Part 3 - The first FEM Project - Eccentric Bolt with an Axial Load

(C) 2021
Engineering Office HTA-Software
Maiwaldstraße 24
77866 Rheinau-Germany
Tel. 0049-7844-98641
www.fem-infos.com
info@fem-infos.com

Table of Contents:

Further Parts:

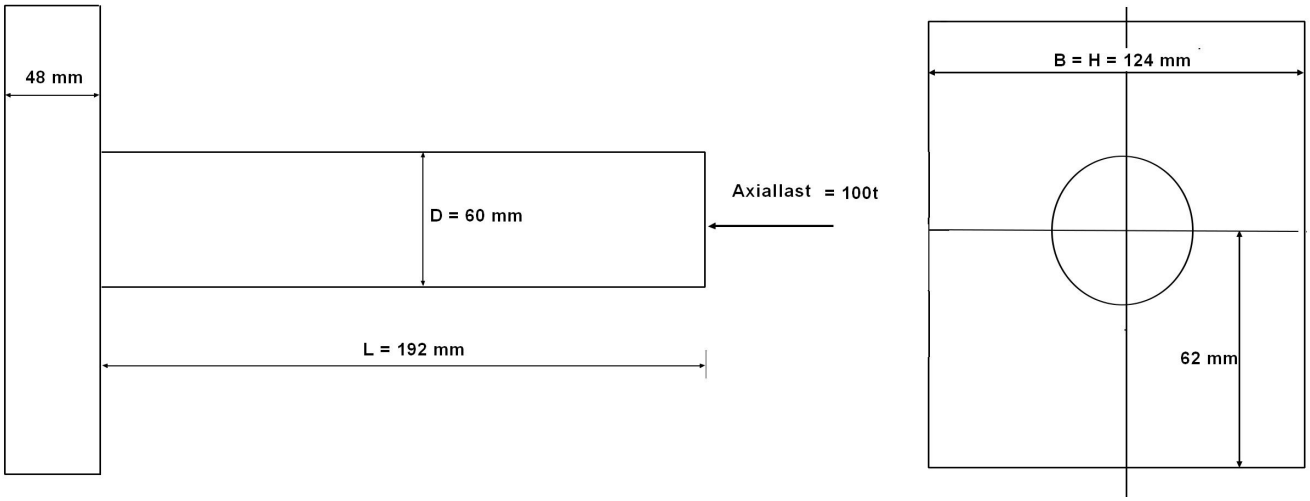
Part 23 - Eccentric Bolt with inserted hole and torsional moment

1. Create the Eccentric Bolt

The eccentric bolt made of steel consists of a cylinder with D = 60 mm and L = 192 mm which is pressed with an axial force of 100t onto a quader with the dimensions 124 mm x 124 mm x 48 mm.

1.1 Exact result

The axial stresses in the cylinder can be calculated exactly with the cross-sectional area A.

