# Part 23 - Eccentric Bolt with a local Refinement and a Torsional Moment



without Refinement

additional Node-List

with Refinement

FEM-System MEANS V12
Demo
www.femcad.de
www.fem-infos.com

**(C) 2021**
**Ing.Büro HTA-Software**
**Maiwaldstraße 24**
**77866 Rheinau-Germany**
**www.femcad.de**
**www.fem-infos.com**

# 1.0  Part 23 - Eccentric Bolt with a local Refinement and a Torsional Moment

A hole with D = 50 mm is inserted into the eccentric bolt from Part 3. The hole is created by re-meshing the FEM mesh with an additional node list. With this new mesh generation, areas can be specifically refined or new element groups can be created.

## 1.1  Exact Results

The axial stress can be calculated exactly using the axial force and the cross section of the circular ring and the torsional stress using the torsional moment and the polar moment of inertia.
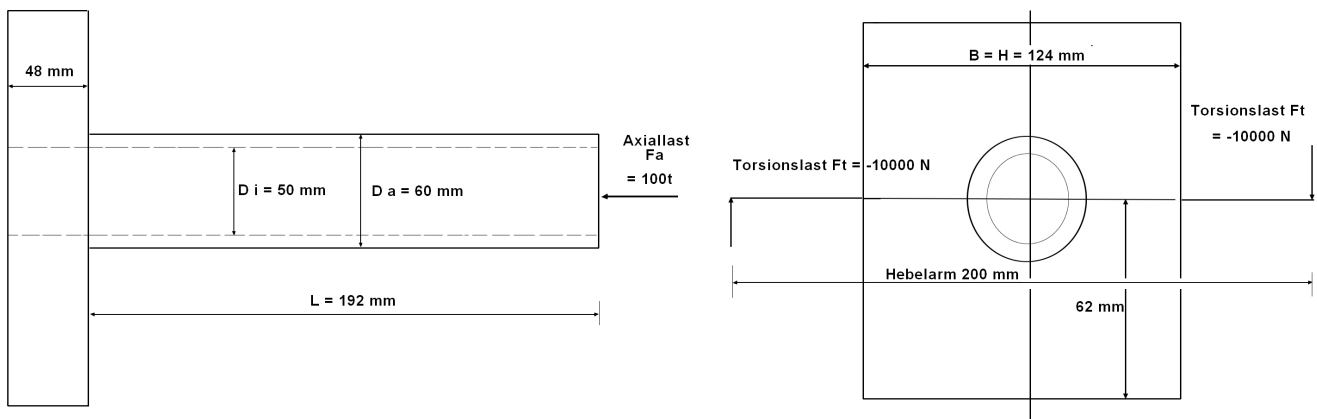


**Axial Stress     =  Axial Force / cross section of circular ring**

$$= \text{1 000 000 N} * 4 \ / \ 3.14159 * (60^2 - 50^2) \ \text{mm}^2 = \ 1157.49 \ \text{N/mm}^2$$

**Torsional Stress = Torsional Moment / polar moment of inertia**

$$= 10000 \ \text{N} * 200 \ \text{mm} * 16 \ / \ 3.14154 * (60^3 - 50^3) \ \text{mm}^3$$

$$= 111.93 \ \text{N/mm}^2$$

## 1.2  Generate the Eccentric Bolt

Select "New" and "3D Tetrahedral Mesh Generator MEANS V12" and generate an FEM mesh from 2454 tetrahedra with the STEP file "Exzenterbolzen.step" from Part 3 with the mesh density "1.6". Then the new dialog box for the local mesh refinement is called up with the menu "Refining" in order to generate an 8x finer FEM mesh with 19632 tetrahedra with the menu "Refine all Elements to 8x TET Mesh". Please save the FEM file "tet4x8.fem" for later.
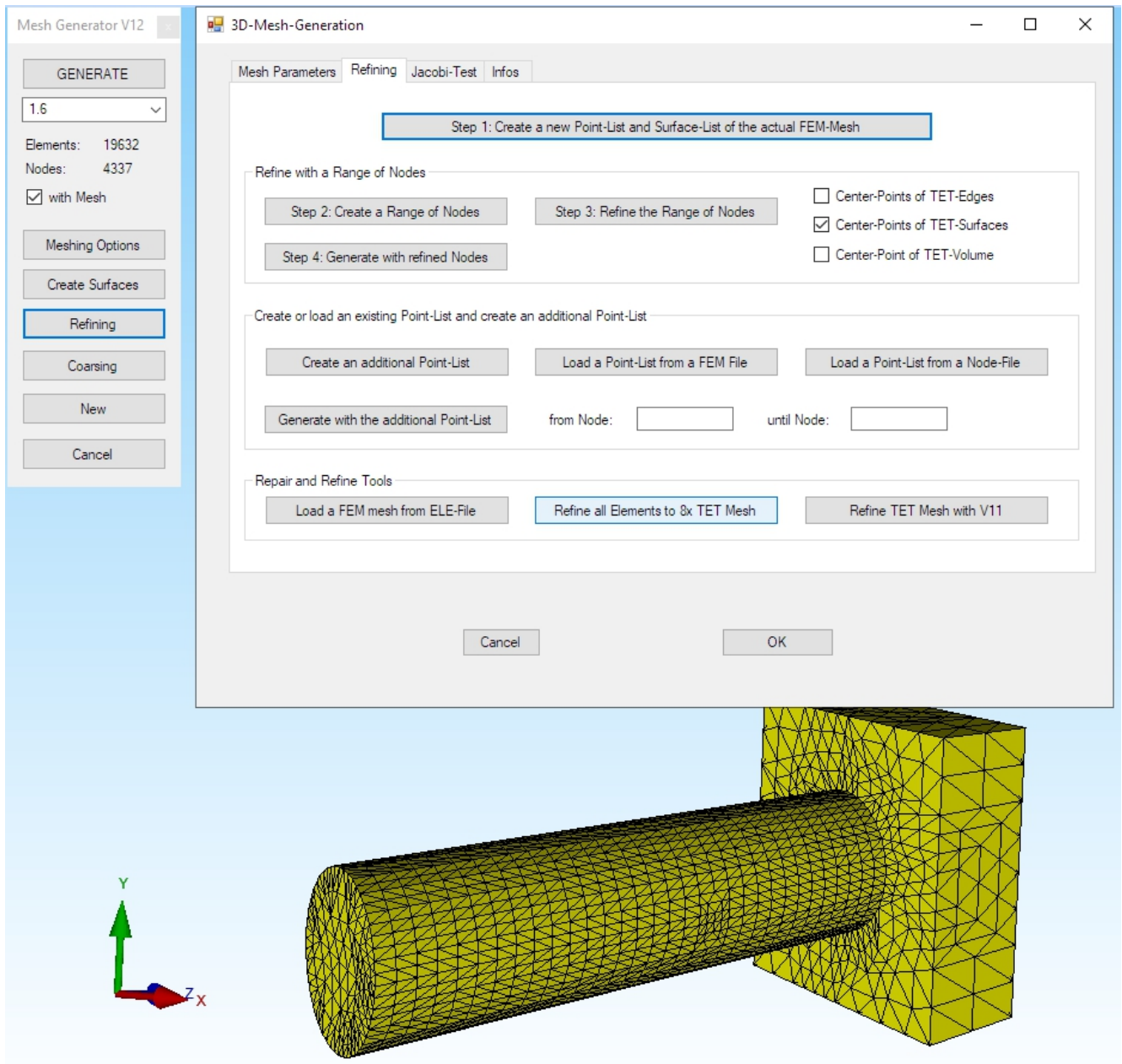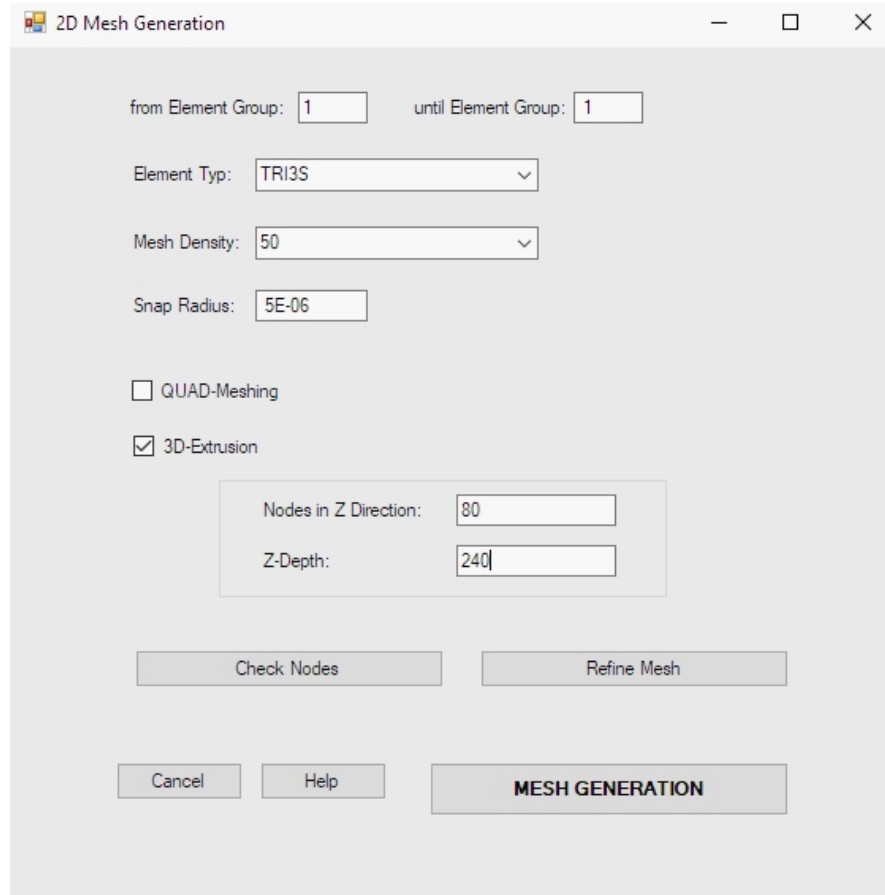
## 1.3 Create a 3D Circular Ring

