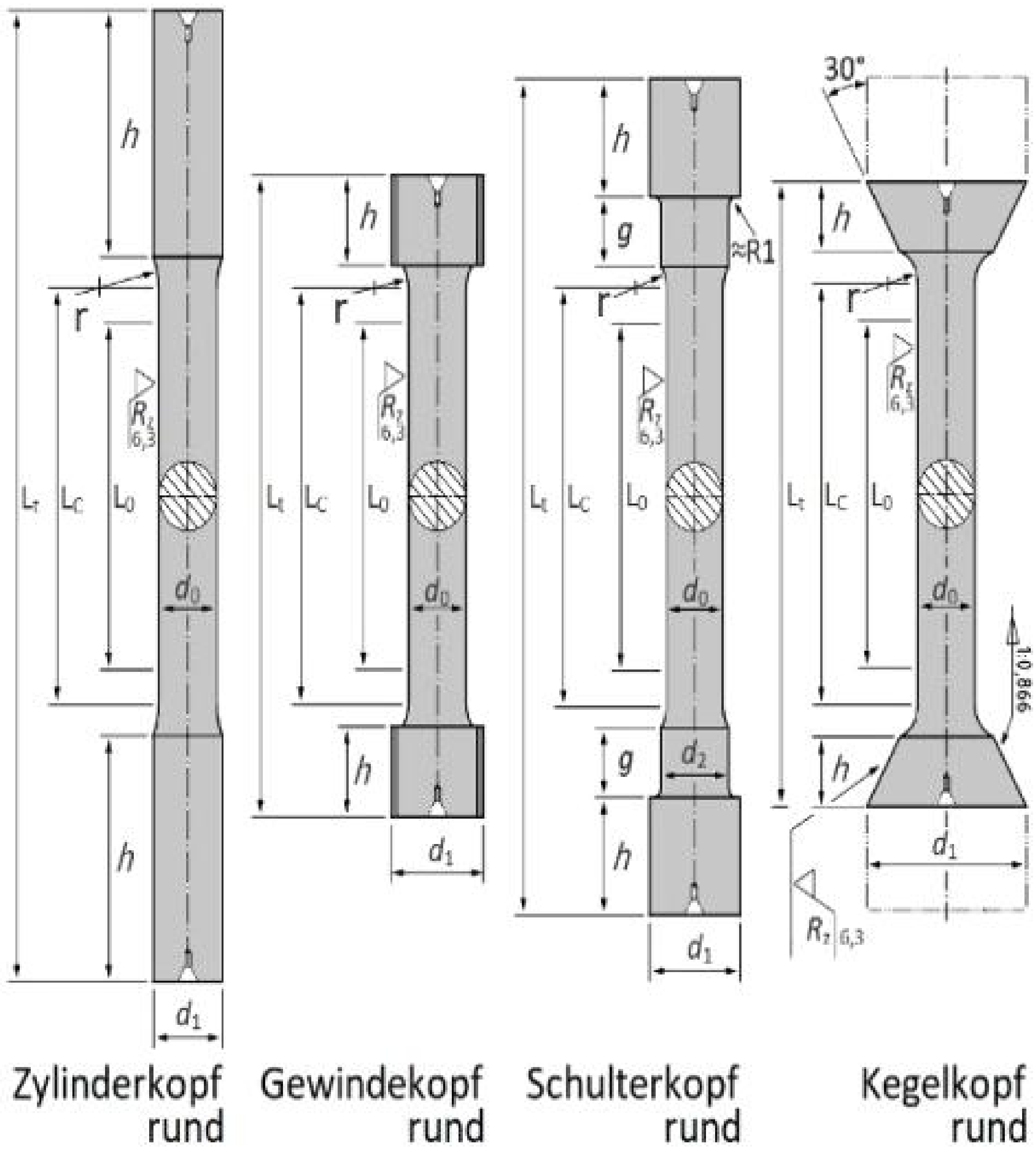
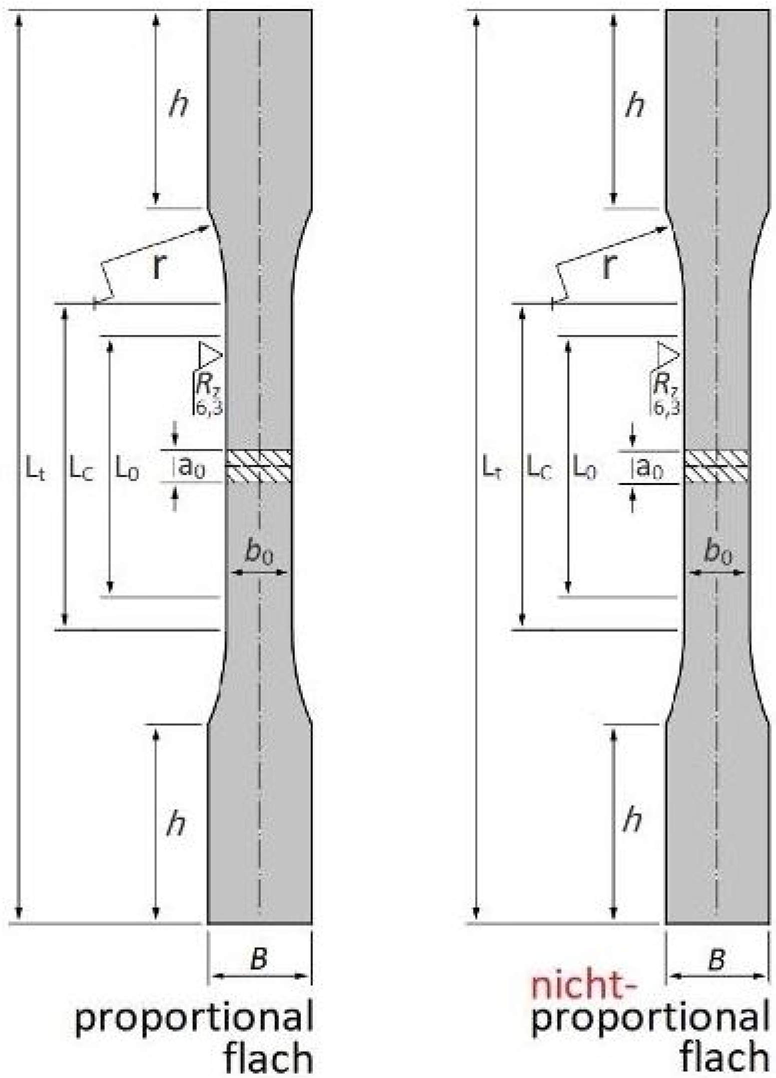
Part 33 - Tensile Test Mesh Generator according to ISO 6892 / DIN 50125

With the new Tensile Test Mesh Generator of the FEM System MEANS V12 (www.fem-infos.com) it is possible to generate tensile test models create of linear or quadratical hexahedrons, pentahedrons or tetrahedrons fully automatically with a few key data and to calculate them non-linearly with a stress-strain diagram .

