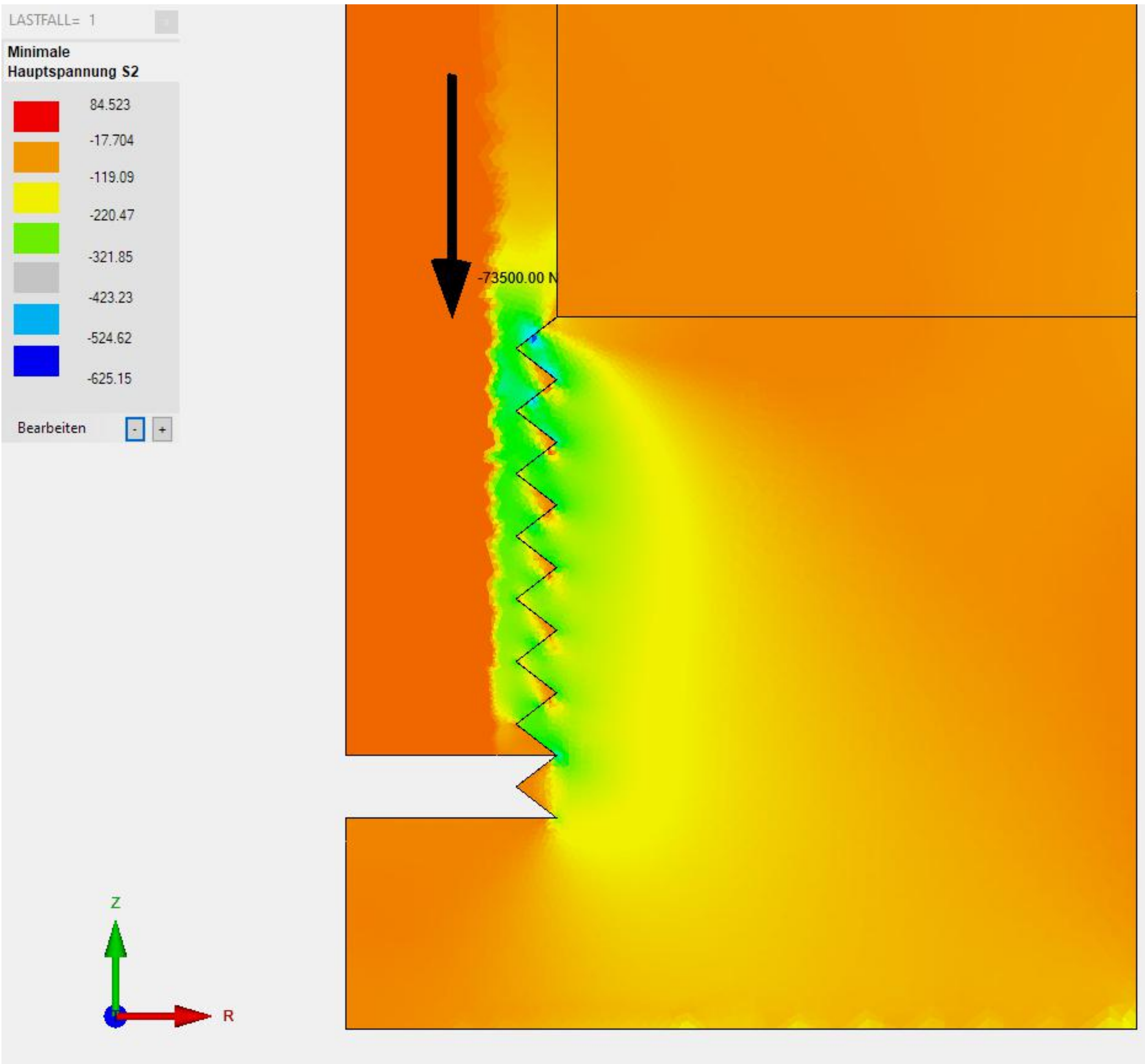
All stresses of the edge nodes along the Z axis are shown in a diagram. However, any other range of nodes can also be defined with “Node Modus”.





Principal Stresses

The maximum and minimum principal stresses S1 and S2 are shown, where S1 is the positive tensile stress and S2 the negative compressive stress.