### Create 2D Circular Ring

Select the register "File" and "New" as well as "2D/3D Beam Model with Line-Modus" to create a 2D circular ring with Ra = 30 mm and Ri = 25 mm.
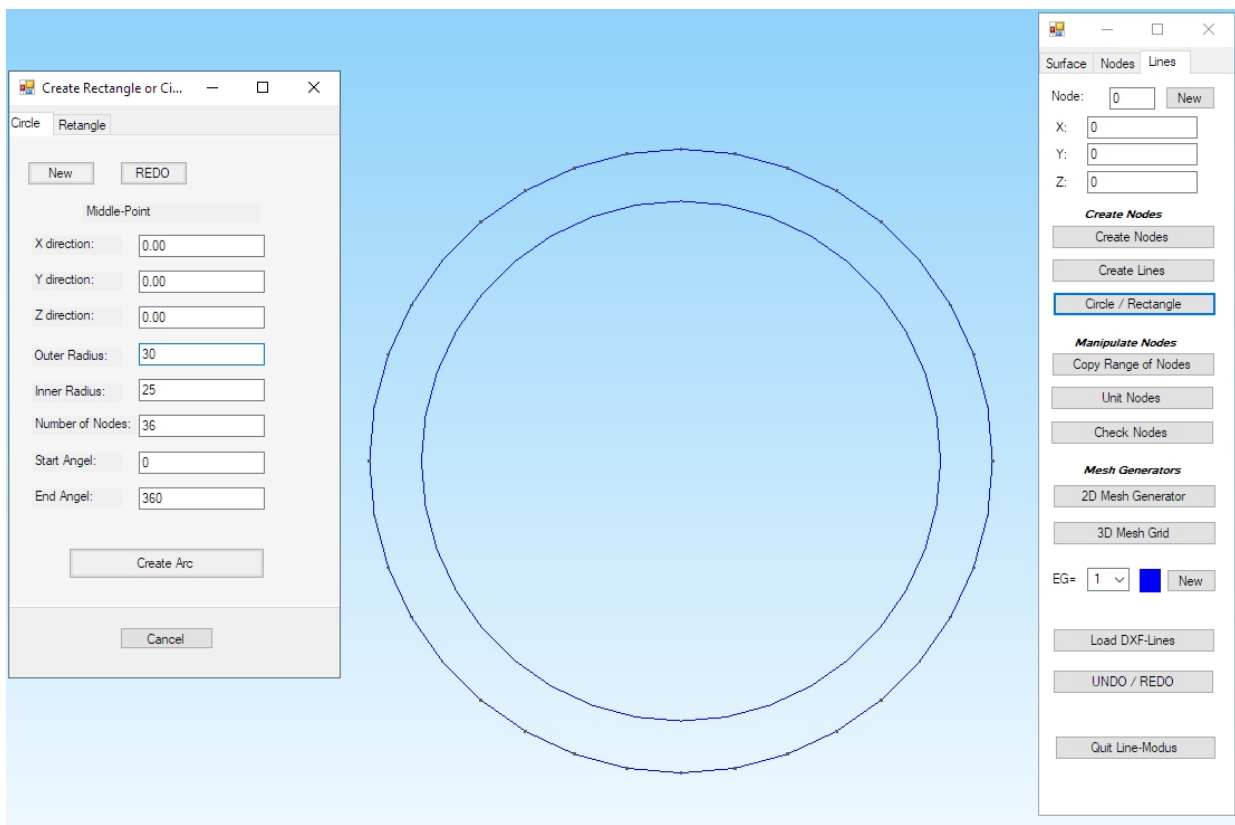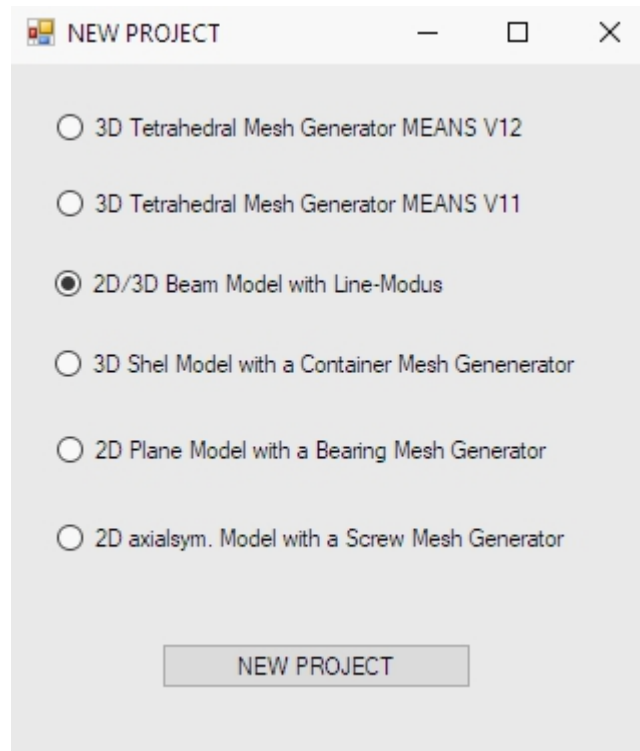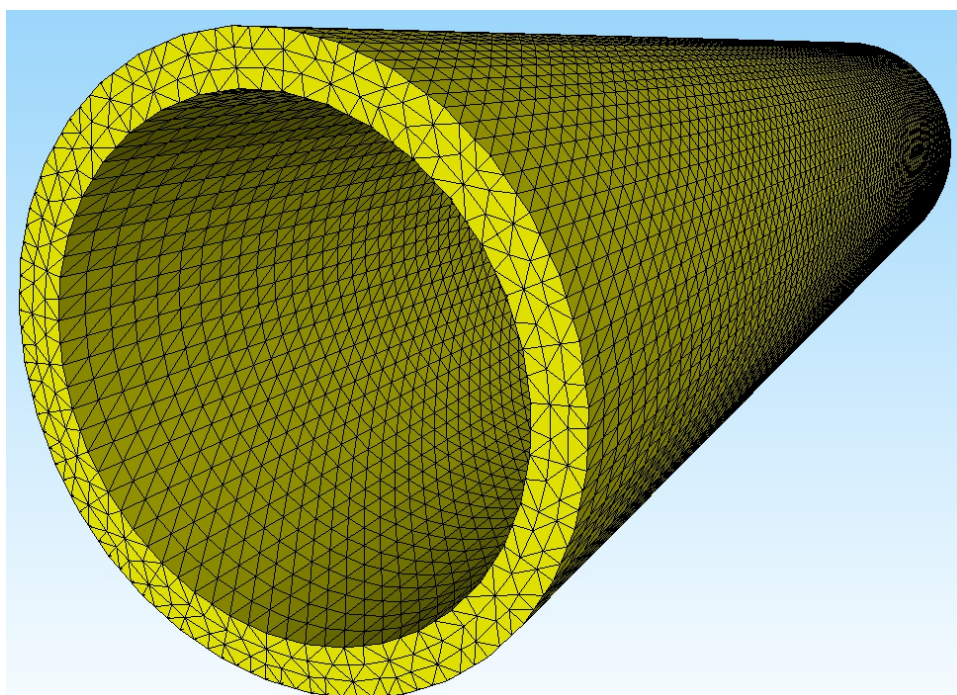
**Create a 3D circular ring**

Select "2D Mesh Generator" in the menu on the right and generate a 3D pentahedron model with a mesh density in the X and Y direction of "50" and the "3D-Extrusion" option.