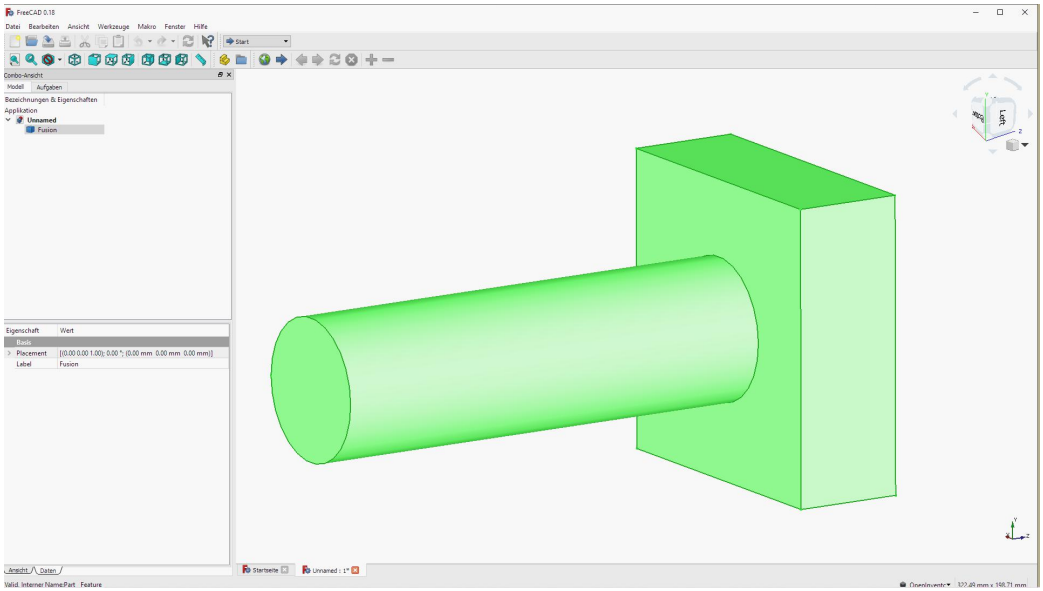


Axial Stress = F_A / A_Z = 1 000 000 N / 3.1416 * D^2 / 4

= 1 000 000 N * 4 / 3.1416 * 60^2 mm^2 = 353.7 N/mm^2

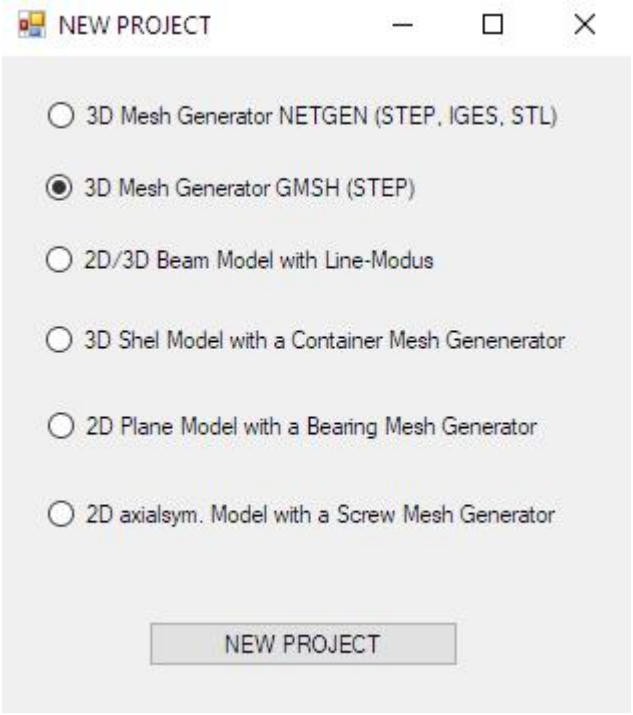
1.2 CAD Model

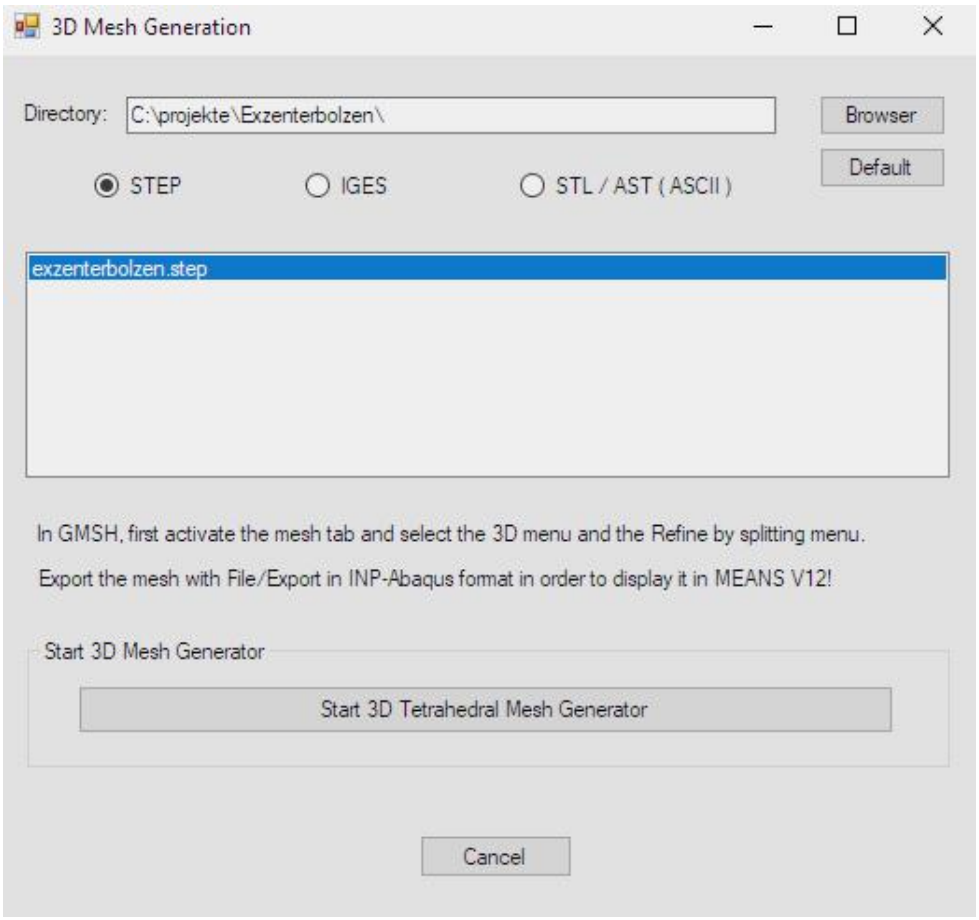
In the CAD system, the eccentric bolt is generated with a cylinder and a quader. After united both solids to a part, the eccentric bolt is saved in STEP format.



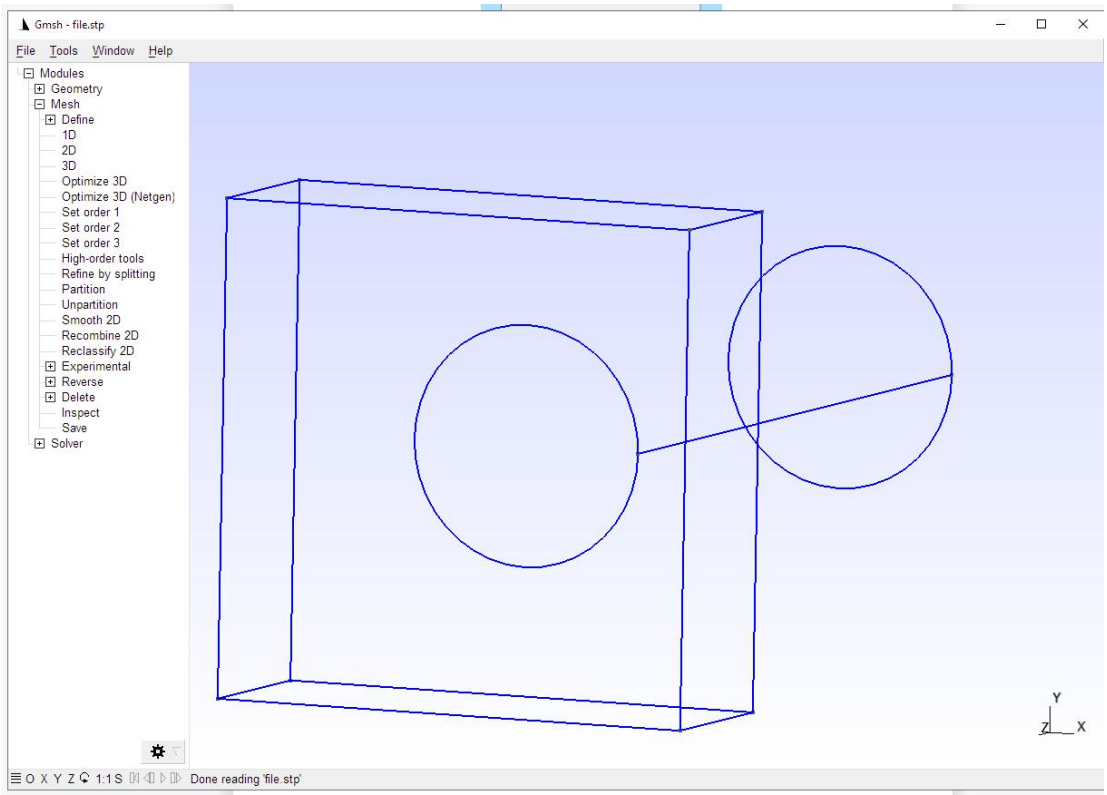
1.3 Generate the FEM Mesh

Start MEANS V12 with the desktop icon  and select the “File” register and the “New” menu and select „3D Mesh Generator GMSH (STEP)“ to generate a FEM-Mesh from the STEP-File „Eccentric_Bolt.STEP” with the mesh generator GMSH.