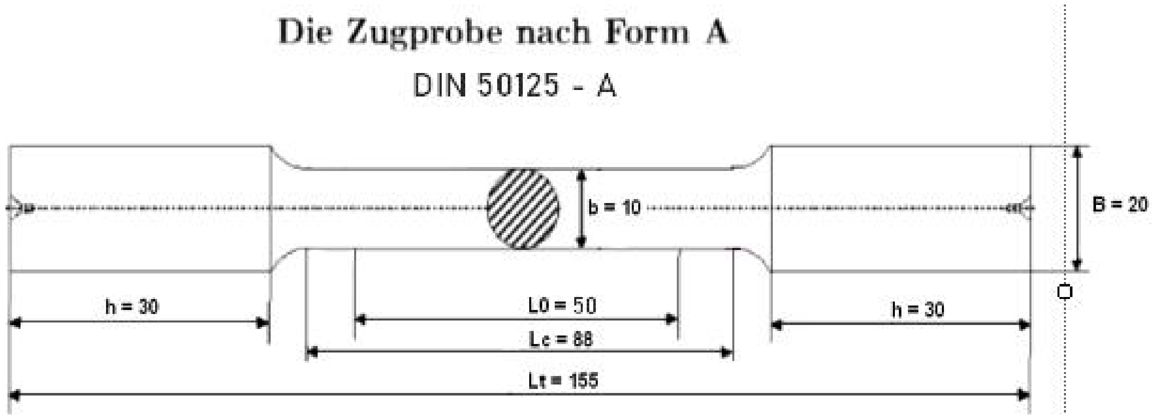
Round Tensile Samples:



Flat Tensile Samples:

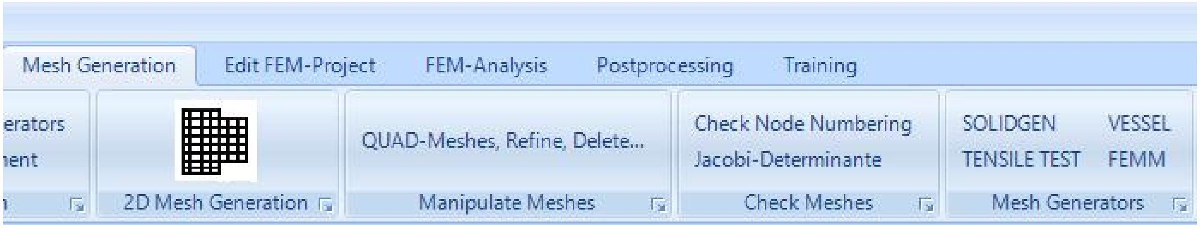


Tensile Sample:



Entering the key data in the tensile test dialog box:

Select the “Mesh Generation” tab and the “TENSILE TEST” submenu and enter the following key data for the tensile test shown above:

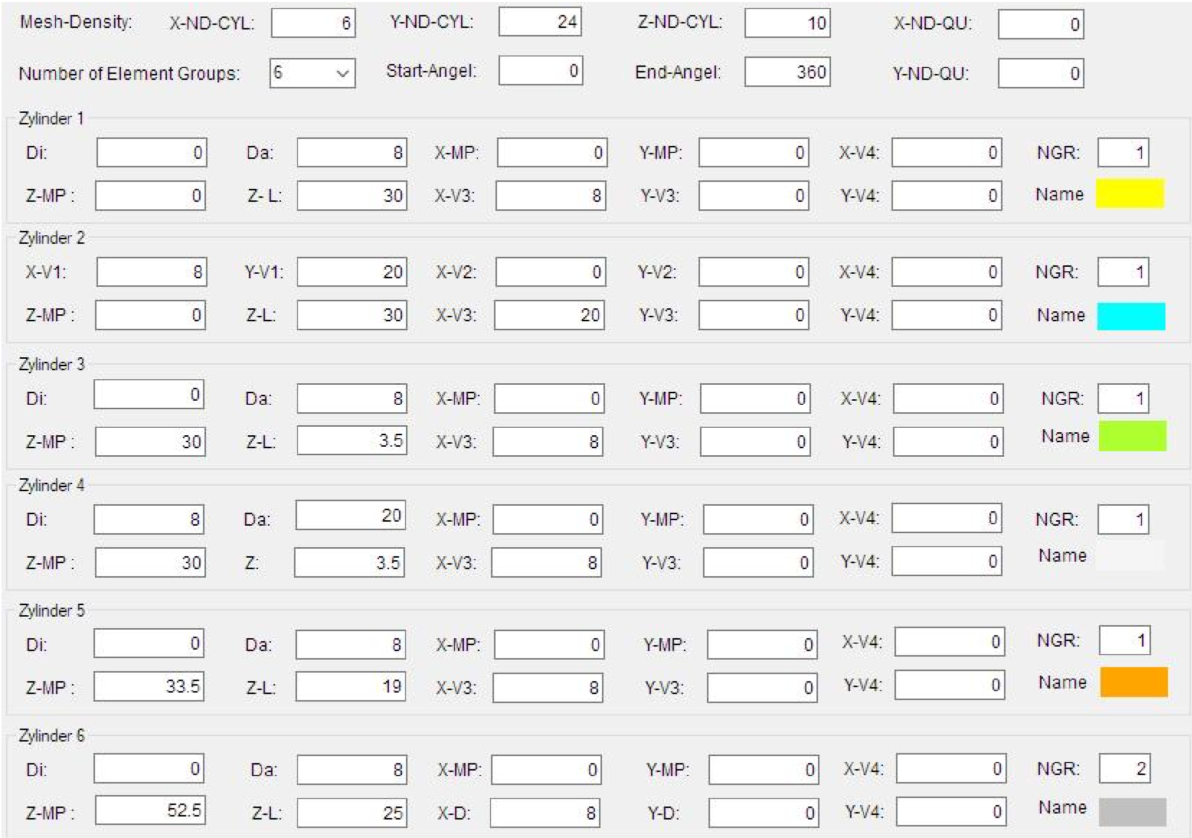


d_0	Probendurchmesser	=	8
d_1	Kopfdurchmesser	=	20
h	Kopfhöhe	=	30
r	Übergangsradius	=	3.5
L_0	Anfangsmesslänge	=	50
L_c	Parallele Länge	=	88
L_t	Gesamtlänge	=	155
d_2	Durchmesser des Ansatzes	=	0
g	Länge des Ansatzes	=	0
a_0	Probendicke	=	0
b_0	Probenbreite	=	0
B	Kopfbreite	=	0

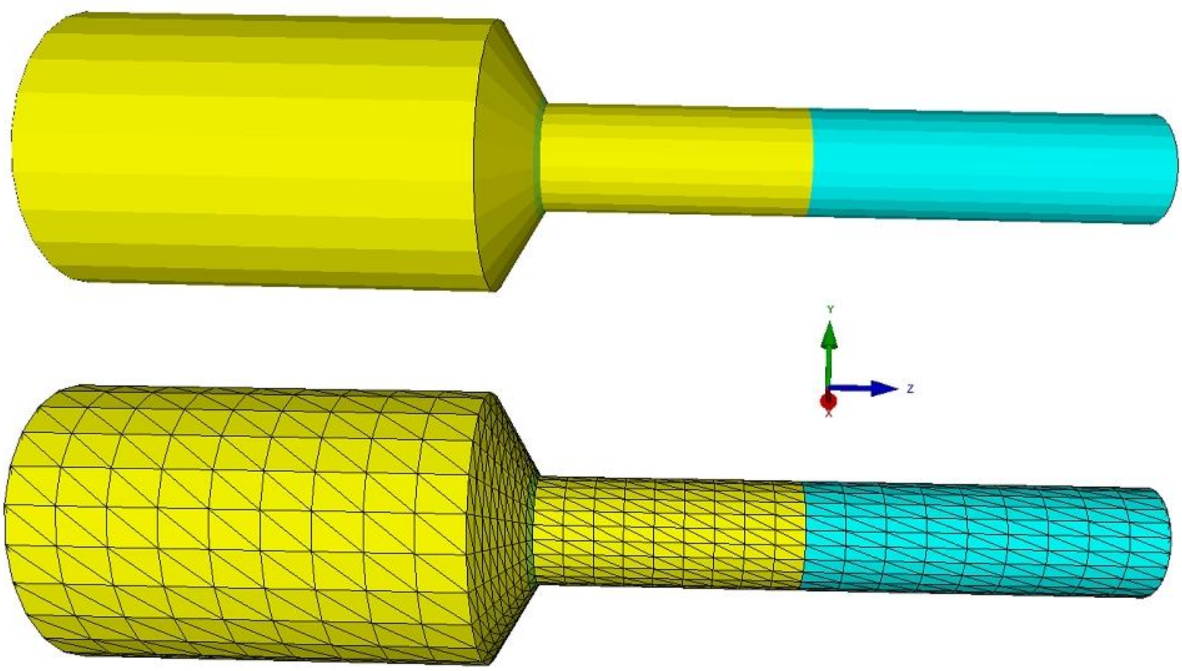
Select the “Create Tensile Sample with SOLIDGEN” button