The new FEM model with a Z-Depth of "240" mm and „Nodes in Z Direction = 80" consists of 36972 PEN6 elements and 25920 nodes and is saved under the name "Circular Ring.fem".

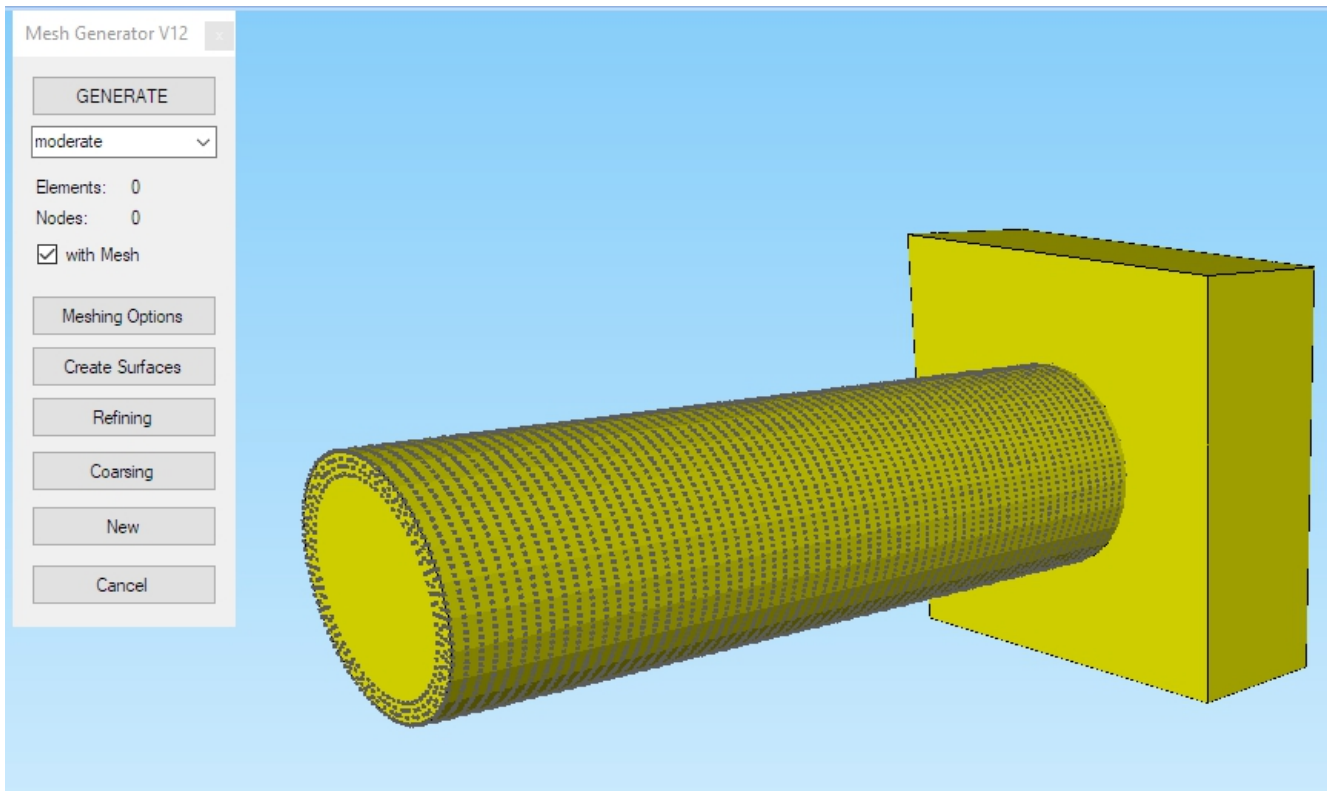## 1.4 Generate FEM mesh with an additional Node-List

Reload the refined Eccentric Bolt "tet4x8.fem" and select the dialogbox for Local Refinement with the "Mesh Generation" and "Local Refinement" tab.

Select "Step 1: Create a new Point-List and Surface-List of the actual FEM-Mesh" to save the Node List of the Eccentric Bolt for the mesh generator.
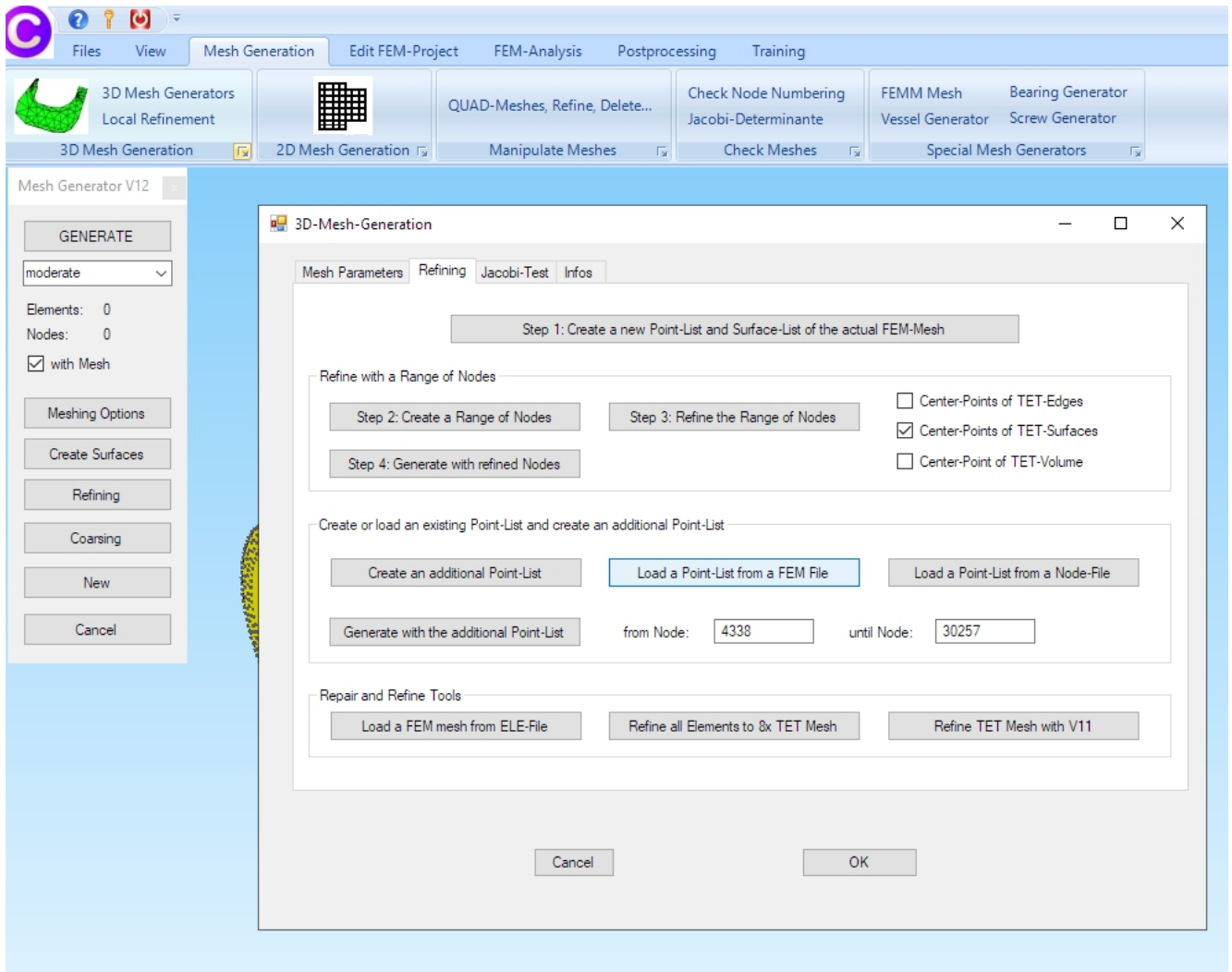


Then select the menu "Load a Point-List from a FEM File" and insert the FEM file "Circular Ring.fem". The nodes of the Circular Ring are now displayed in the rendering or wire frame model.