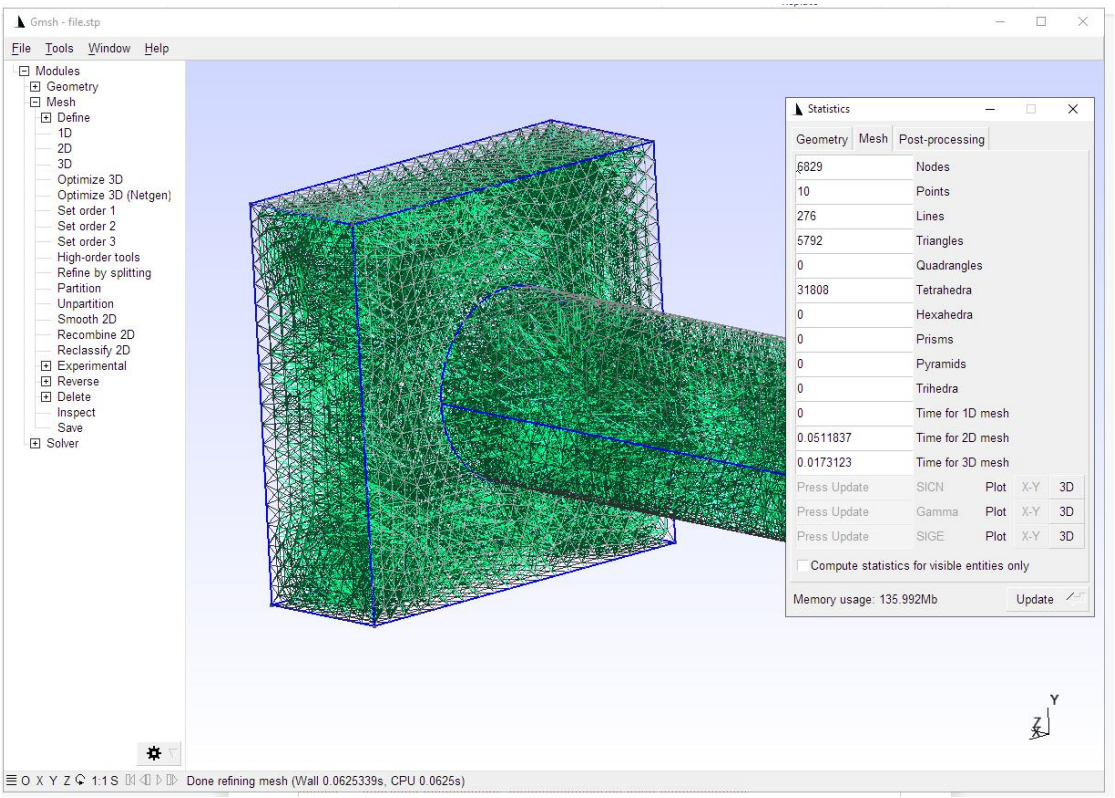




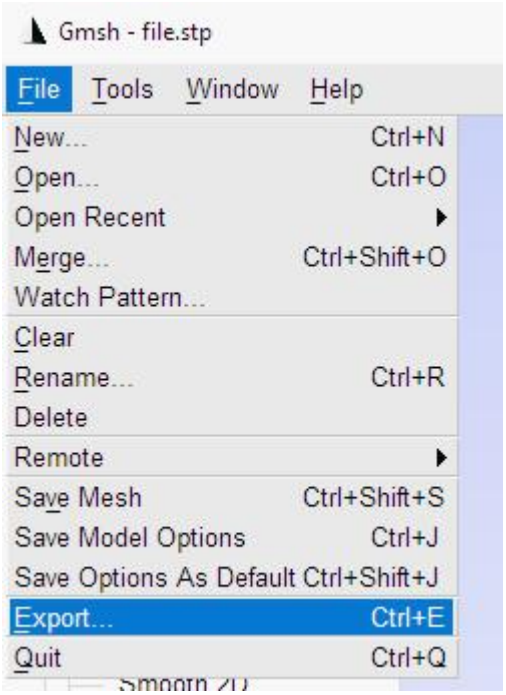
In GMSH, the eccentric bolt is first shown in a blue wireframe. In the menu on the left, first select "3D" and then "Refine by splitting" to refine the FEM mesh 8 times more finely. After 2 clicks you get an FEM mesh with 31 808 tetrahedra.



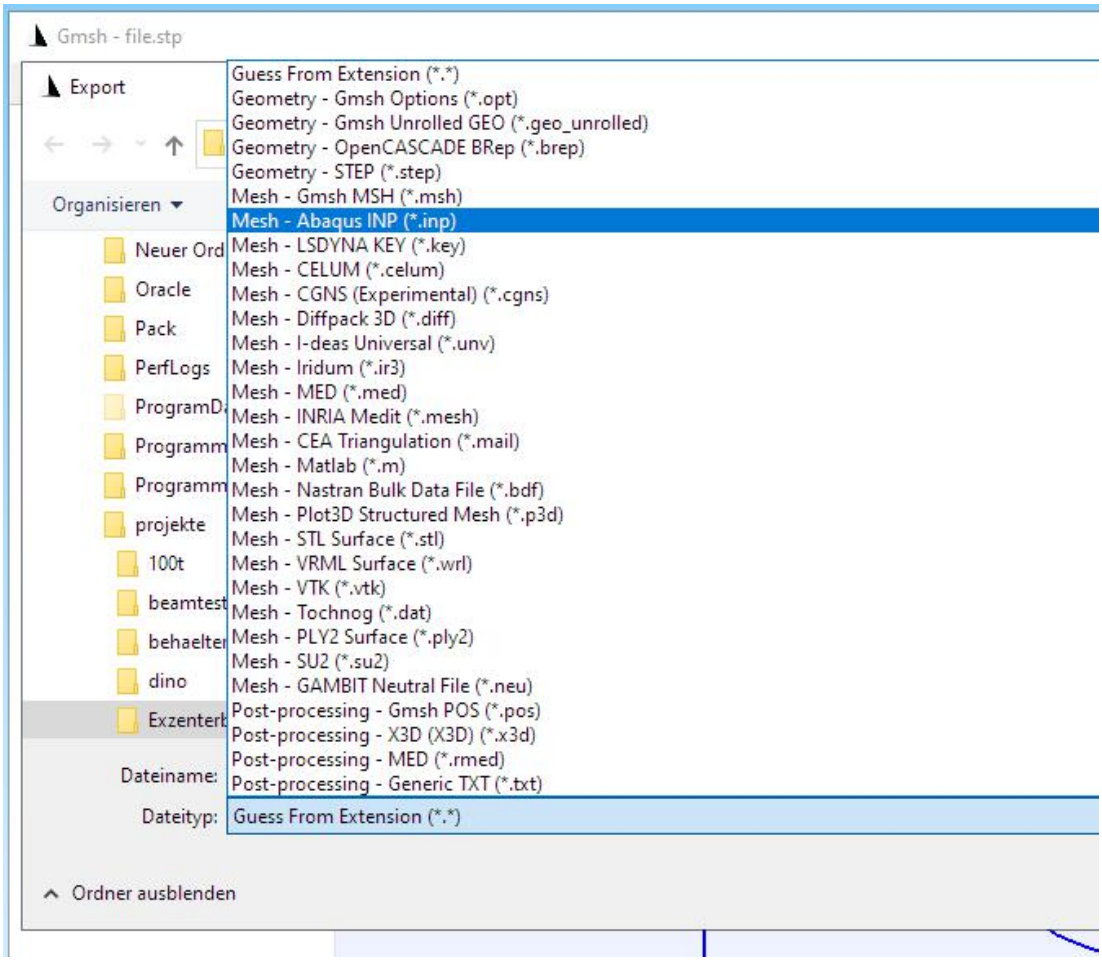
Please also always look at the info line below, if no tetrahedra can be created "Warnings" and "Errors" are displayed. In such cases, unfortunately, GMSH must be terminated and an attempt must be made to mesh with NETGEN.