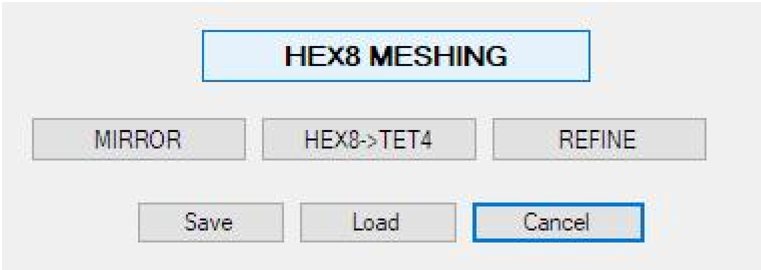
to generate the tensile sample using the SOLIDGEN mesh generator with 6 cylinders and mesh density:

Number of nodes in the radial direction X-ND = 6
Number of nodes around the circumference Y-ND = 24
Number of 2D meshes in the Z direction Z-ND = 10



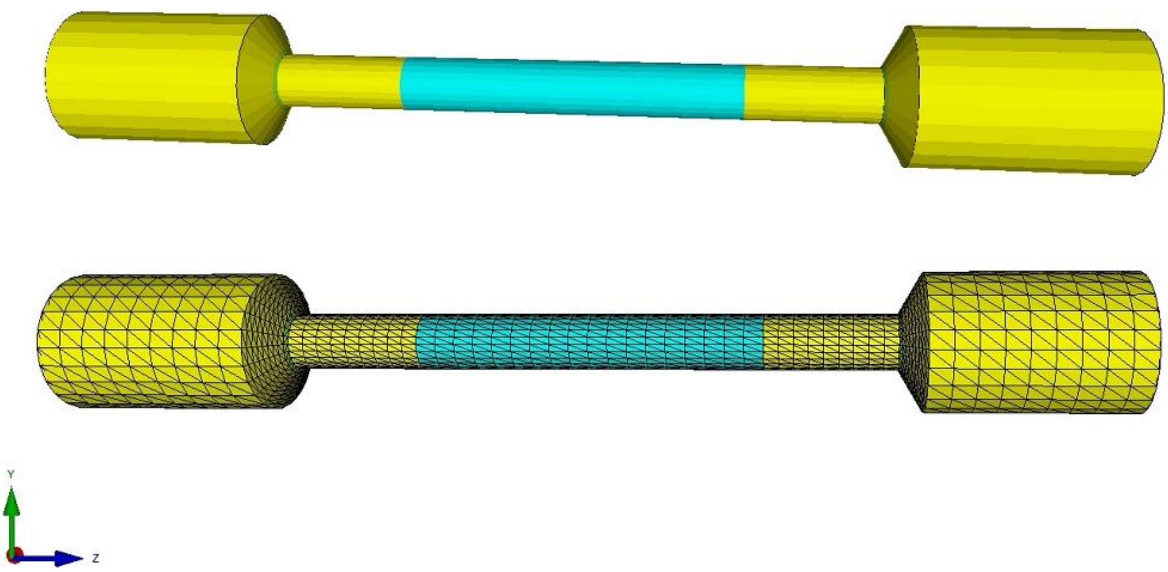
Select menu “HEX8 Meshing” to create a linear FEM model from 6480 HEX8 and 923 PEN6 elements as well as 6637 nodes and 3 element groups.





Mirror model

Select “Mirror” menu to create a FEM model from 12,960 HEX8 elements and 13,153 nodes and 3 element groups.



TET4 Model

Select the menu “HEX8->TET4” to convert the FEM model into a tetrahedral mesh.

Refine

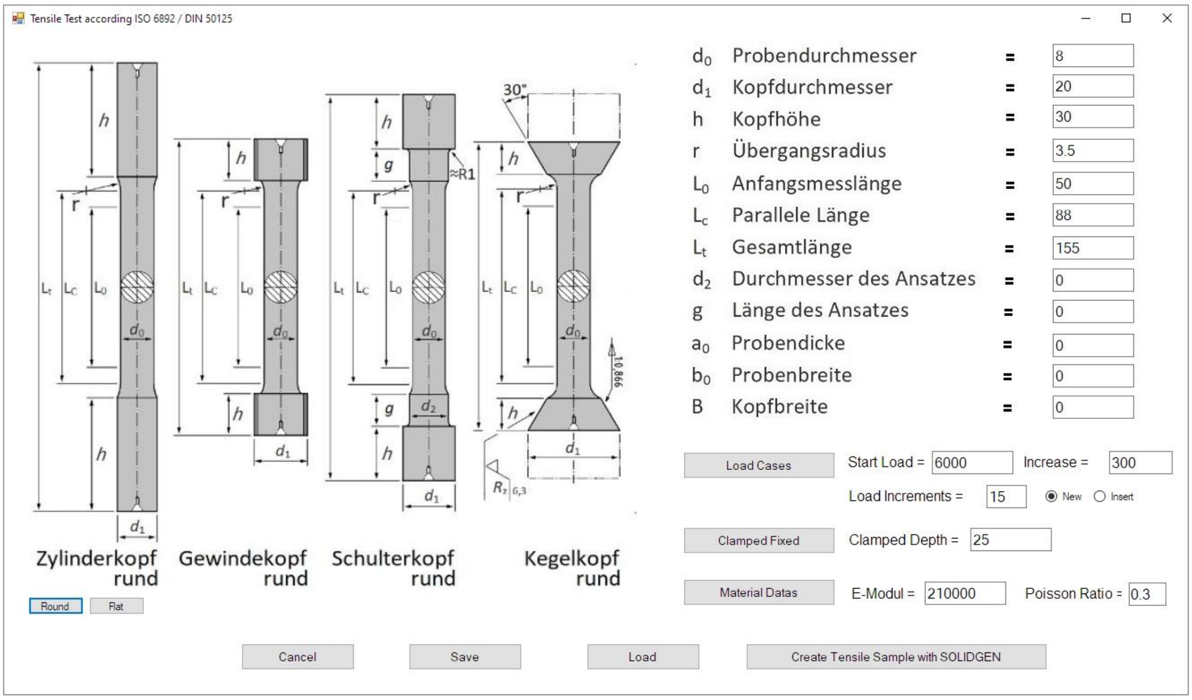
Select the “Refine” menu to create an 8x finer mesh from the tetrahedral mesh

Increase in accuracy

For higher accuracy, the FEM model can later be converted into TET10 or HEX20 and PEN15 mesh in the quick solver.

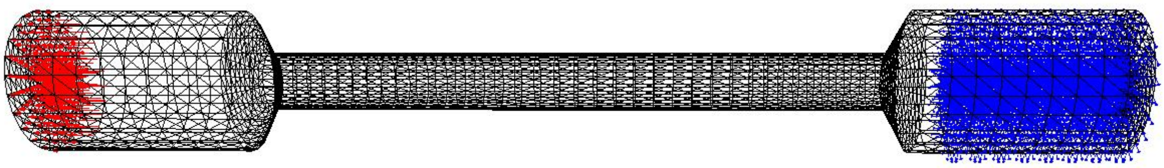
Generate Load Cases

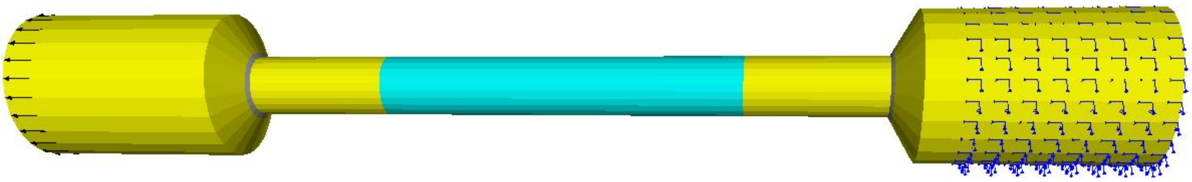
In the tensile test dialog box, define an increasing axial load in the Z direction with a starting load of 6000 N and a load increase of 500 N and 15 load increments at the top of the tensile test at Z = 0 mm.
Select “Insert” to be able to examine the area of the yield point in more detail with finer load case steps, for example.



Generate boundary conditions

At the end of the tensile sample, define a clamping in the X, Y and Z directions with a clamping depth of 25 mm.