In addition, the additional nodes from 4338 to 30257 are entered in the two node fields, which can also be edited.
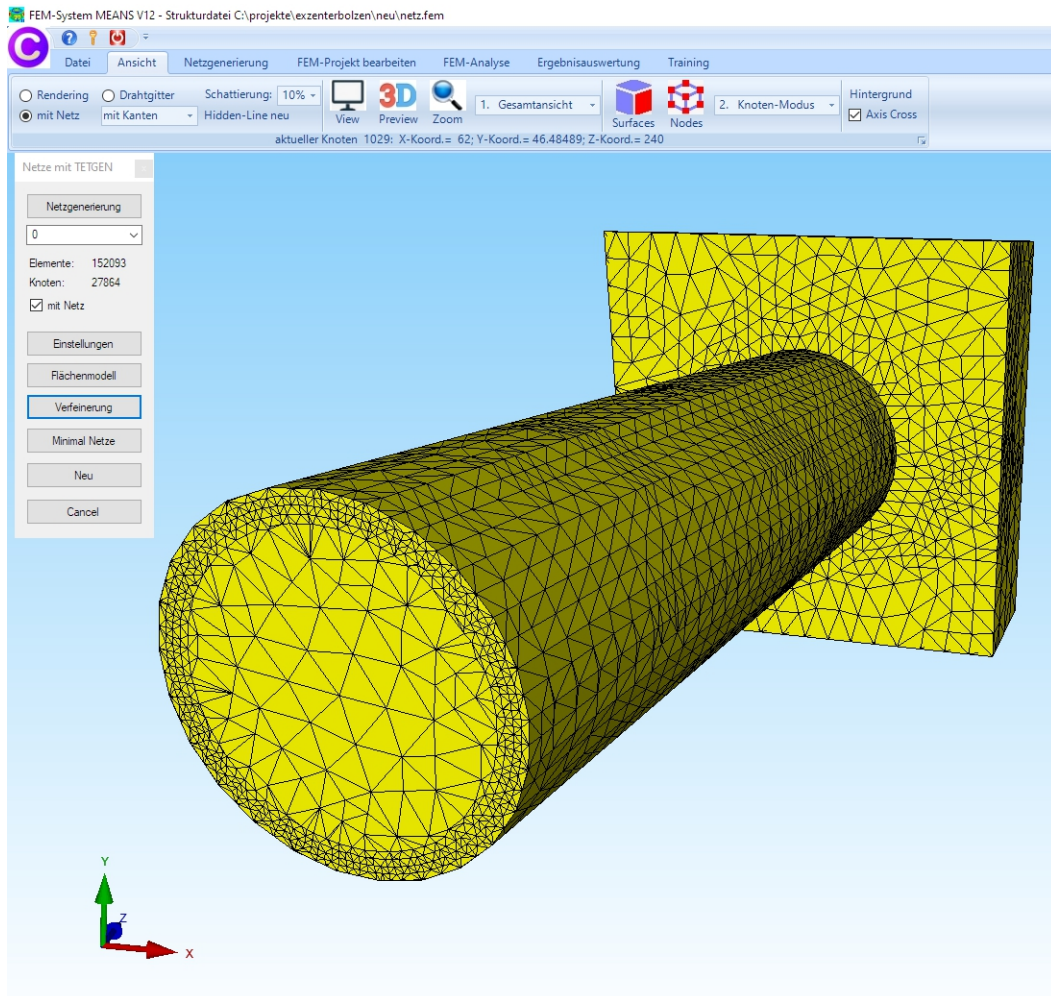
### Generate with the additional Node-List

Select the menu "Generate with the additional Node-List" to re-mesh the FEM mesh of the Eccentric Bolt with the Circular Ring as an additional Node-List.
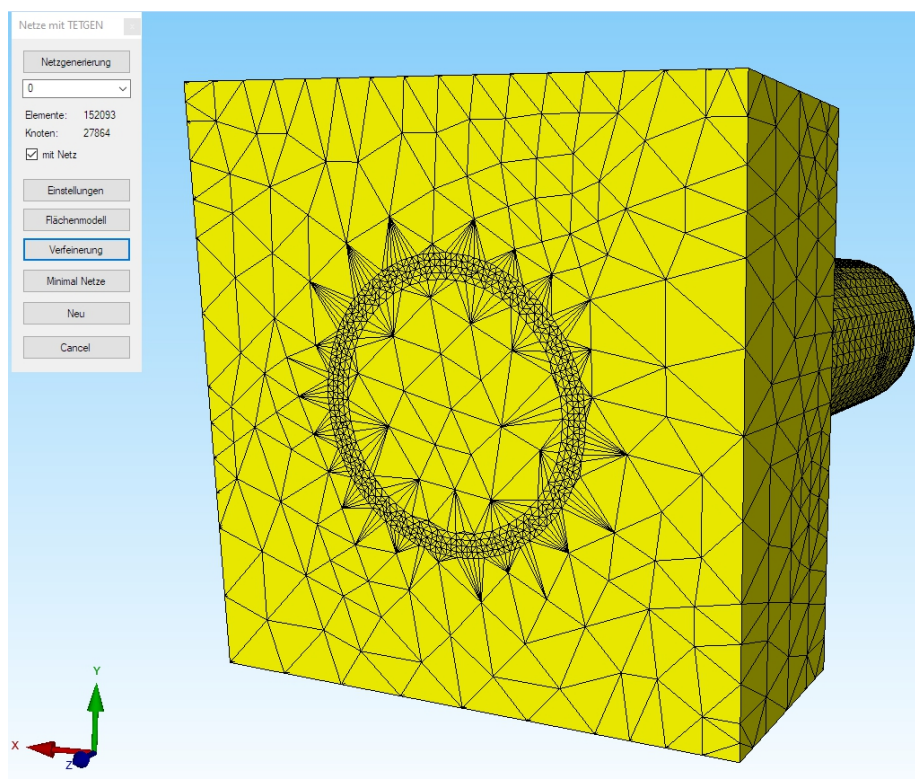
After about 10 minutes of computing time, an FEM model with 140 199 TET4 elements and 24 630 nodes generated and displayed.

The circle hole only goes through the entire eccentric bolt if the circular ring is exactly 240 mm long. Annular nodes that lie outside the eccentric bolt are not generated.