Exact solution according to Roloff-Matek:

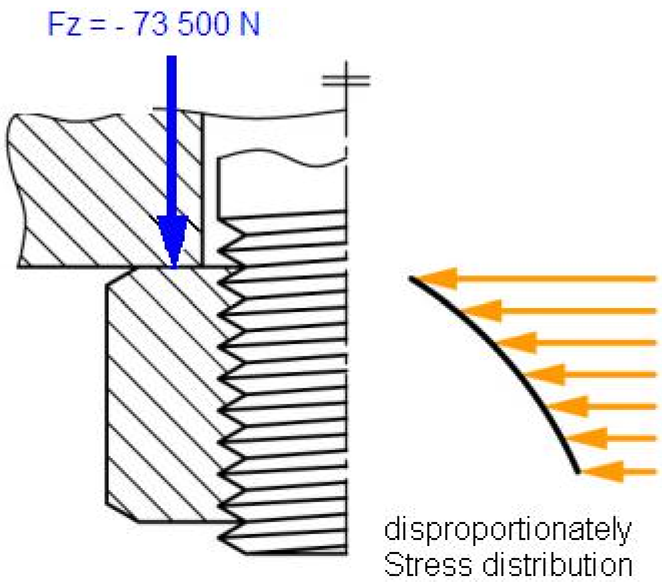
Maximum tensile / compressive stress = F / A = Fz / (P * dmin² / 4)

= 73 500 N / (P * 12.93 mm * 12.93 mm / 4)

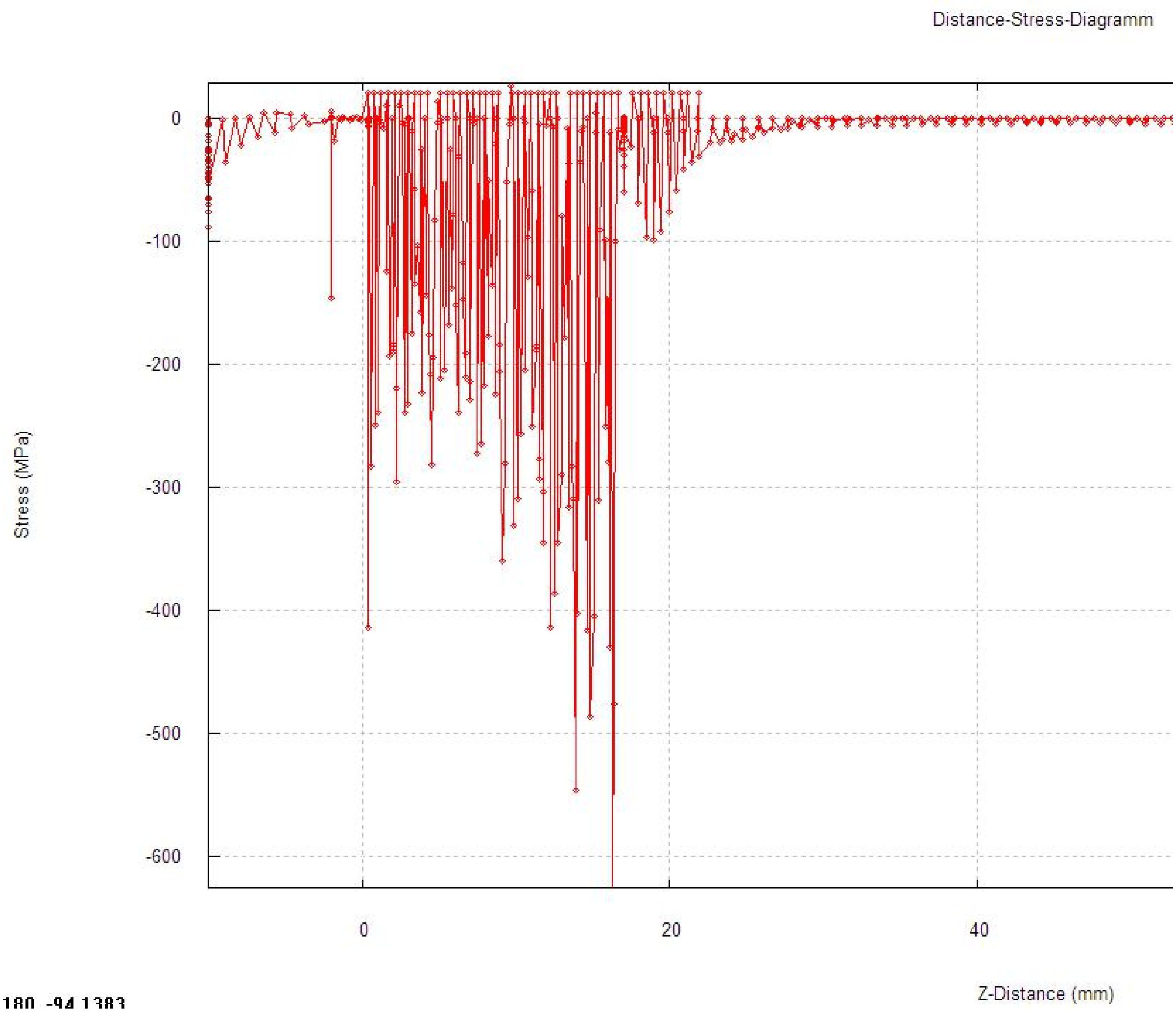
= 559 N / mm²

With this stresses and the minimum screw diameter, the actual screw stresses cannot be determined, because the first thread turns bear the forces disproportionately.