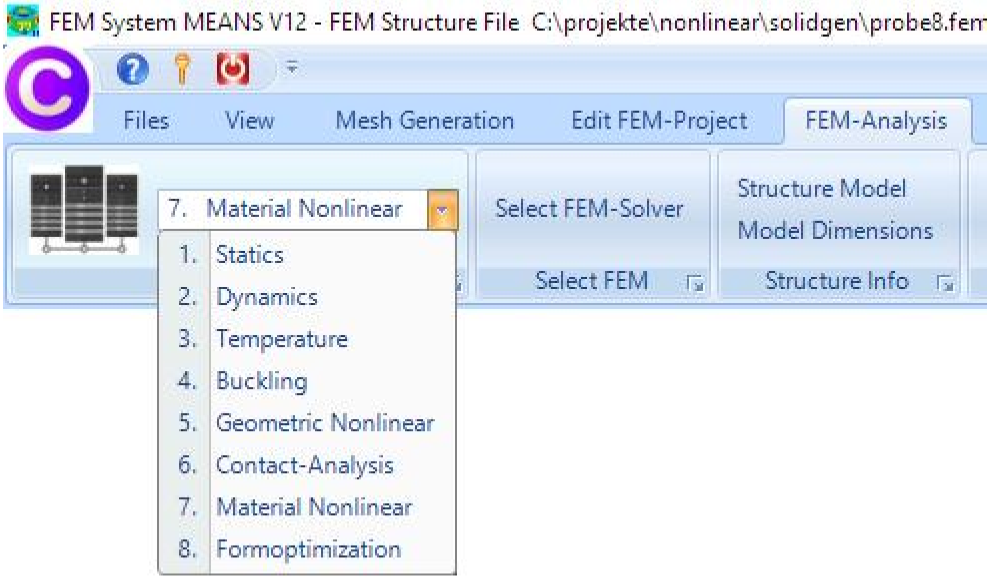


Generate material data

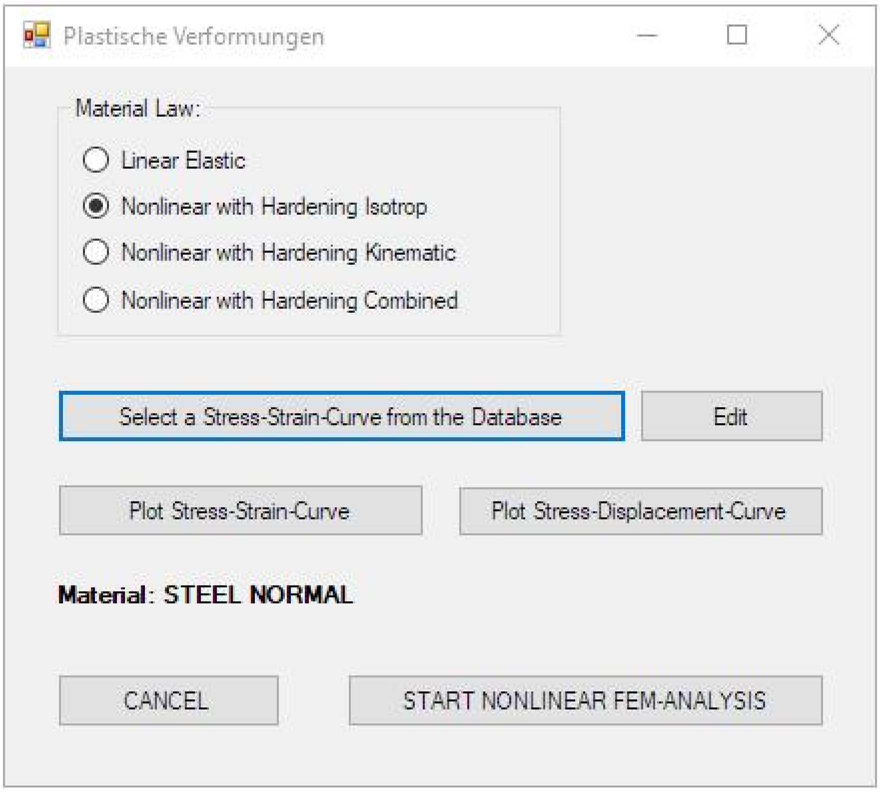
Define with an elastic modulus of 210,000 N/mm² and a Poisson's ratio of 0.3 for the material steel, which is always preset.

Nonlinear FEM Analysis

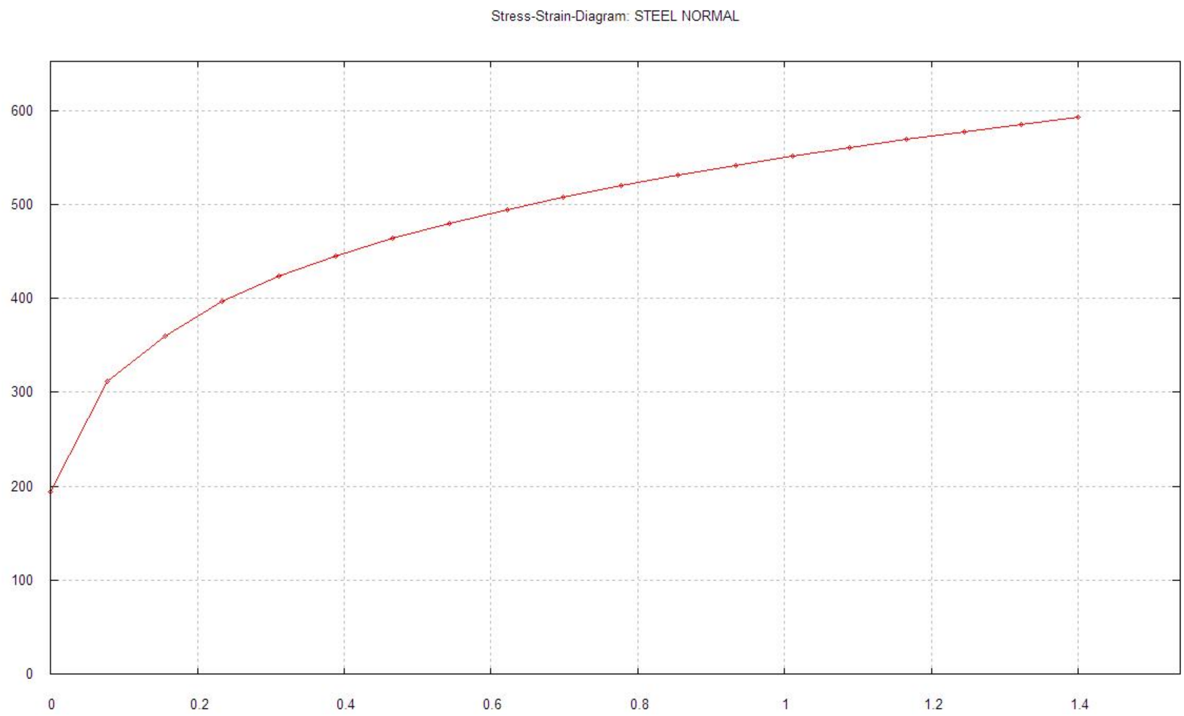
Select the “FEM Analysis” tab and “Material Nonlinear” menu.



Select the FEM analysis “Nonlinear with Hardening Isotropic” and the stress-strain diagram “STEEL NORMAL”.