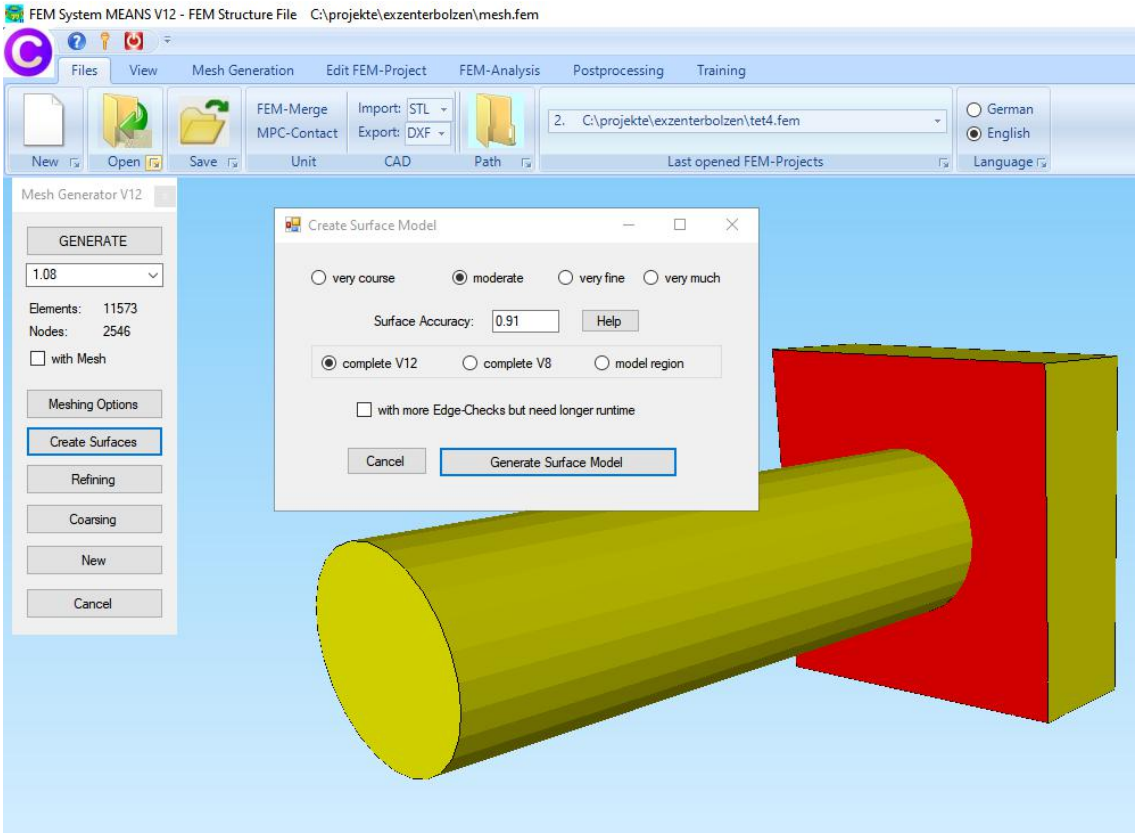
Now export the generated FEM mesh with the menu “File” and “Export” and select the format “Mesh - Abaqus INP (* .inp)”.



If the mesh is saved in the same directory with the name "file", it is automatically imported and displayed by MEANS V12.

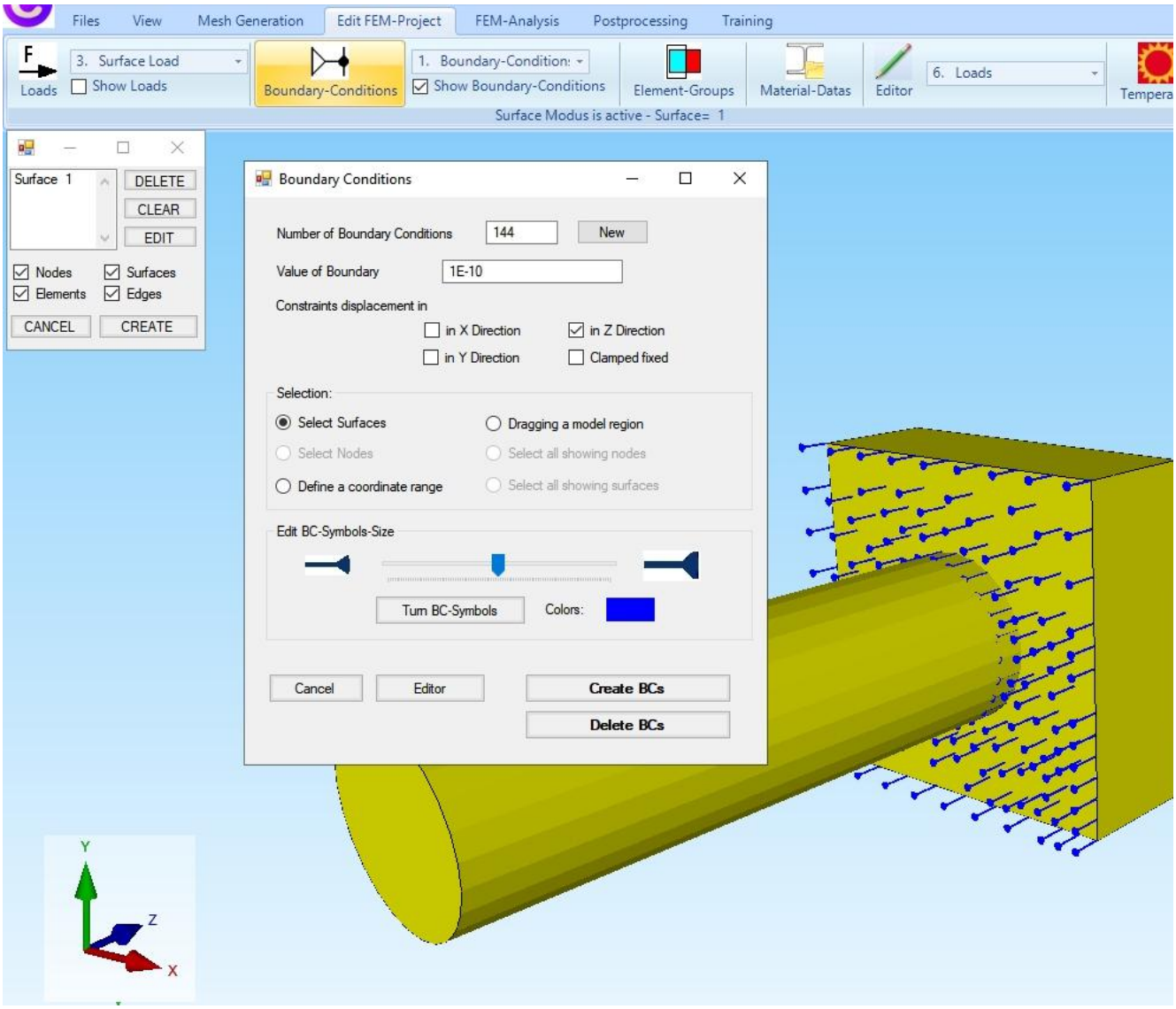


After the GMSH export, select "Generate Surface Model" to be able to select the 8 main surfaces of the eccentric bolt for the surface load and clamping.



1.4 Boundary Conditions

Use the "Edit FEM-Project" register and the menu "Boundary-Conditions" to create with the selection "Surfaces" the BCs in the Z-direction on the front side of the quader with the Surface 1. In the selectbox choose "Create" to create 144 Boundary Conditions.