This shows the great advantages of this FEM contact calculation, which reflects the disproportionate stress distribution in the thread very well.

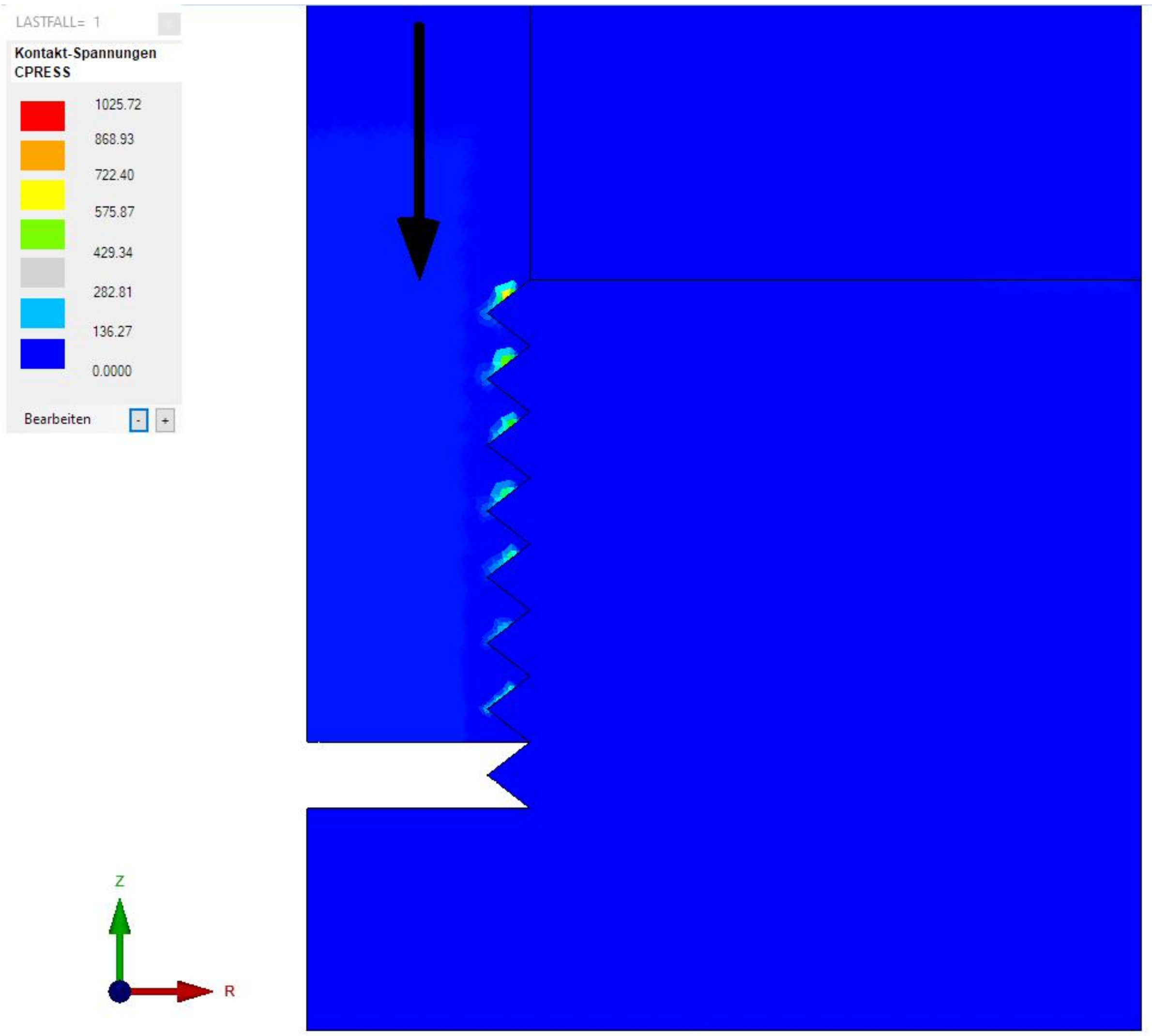


Principal Stress-Distance-Diagram



Contact-Stresses

The contact stresses or Hertzian contact stress are the greatest stresses that arise in the middle of the contact surface of the thread.



Contact-Stress-Diagram

Here you can see very well how the contact stresses decrease disproportionately per thread.