Select “Plot Stress-Strain Curve” to display the stress-strain curve:



STEEL NORMAL

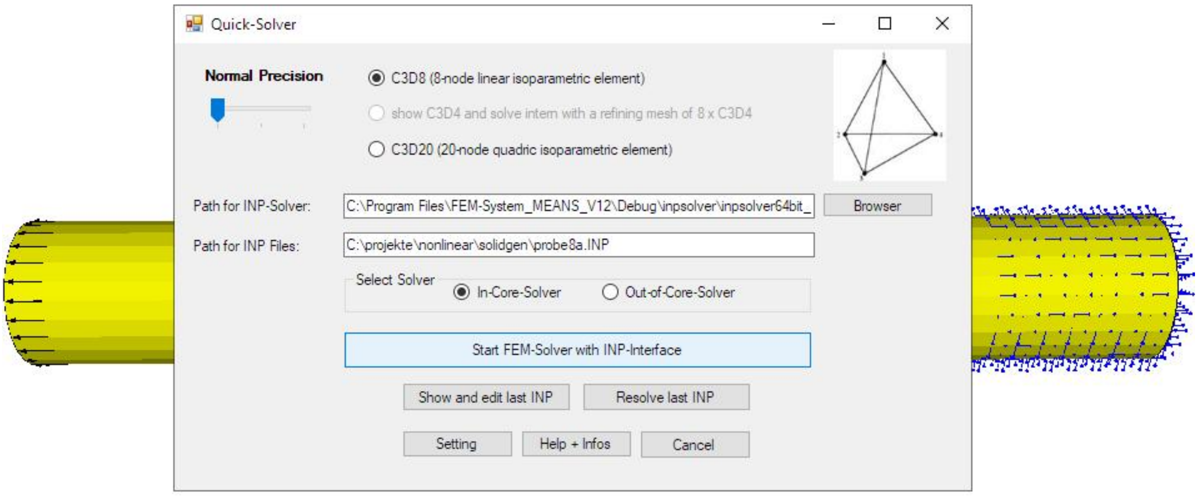
The displacements and stresses increase linear up to the yield strength of 190 N/mm² (= Hook's straight line), then the plastic deformations begin with a flattening of the curve, i.e. the strains increase more than the stresses.

The curve can also be edited using any text editor and “Edit” menu:

```
STEEL NORMAL
          0.000000000E+00          1.94000000D+02
          0.077700000E+00          3.11210000D+02
          0.155500000E+00          3.59950000D+02
          0.233000000E+00          3.96540000D+02
          0.311000000E+00          4.23890000D+02
          0.388000000E+00          4.45580000D+02
          0.466000000E+00          4.63940000D+02
          0.544000000E+00          4.80260000D+02
          0.622000000E+00          4.94950000D+02
          0.699000000E+00          5.08310000D+02
          0.777000000E+00          5.20260000D+02
          0.855000000E+00          5.31420000D+02
          0.933000000E+00          5.41870000D+02
          1.011000000E+00          5.51710000D+02
          1.088000000E+00          5.60910000D+02
          1.166000000E+00          5.69560000D+02
          1.244000000E+00          5.77820000D+02
          1.322000000E+00          5.85720000D+02
          1.399000000E+00          5.93280000D+02
-END-
```

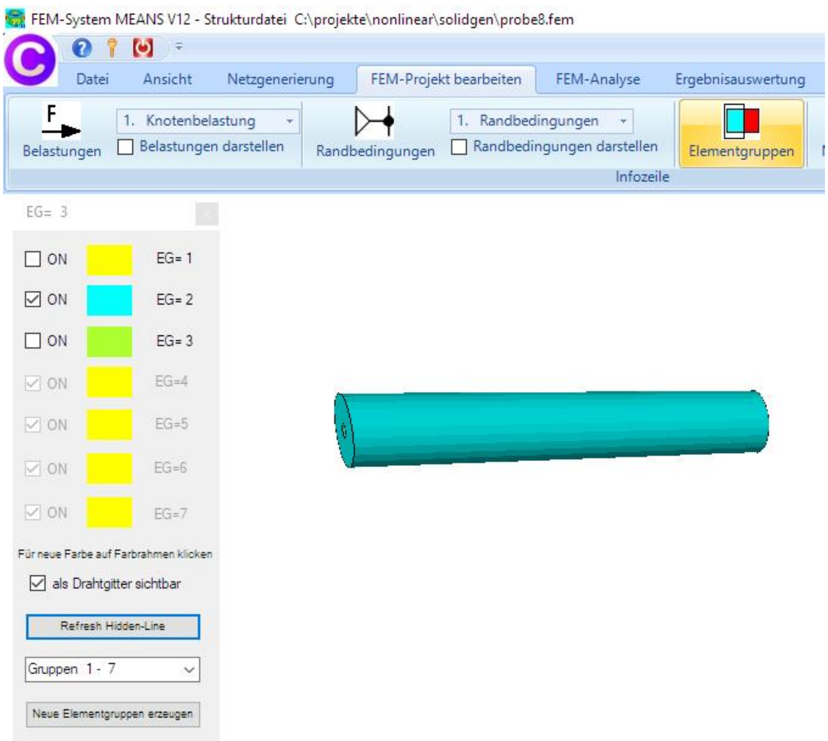
Start Quick Solver

Select the “START NONLINEAR FEM ANALYSIS” button to start the quick solver with the setting “C3D8 (8-node linear isoparametric element)”.



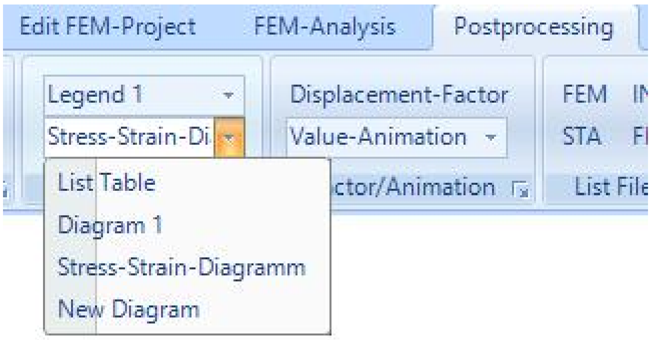
Stress-strain diagram for element group 2

Use the “Edit FEM project” tab to only show element group 2 so that only the results of the Gage Length L0 and no secondary stresses are evaluated.



Show Stress-Strain diagram

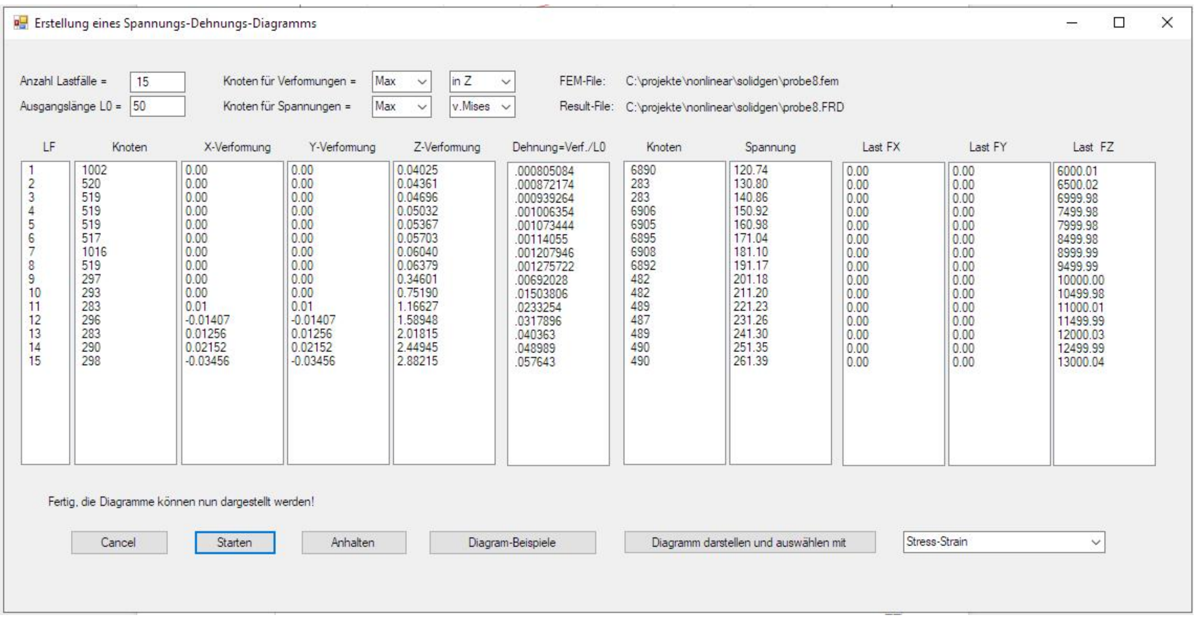
After the nonlinear FEM calculation, you can first display the important stress-strain curve using the “Postprocessing” and “Stress-strain diagram” tabs.