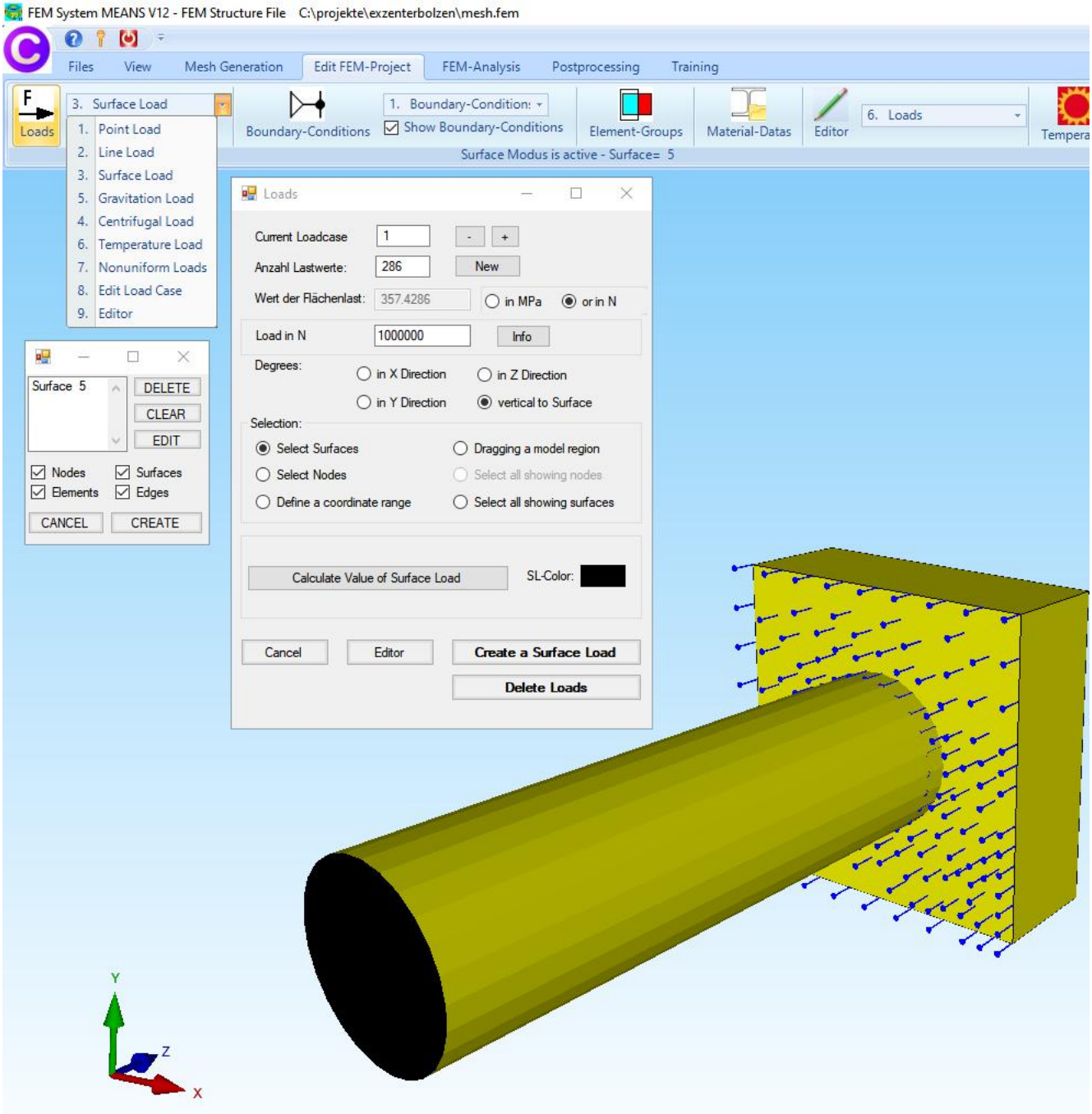


1.5 Axial Load

Choose the “Edit FEM project” register and the “Surface Load” menu.

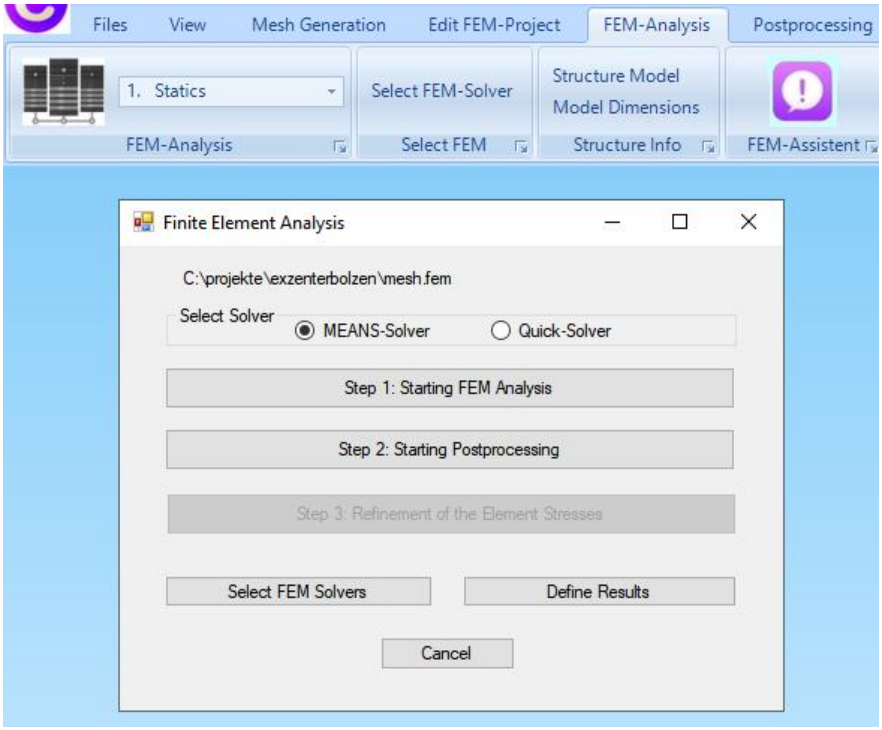
In the next dialog box select Load case “1” and the option “or in N” and enter the value “1000000” and the degree of freedom “vertical to Surface”.

With the selection “Surfaces” and the “Create a Surface Load” menu, generate a surface load by clicking the front of the cylinder with the Surface 5. In the selectbox choose “Create” to create 286 loads.