## 1.5 Create circular element group 2

Select the register "Edit FEM-Project" and "Element-Groups" and the menu "Create Element-Groups". In the next dialogbox, select the option "Create element group with a Circular Arc" and the button "Create Element Group" to create circular element group 2 with an outer radius of "25.505" and a very high grid of "1500".
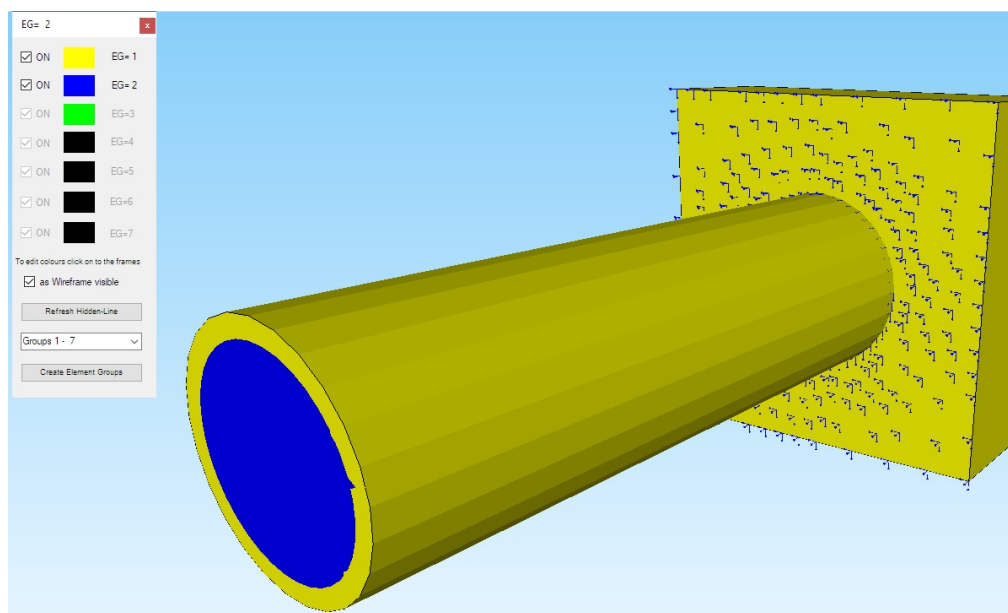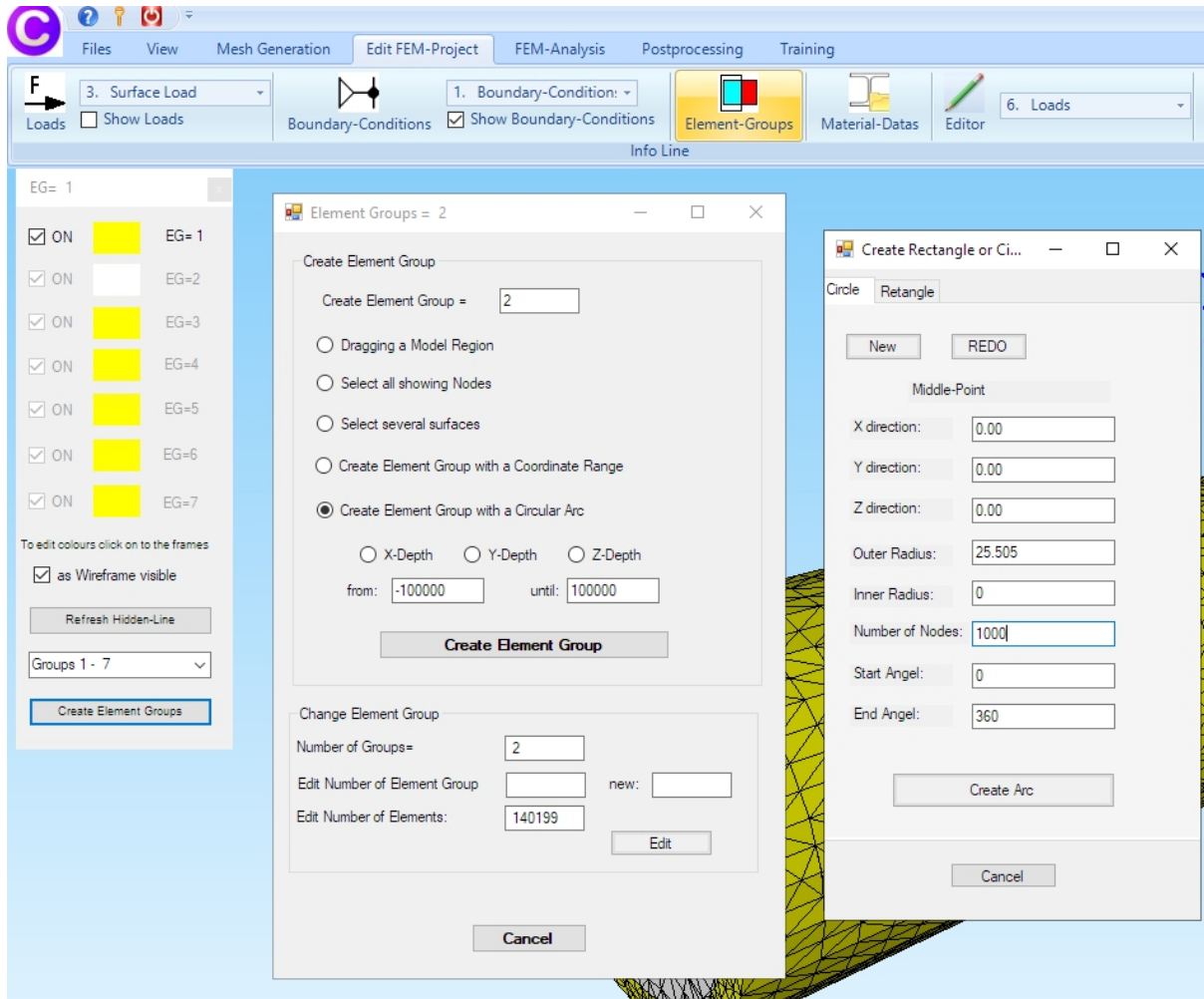
### 1.6 Simulate Drilling

Element group 1 is already preset for "Steel". In order to simulate the drilling, the material data of element group 2 are simply set to a very small value of "0.1". To do this, select the "Edit FEM Project" tab and "Material Data".
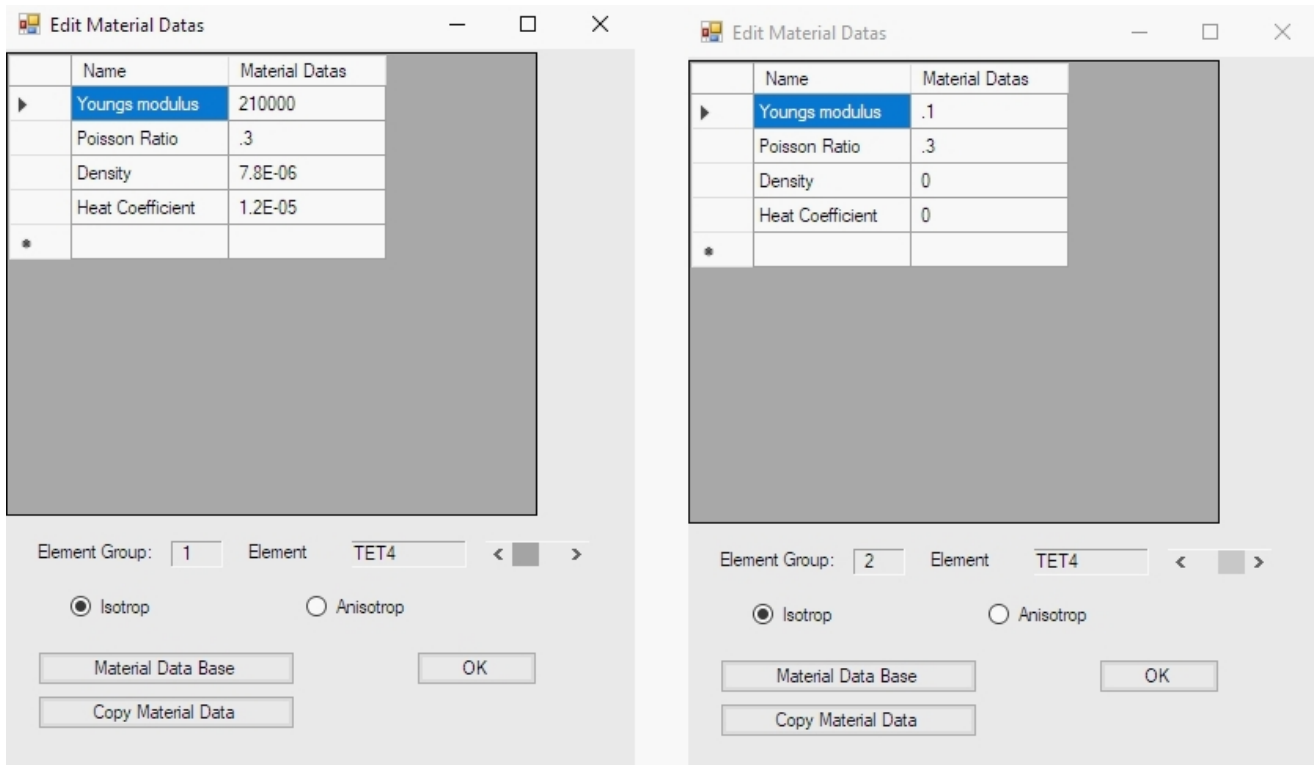
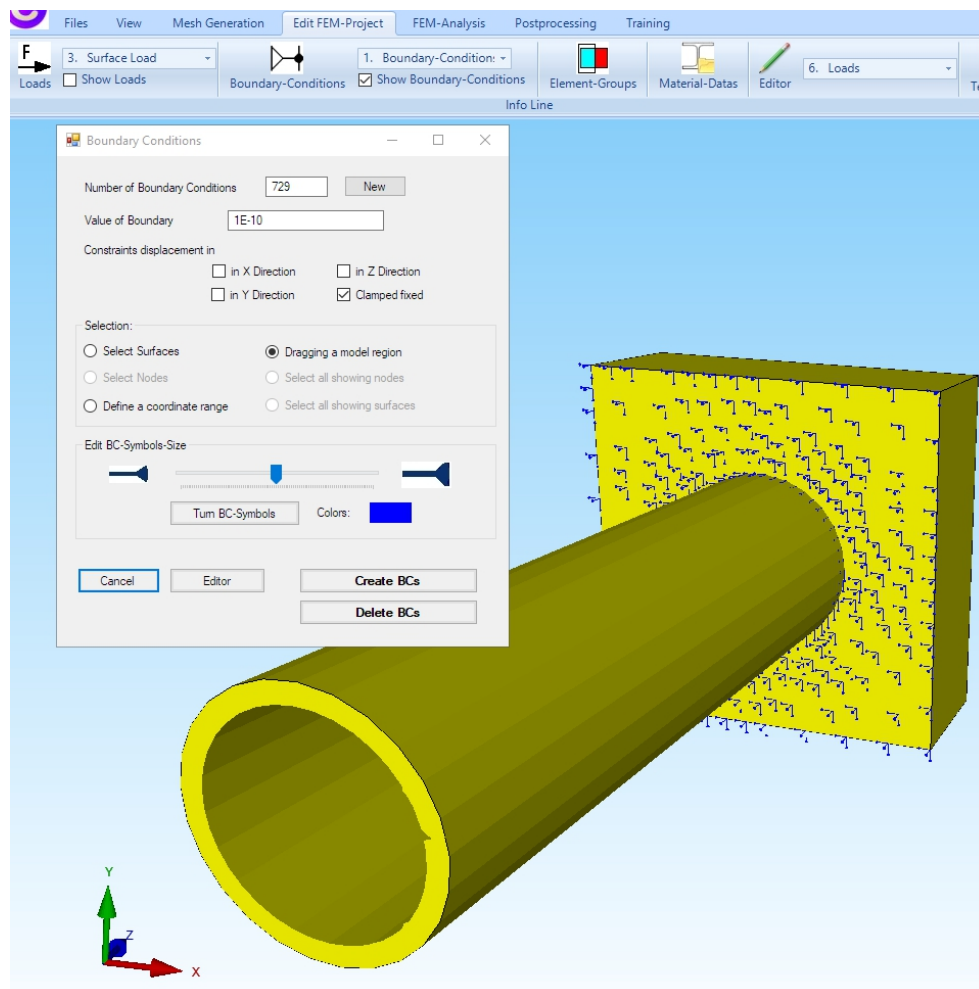