Set Gage Length L0

Enter the Gage Length = 50 mm in the dialog box and select the “Start” button so that the Displacements, v.Mises Stress and the Load per load increment are read and clearly displayed in the table.

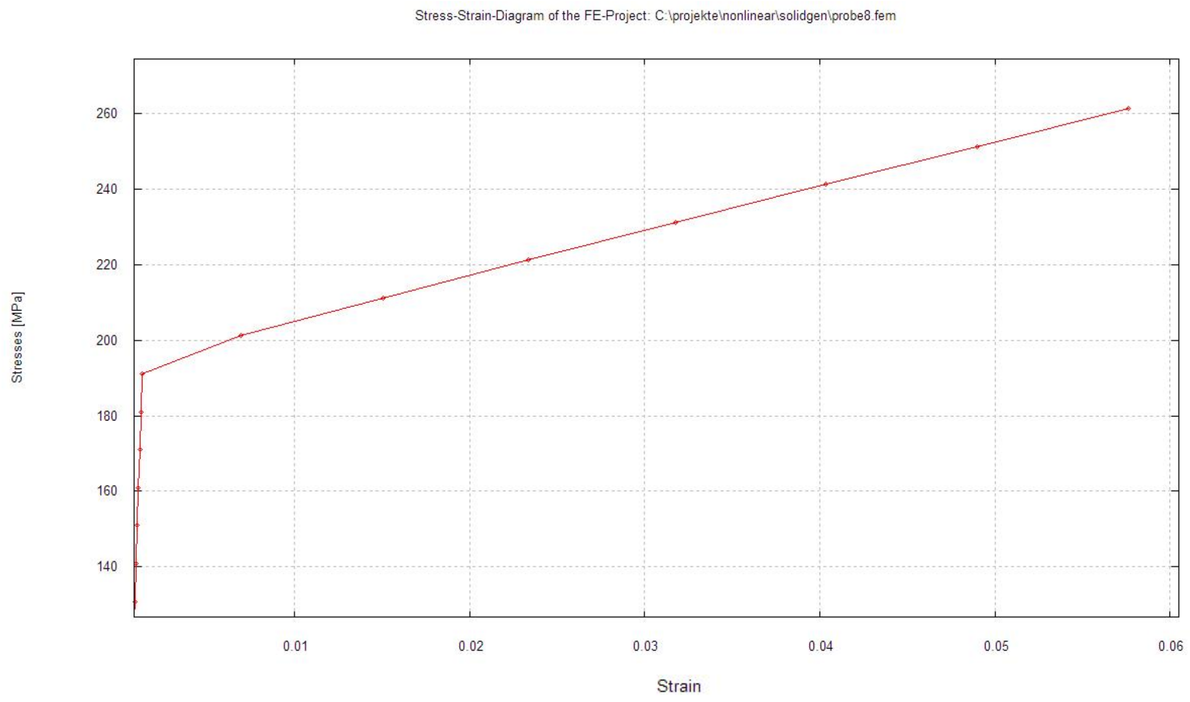
In addition, the strains are output from the displacements divided by L0.



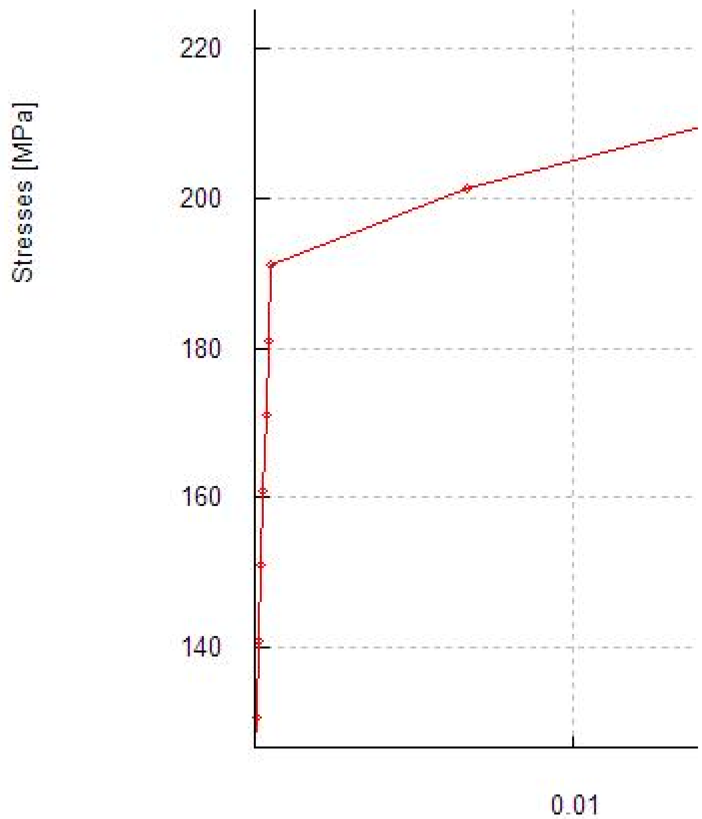
LF	Knoten	X-Verfomung	Y-Verfomung	Z-Verfomung	Dehnung=Verf./L0	Knoten	Spannung	Last FX	Last FY	Last FZ
1	1002	0.00	0.00	0.04025	.000805084	6890	120.74	0.00	0.00	6000.01
2	520	0.00	0.00	0.04361	.000872174	283	130.80	0.00	0.00	6500.02
3	519	0.00	0.00	0.04696	.000939264	283	140.86	0.00	0.00	6999.98
4	519	0.00	0.00	0.05032	.001006354	6906	150.92	0.00	0.00	7499.98
5	519	0.00	0.00	0.05367	.001073444	6905	160.98	0.00	0.00	7999.98
6	517	0.00	0.00	0.05703	.00114055	6895	171.04	0.00	0.00	8499.98
7	1016	0.00	0.00	0.06040	.001207946	6908	181.10	0.00	0.00	8999.99
8	519	0.00	0.00	0.06379	.001275722	6892	191.17	0.00	0.00	9499.99
9	297	0.00	0.00	0.34601	.00692028	482	201.18	0.00	0.00	10000.00
10	293	0.00	0.00	0.75190	.01503806	482	211.20	0.00	0.00	10499.98
11	283	0.01	0.01	1.16627	.0233254	489	221.23	0.00	0.00	11000.01
12	296	-0.01407	-0.01407	1.58948	.0317896	487	231.26	0.00	0.00	11499.99
13	283	0.01256	0.01256	2.01815	.040363	489	241.30	0.00	0.00	12000.03
14	290	0.02152	0.02152	2.44945	.048989	490	251.35	0.00	0.00	12499.99
15	298	-0.03456	-0.03456	2.88215	.057643	490	261.39	0.00	0.00	13000.04

Stress-Strain diagram

Select the “Plot Diagram with Selection” button as well “Stress-Strain” to display the diagram with GNUPLOT.