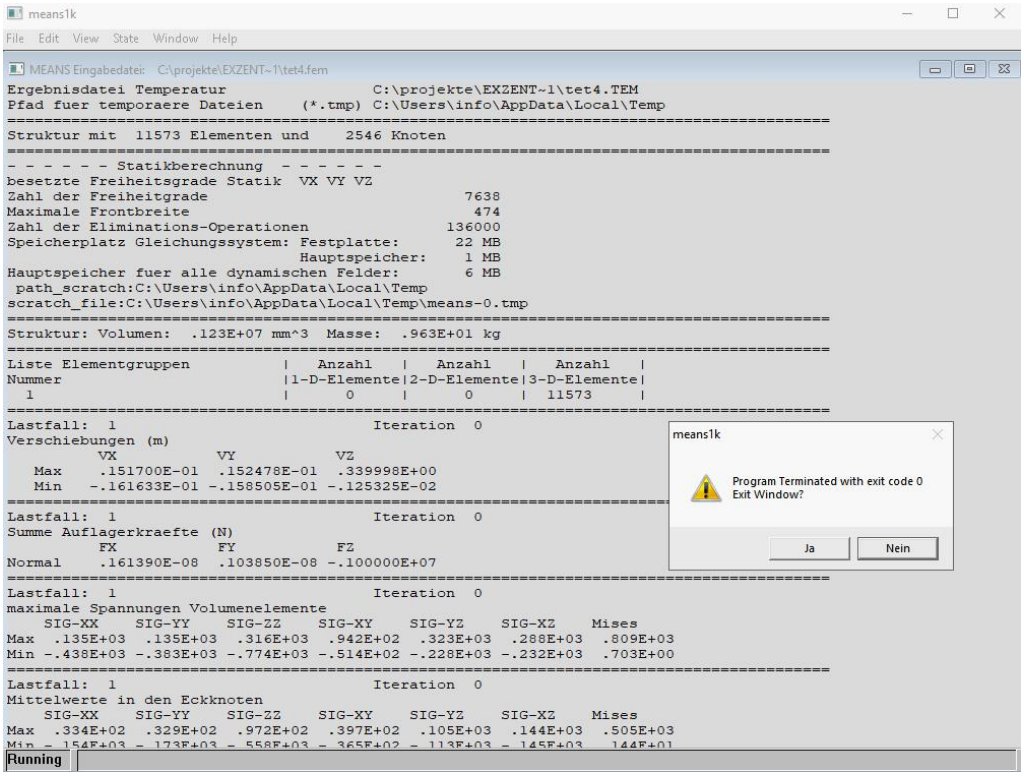
1.6. FEM-Analysis

Choose the Register „FEM-Analysis“ and the Icon  to start a FEM-Calculation with the MEANS-Solvers or the Quick-Solver.



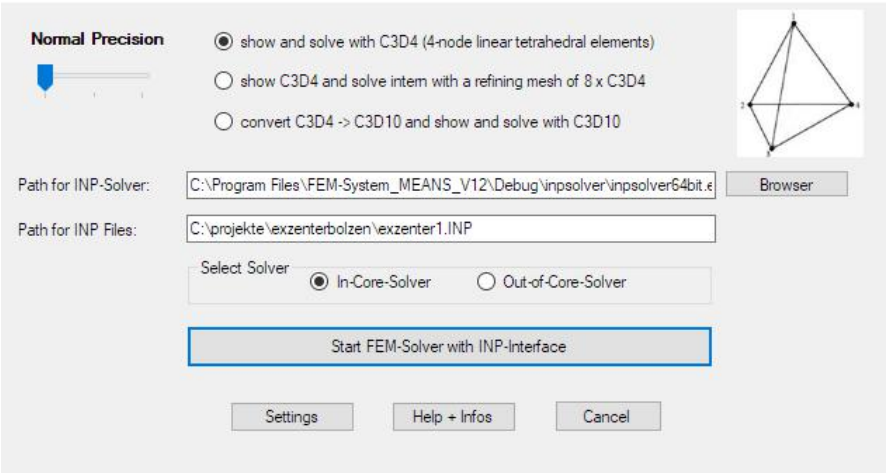
1.6.1 MEANS-Solver

The MEANS-Solver developed by HTA-Software, which needs a longer time to calculate than the Quick-Solver, is started.



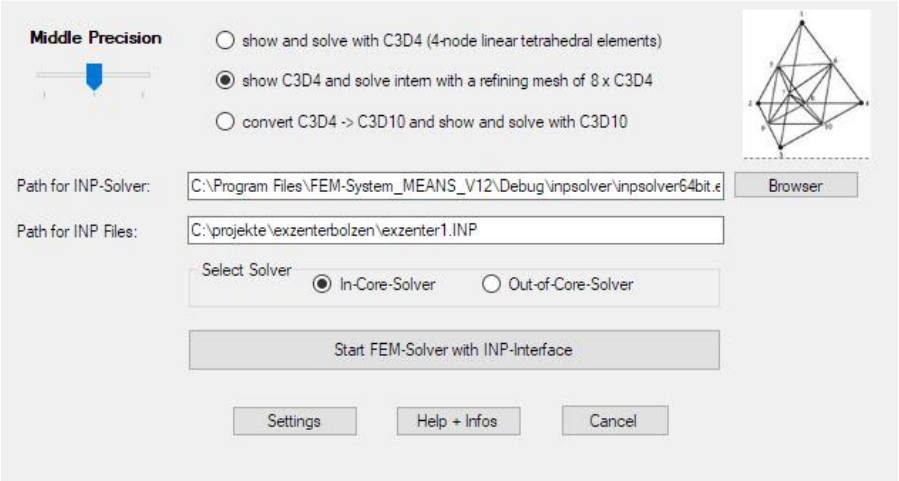
1.6.2 Quick-Solver with Tetrahedral Element TET4

The Quick-Solver with the linear Tetrahedral Element TET4 is started.



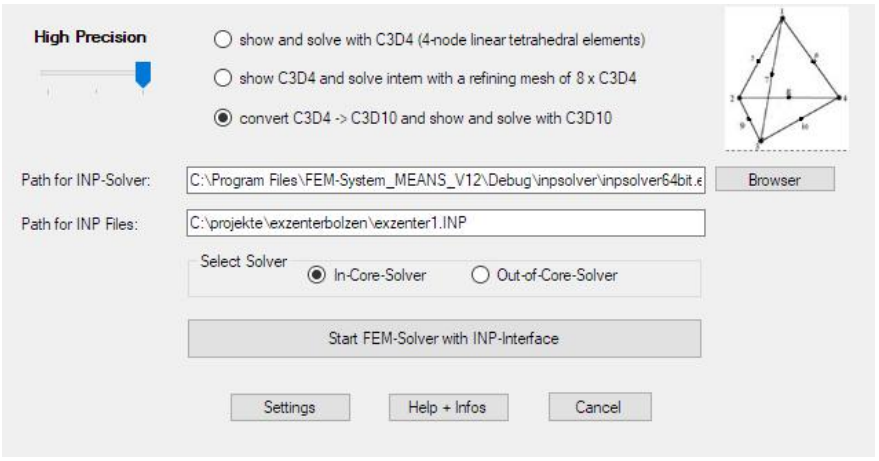
1.6.3 Quick-Solver with Tetrahedral Element TET4X8

The Quick-Solver with the Tetrahedral Element TET4X8 developed by HTA-Software is started and can calculate 8 times more accurately than TET4.