### 1.7 Delete element group 2

To create the hole, element group 2 must be deleted. Select the "Mesh Generation" tab and "Quad Meshes, Refine, Delete" and then "Delete" again to delete element group 2 with a subsequent mesh check.
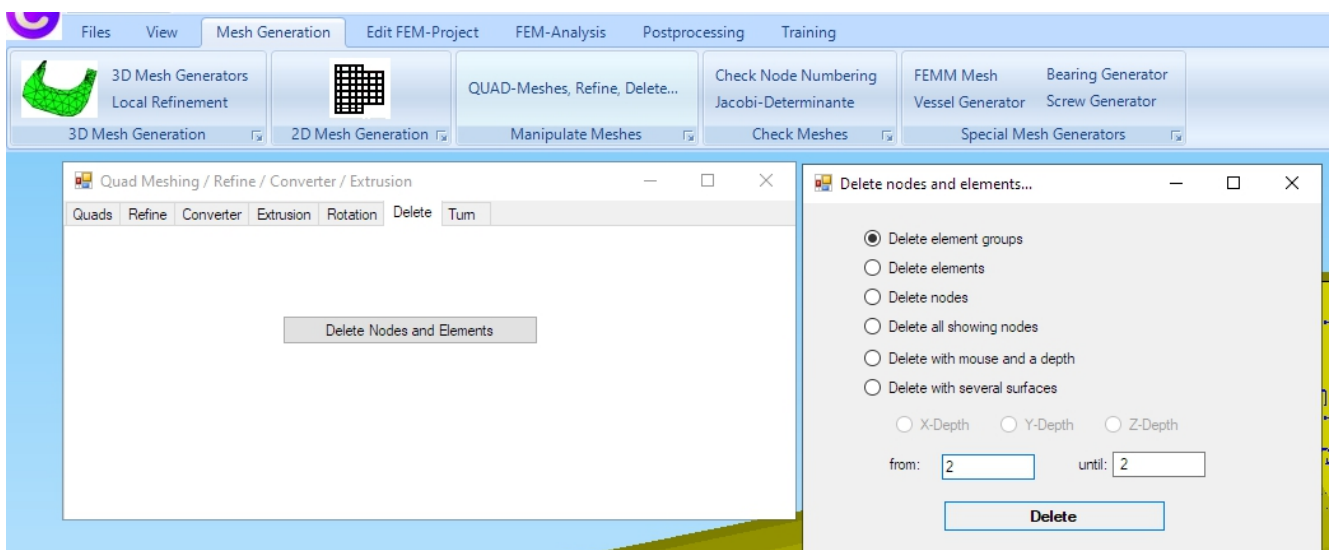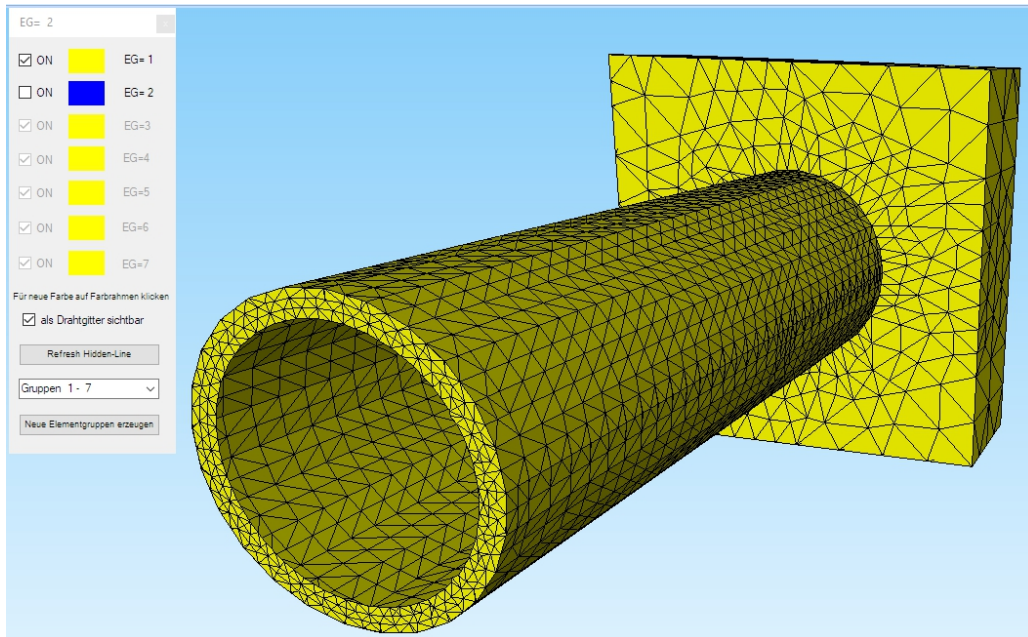
## 1.8 Boundary Conditions

With the "Edit FEM-Project" and "Boundary Condition" tab, the front surface of the quader is clamped fix in the X, Y and Z directions.