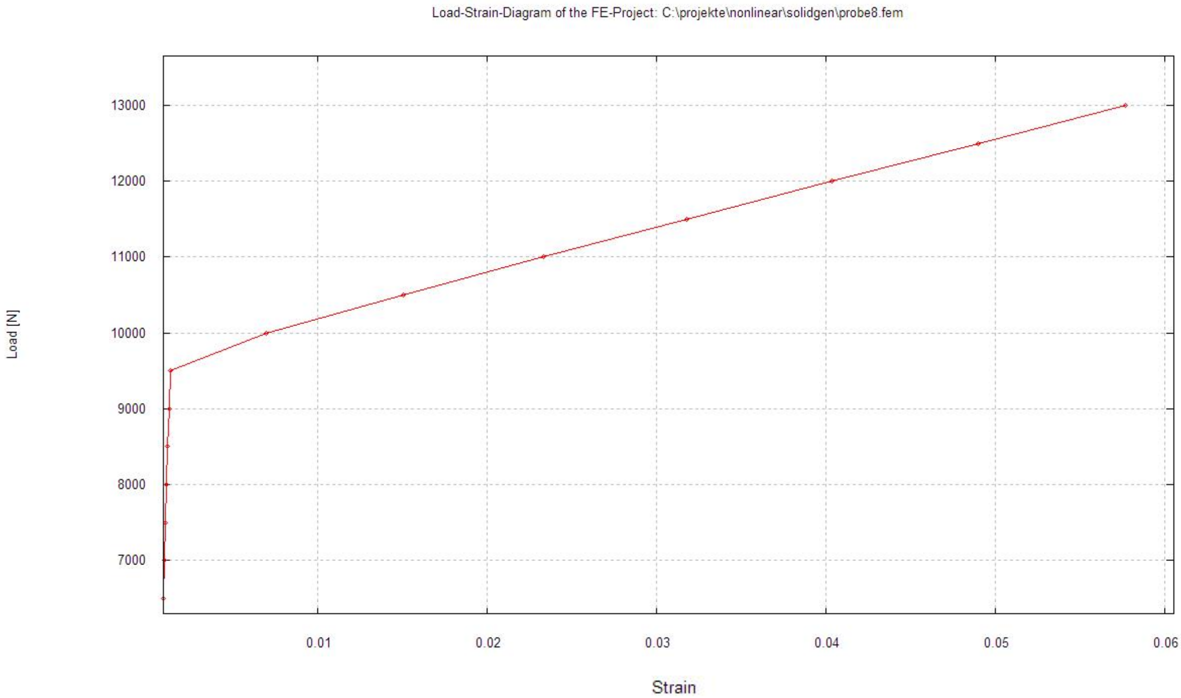


The plastic deformations begin at the yield strength of 190 N/mm²

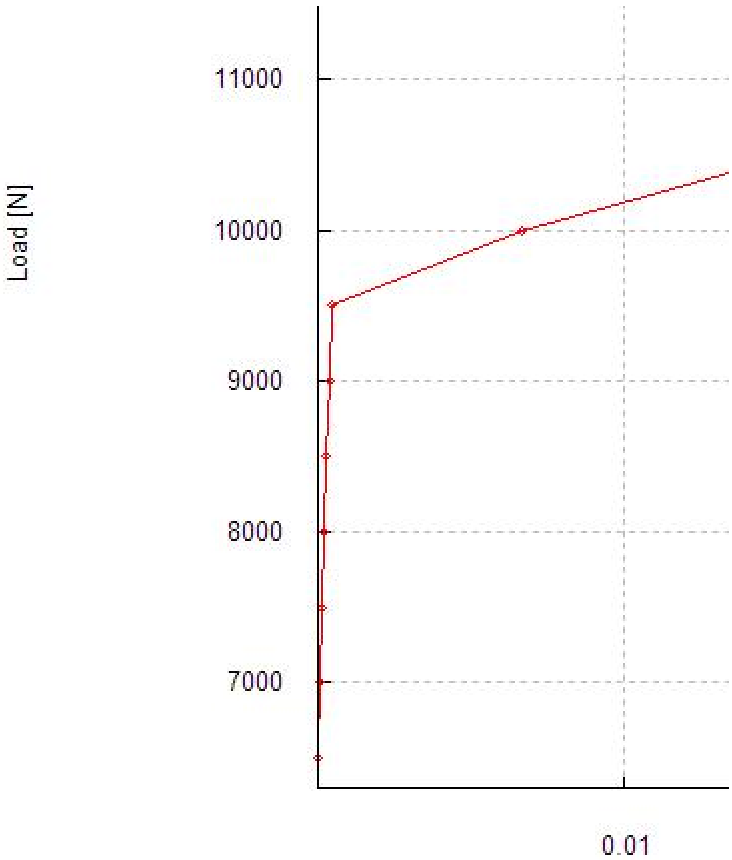


Load-Strain diagram

Select the “Plot Diagram with Selection” button as well “Load-Strain” to display the diagram with GNUPLOT.

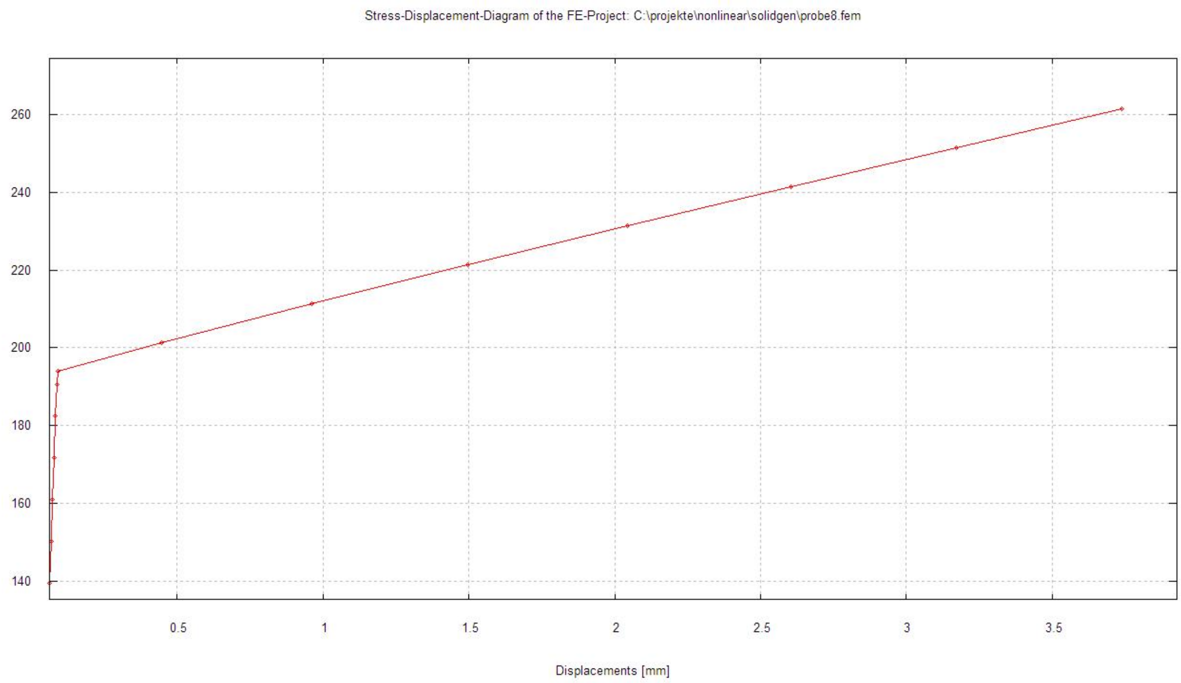


The plastic deformations begin at an axial load of 9500 N



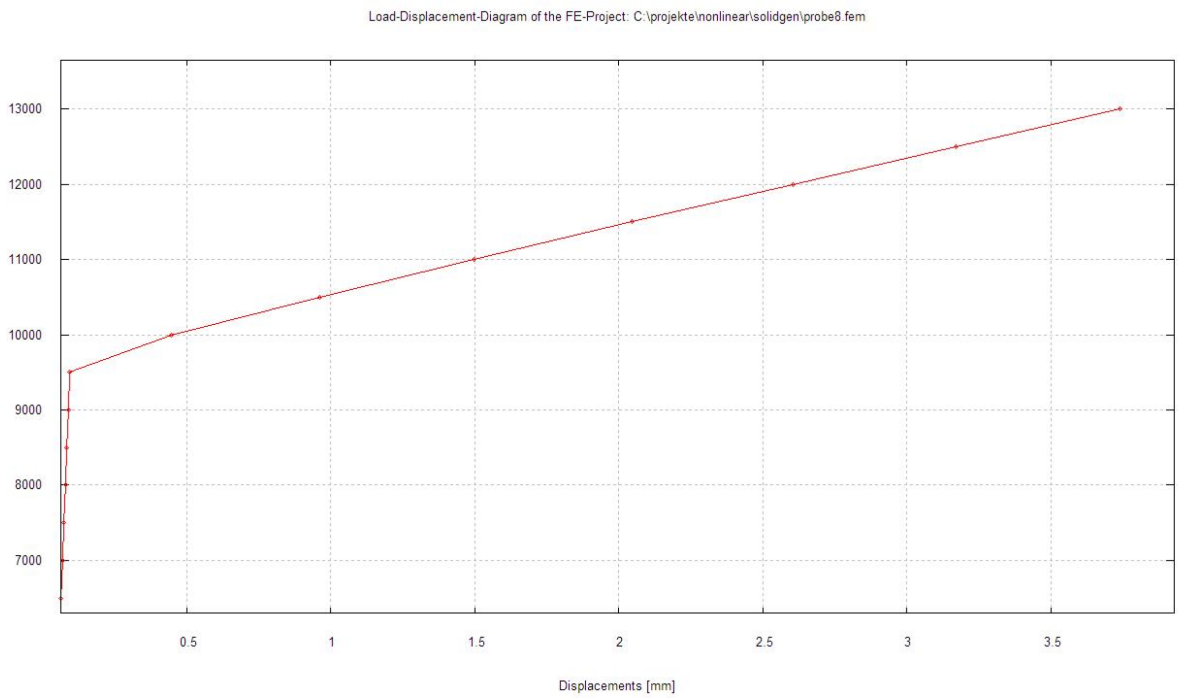
Stress-Displacement diagram

Select the “Plot Diagram with Selection” button as well “Stress-Displacement” to display the diagram with GNUPLOT.