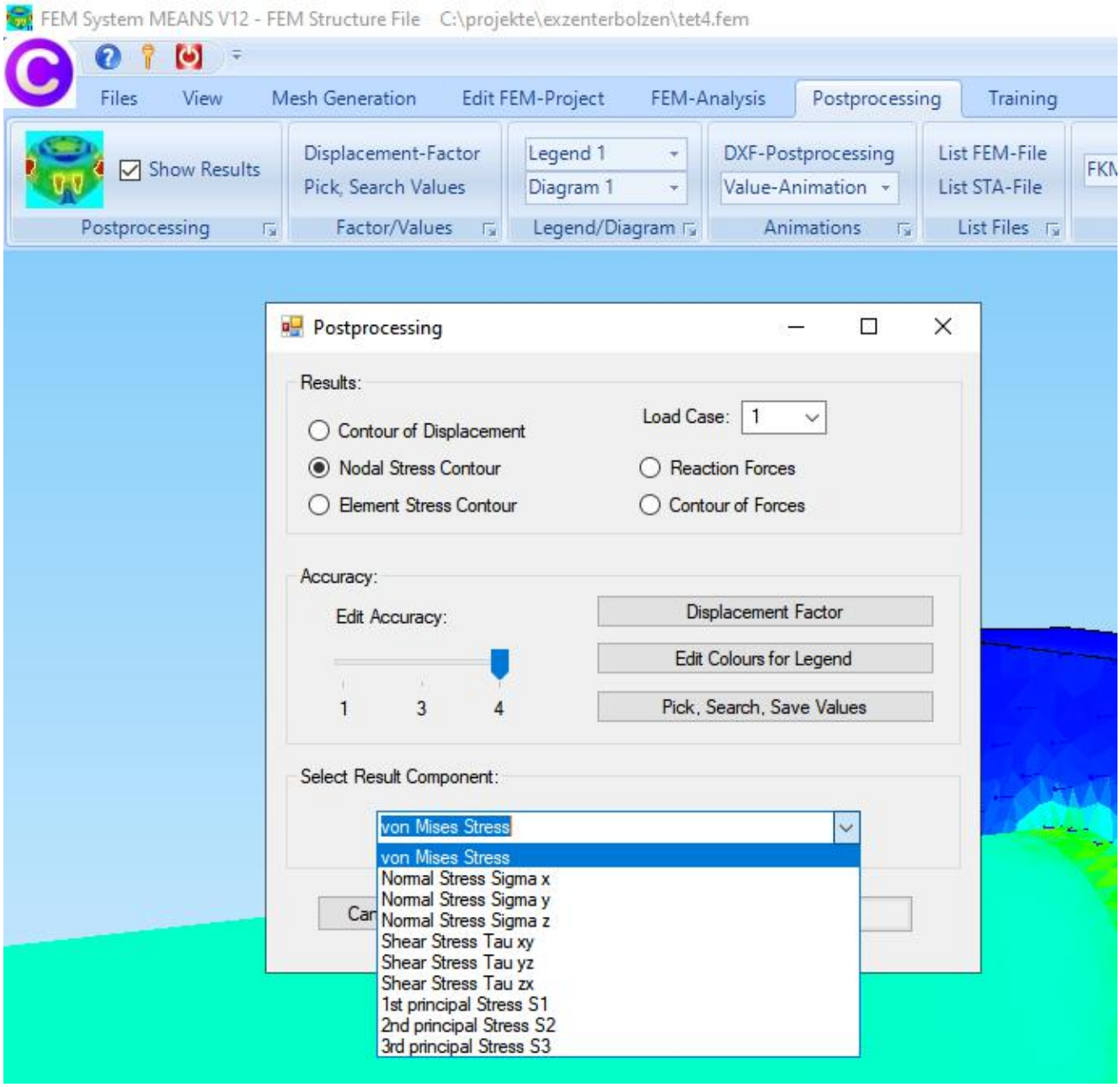
1.6.4 Quick-Solver with Tetrahedral Element TET10

The Quick-Solver with the quadratic Tetrahedral Element TET10 is started.



1.7 Results

Choose the Register „Postprocessing“ and the Icon  to display the results in color as a displacement or stress distribution.

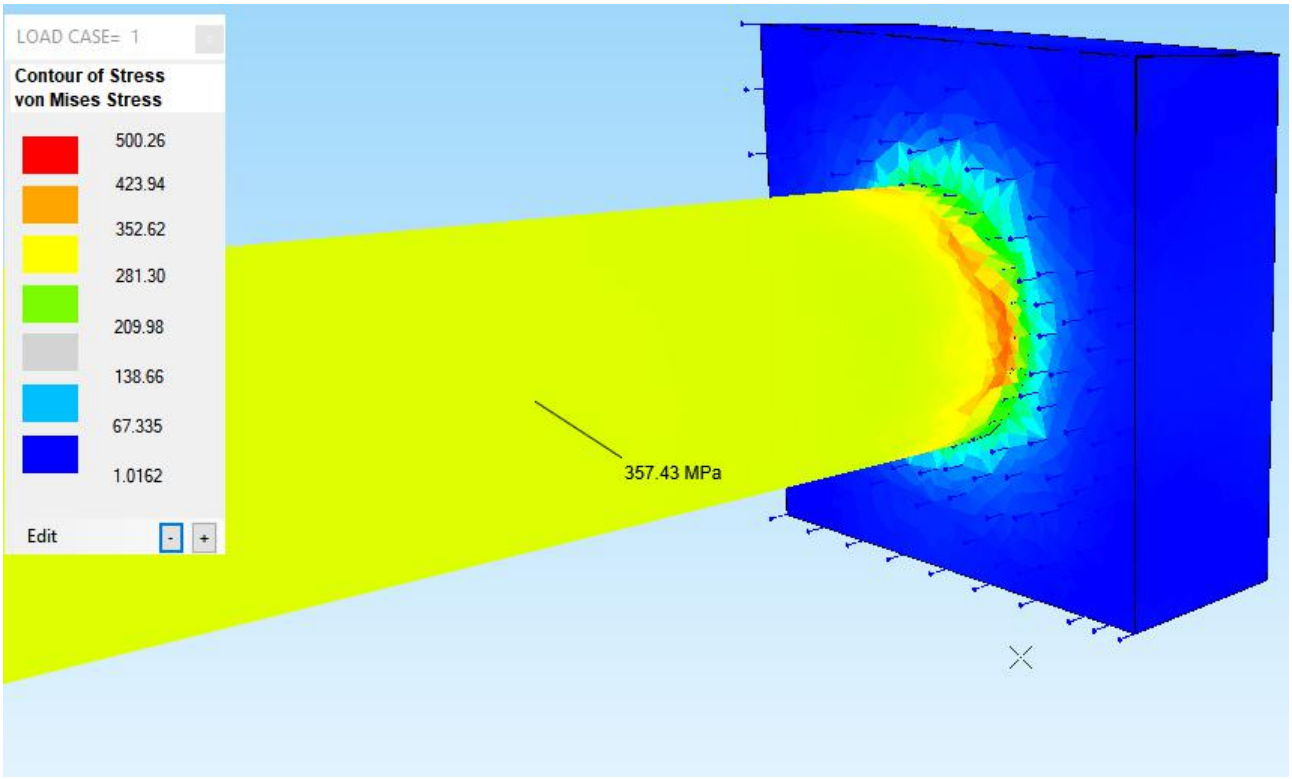


A comparison of the results with the different tetrahedral elements of MEANS V12 follows:

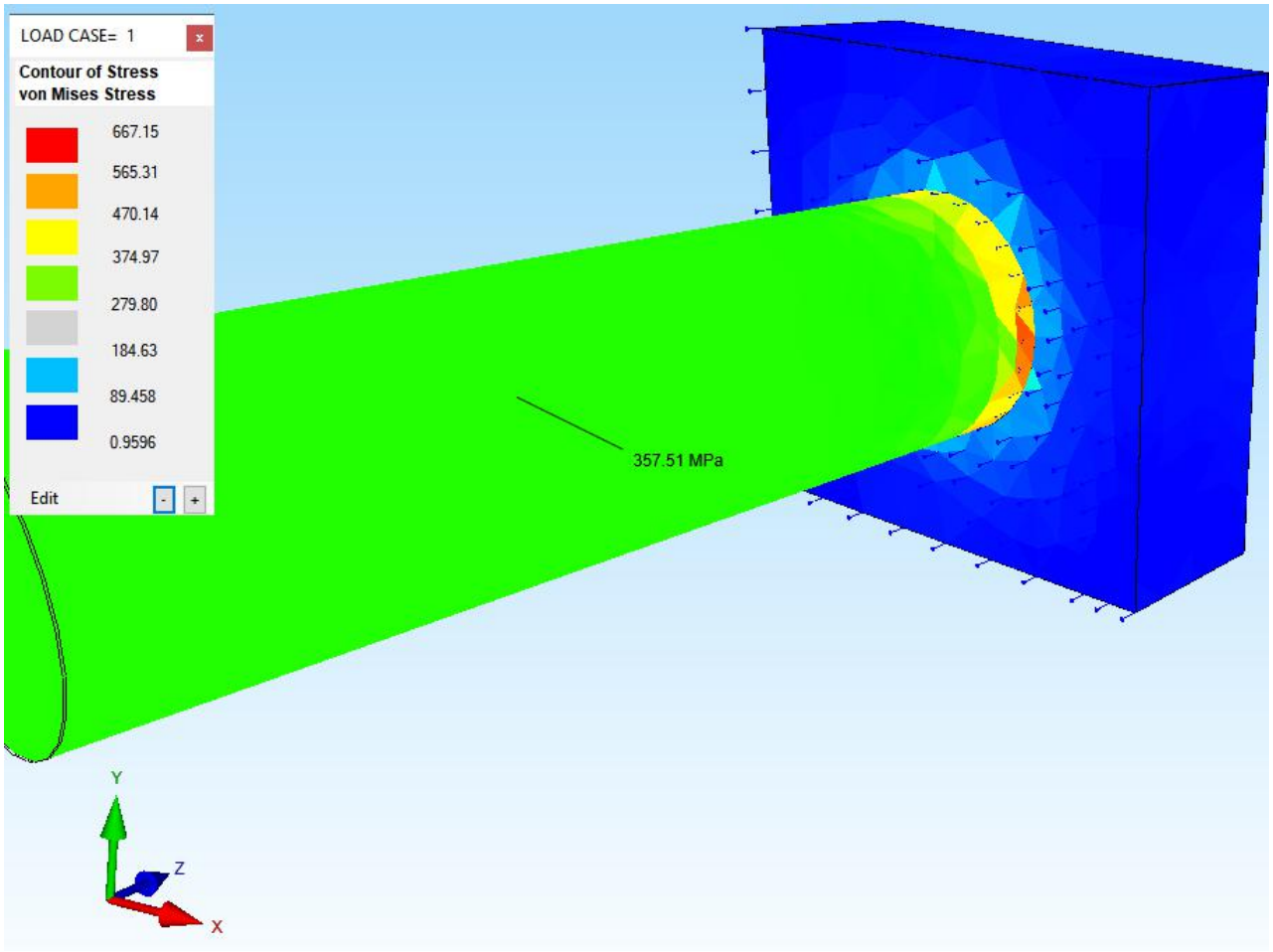
- TET4
- TET4X8
- TET10

shows that the accuracy of TET4 to TET10 is getting more and more better, but the computing time is getting longer and longer.

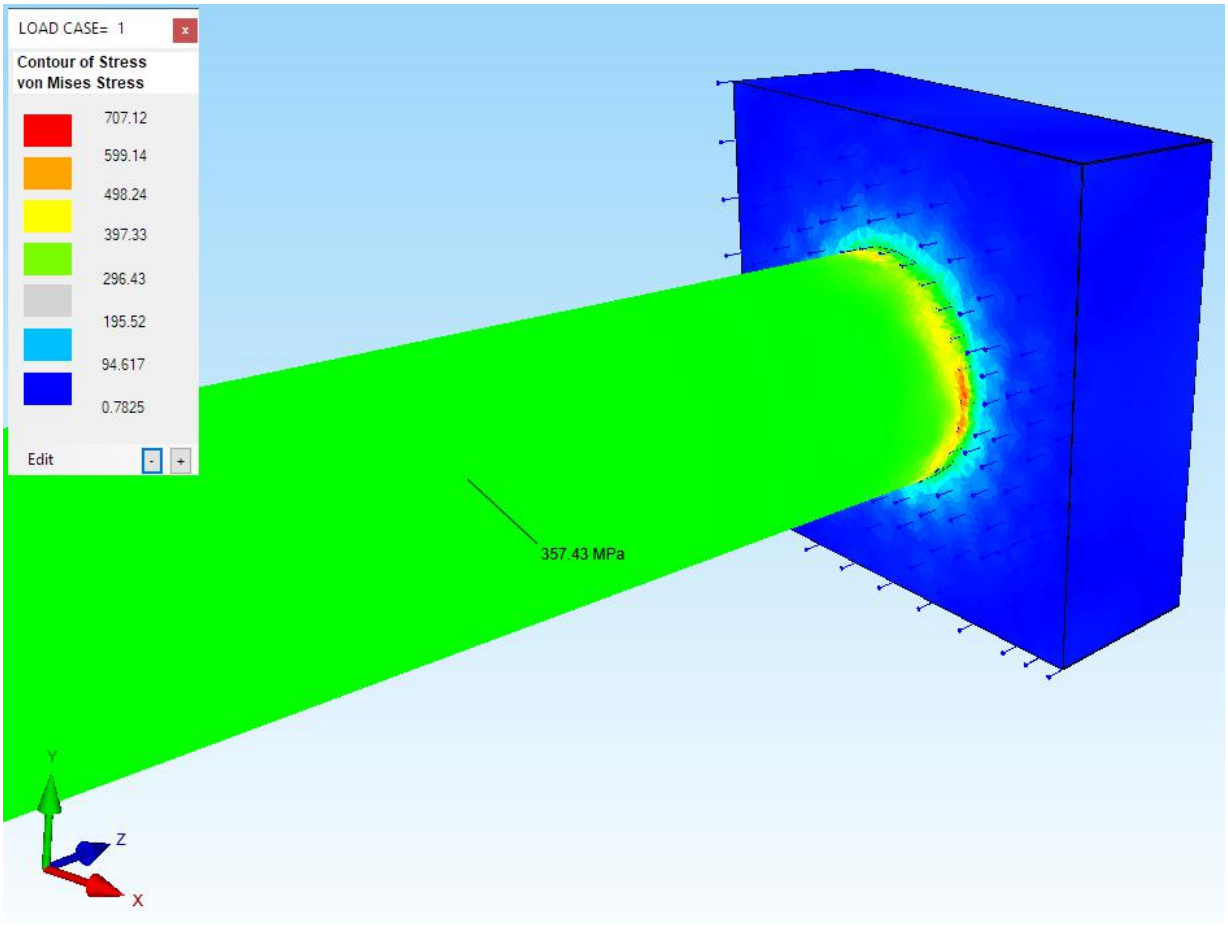
v.Mises Nodal Stresses of TET4



v.Mises Element Stresses of TET4



v.Mises Nodal Stresses of TET4X8



v.Mises Nodal Stresses of TET10