## 1.9 Create Surface Load

To create the surface load, first hide EG 2 and refresh the Hidden-Line so that Surface 3 is selected for the Surface Load with 100t or „1 000 000" N. Select "Edit FEM-Project" and "Surface load" to create the load with surface 3.
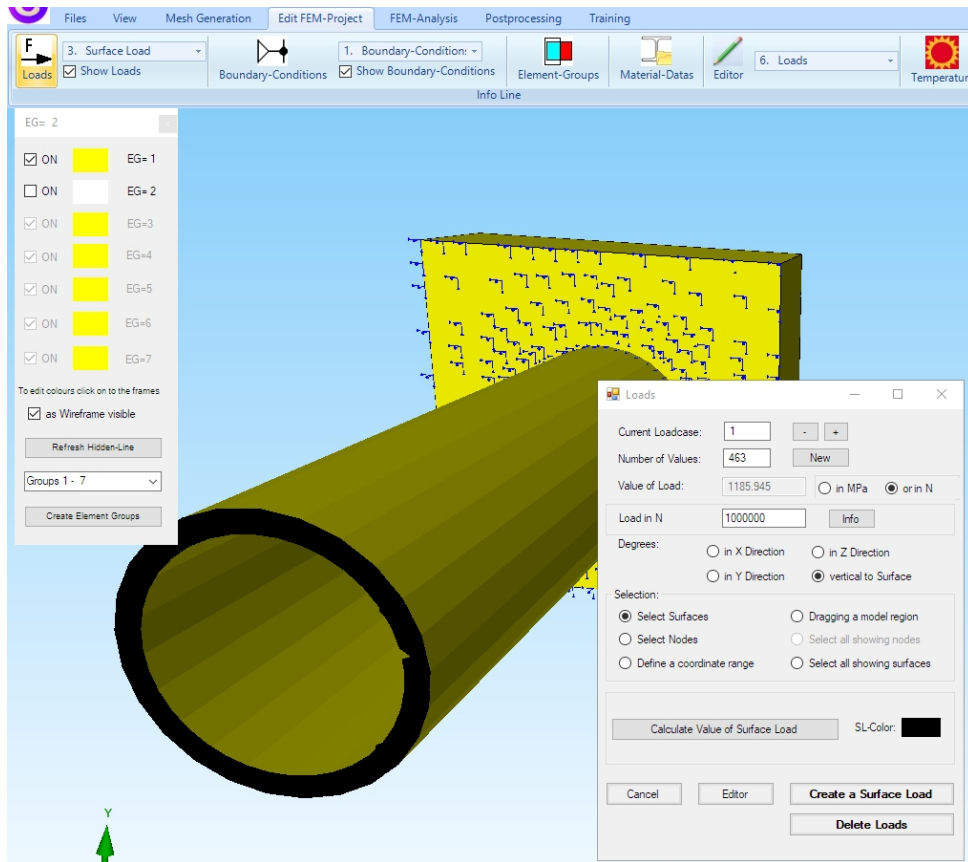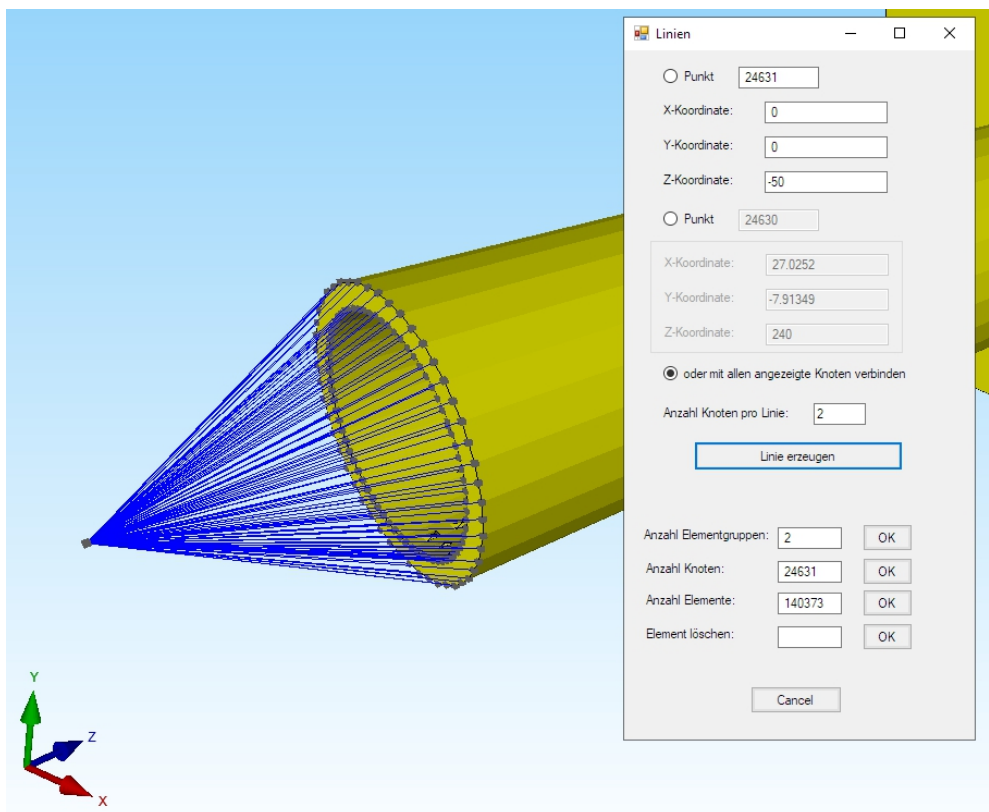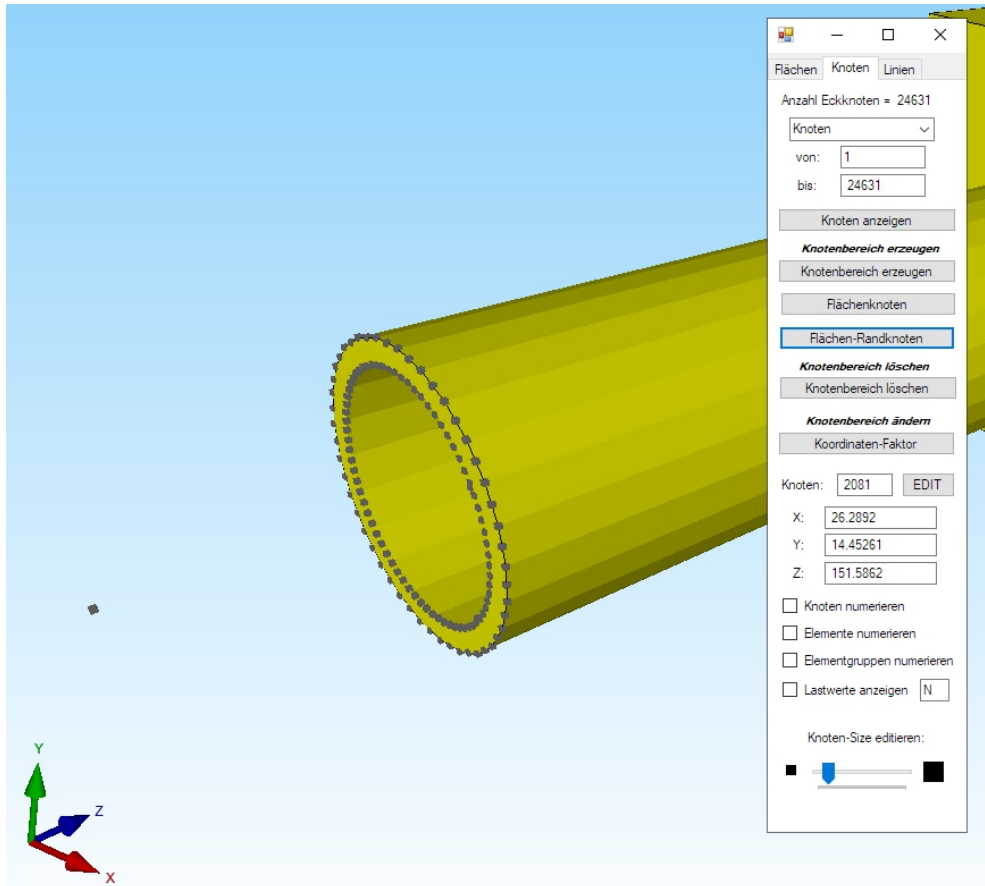


## 1.10 Create the Torsional Moment

The torsional moment is a bit more complex because a beam model for the tetrahedron mesh has to be generated. In the Line-Mode enter the node 24631 (0/0 / -50) with "New".
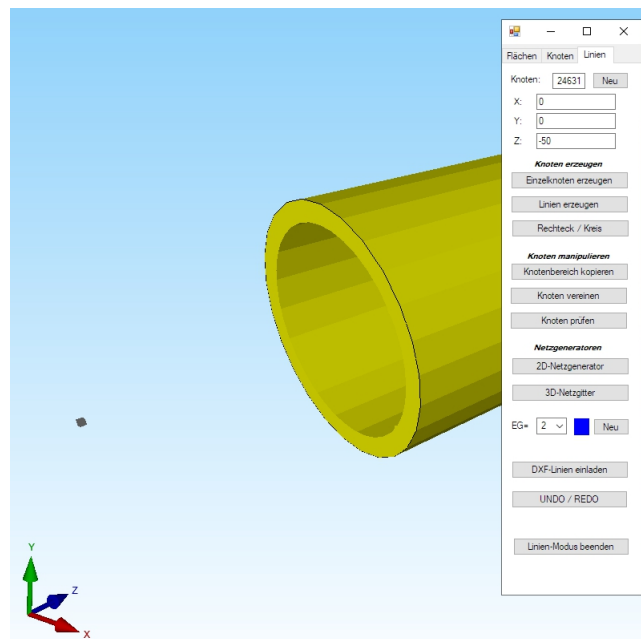
Then use the "Surface Edge Node" menu to create a node range of surface 3.