Load-Displacement diagram

Select the “Plot Diagram with Selection” button as well “Load-Displacement” to display the diagram with GNUPLOT.

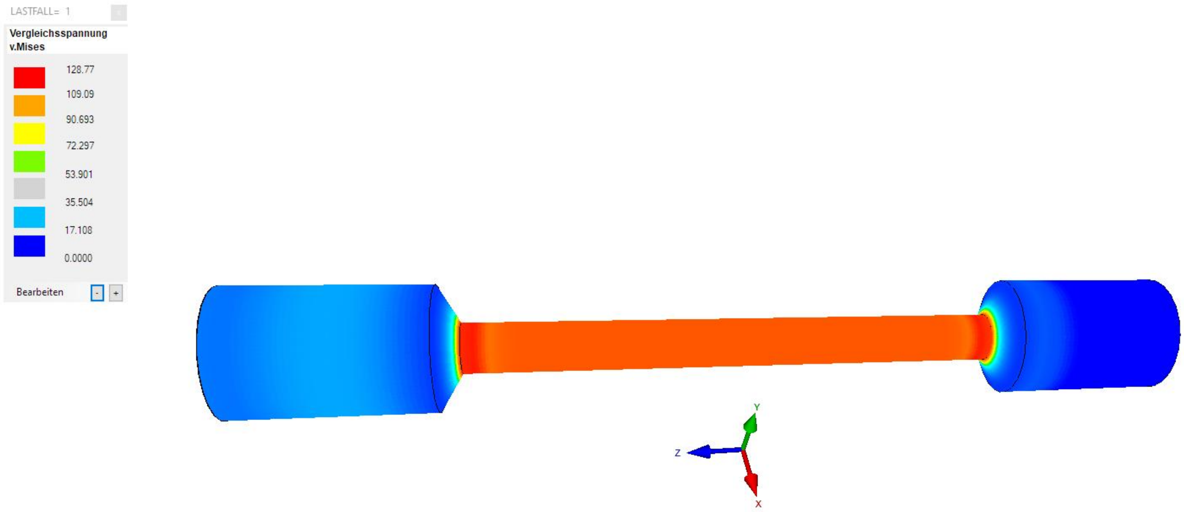


Stress Distribution

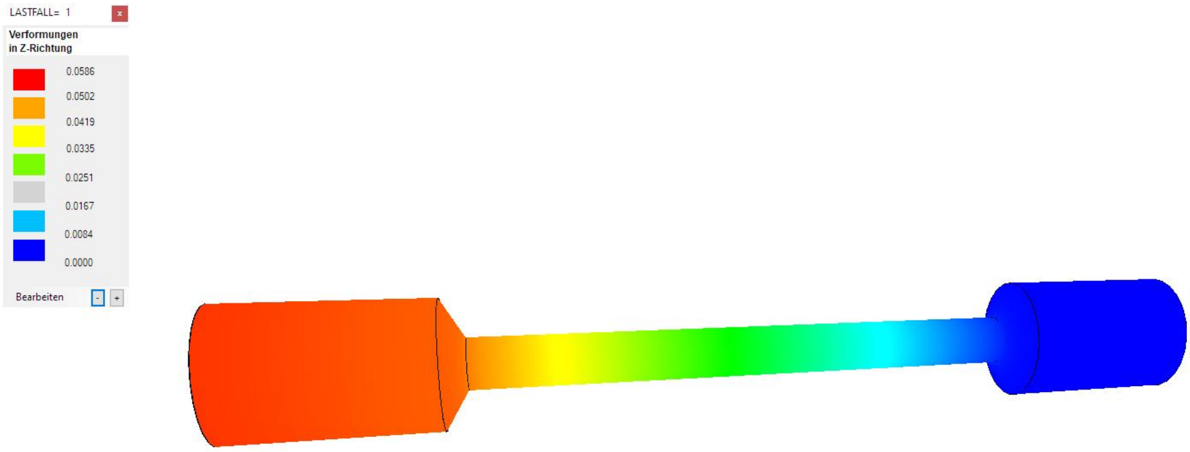
Select the “Postprocessing” and “Nodal Stresses” tab to display the v.Mises stress or displacement distribution for load cases 1-15.

Load Case 1

v.Mises Stress = 128 N/mm²

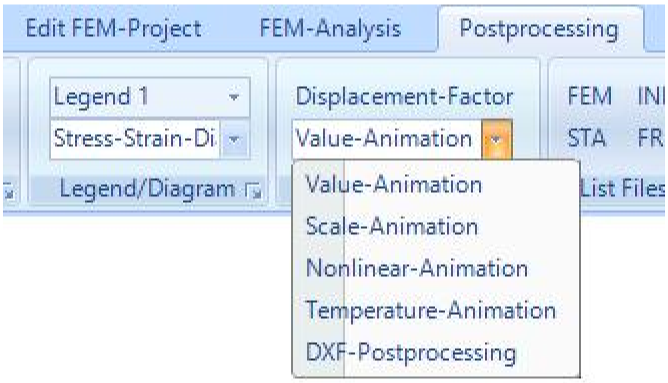


Displacements in Z direction = 0.05 mm



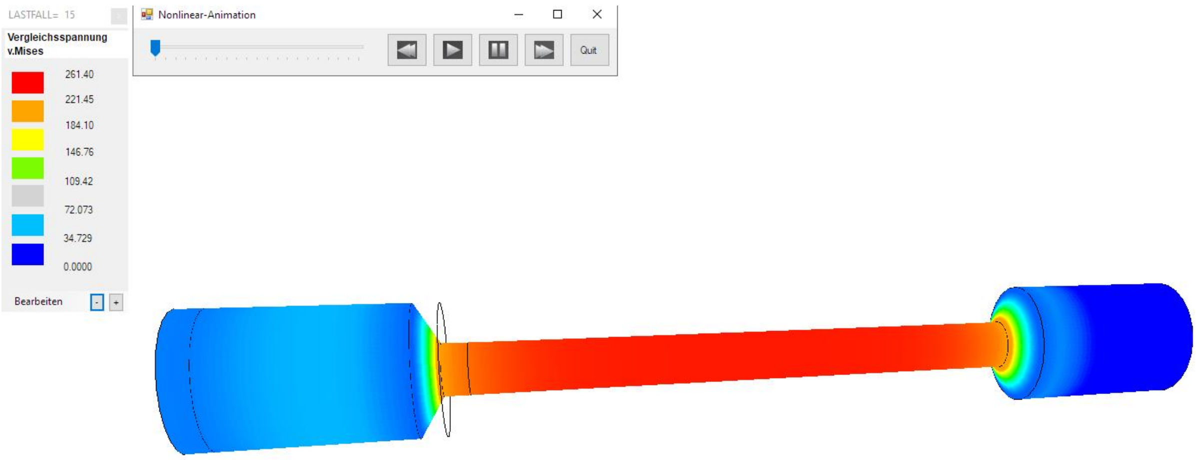
Nonlinear-Animation

Select the “Postprocessing” and “Nonlinear-Animation” tabs to animate all load cases one after the other.



Load Case 15

v.Mises Stress = 261 N/mm²



Displacements in Z direction = 3.73 mm