Now create the nodes 24632 (-100 / 0 / -50) and 26633 (100/0 / -50) in the Line-Mode and connect the two lines 24631/24632 and 24 631/24633 with "Create Lines".
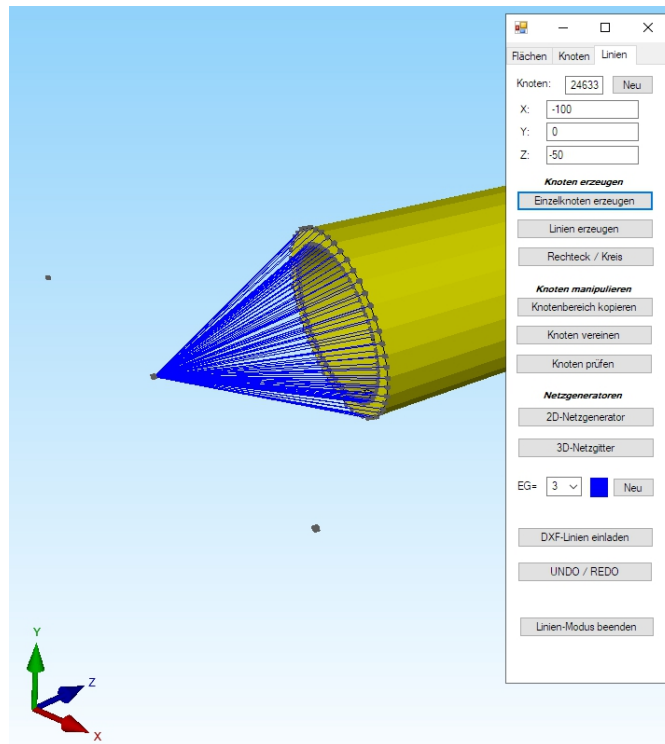
Select the "Edit FEM-Project" tab and "Editor" to create load case 1 with a nodal load. Edit a "2" in the editor with "Number of loads/Load Case" and enter nodes 24632 and 24633 in the Y direction with "FHG = 2" and a load value of "10000" and "-10000".
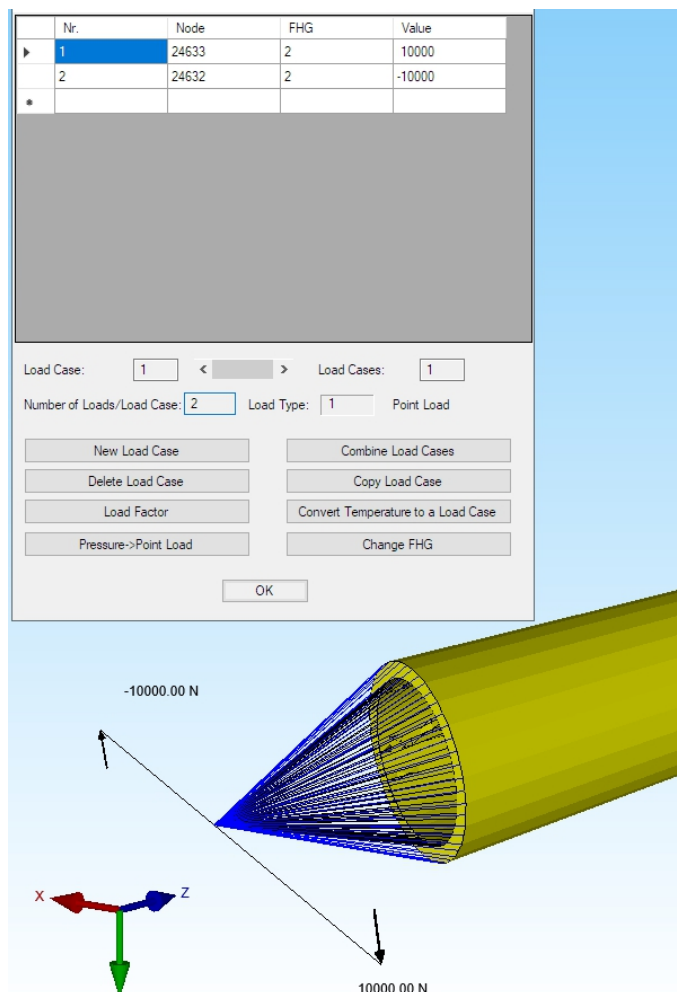
## 1.11 Results Evaluation

After the FEM analysis, the "Result evaluation" tab and the icon menu [icon] can be used to evaluate the axial and torsional stresses on the cylinder.

**Load case 1: Axial Stress on the cylinder = 1159 MPa (exact = 1157 MPa)**
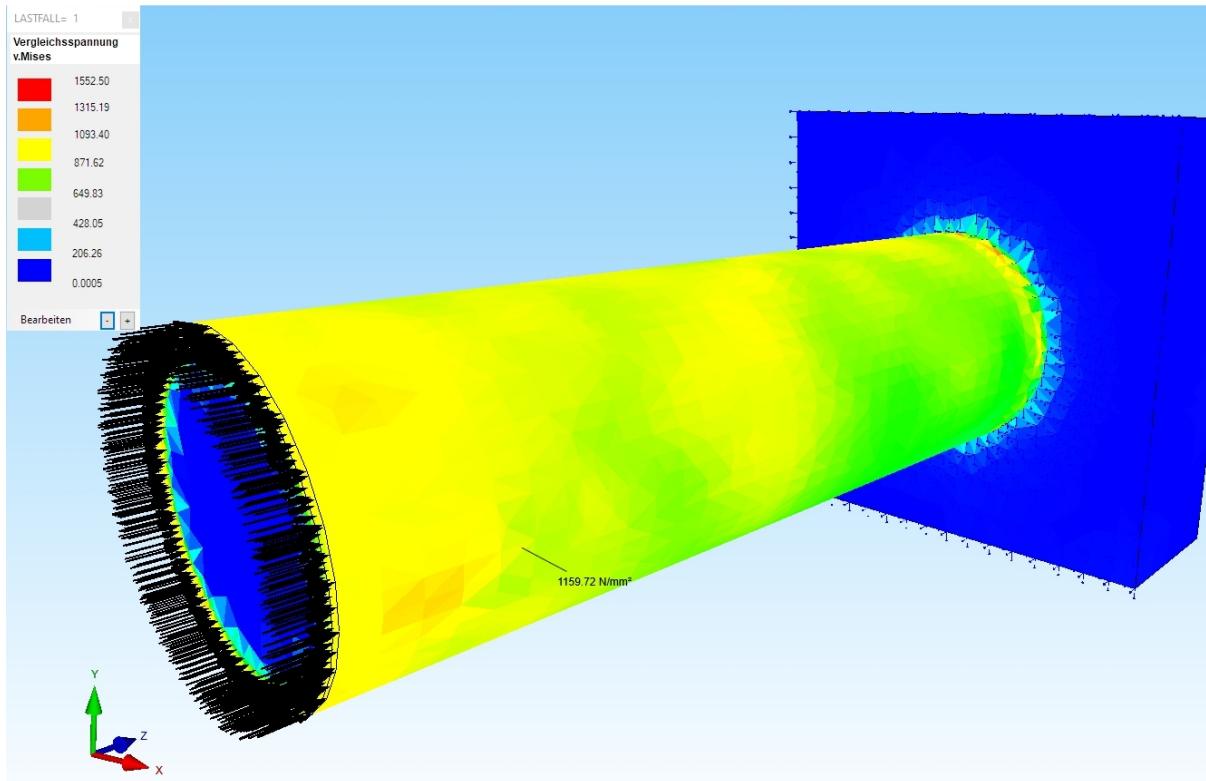


**Load Case 2: Torsional Stress of the cylinder = 127 MPa (exact 111 MPa)**