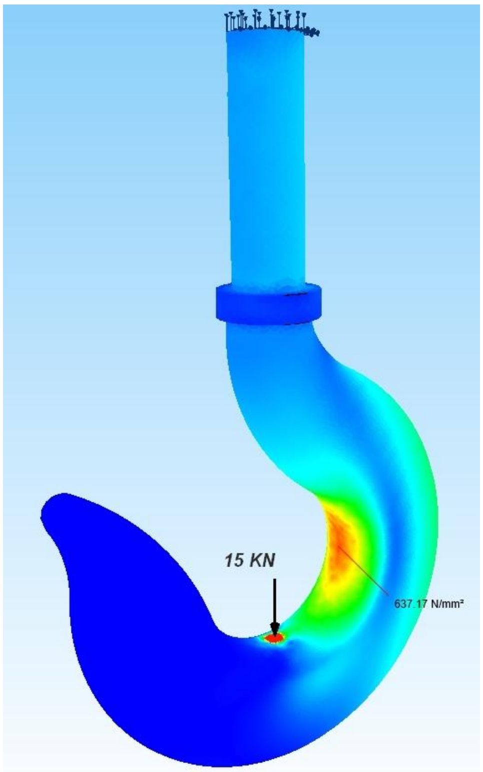
FEM-System

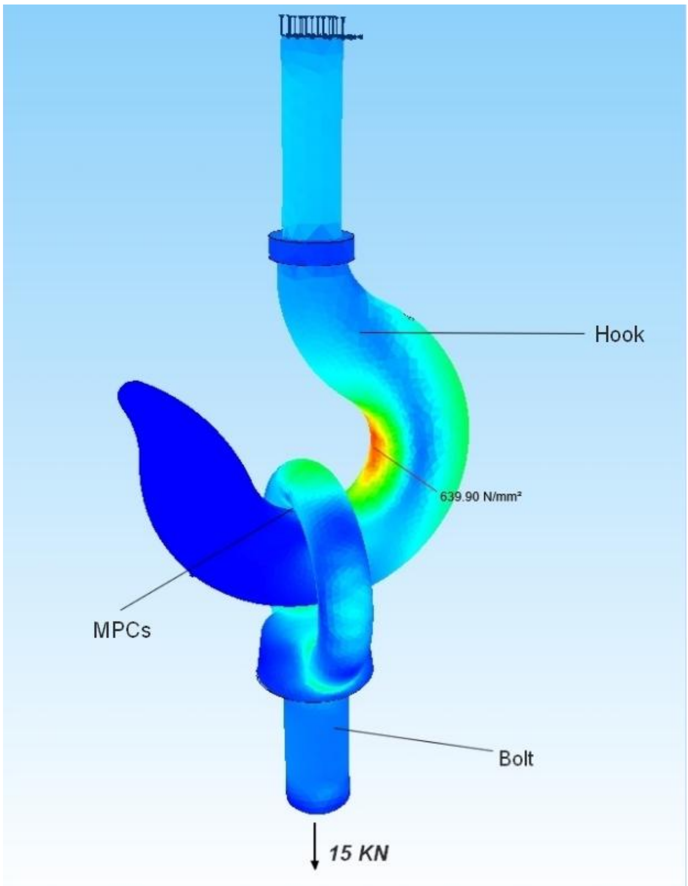
MEANS V11

FEM Calculation of a Crane Hook as a single Part and as an Assembly

Crane Hook with MEANS-LITE



Crane Hook Assembly with MEANS HIGH-END



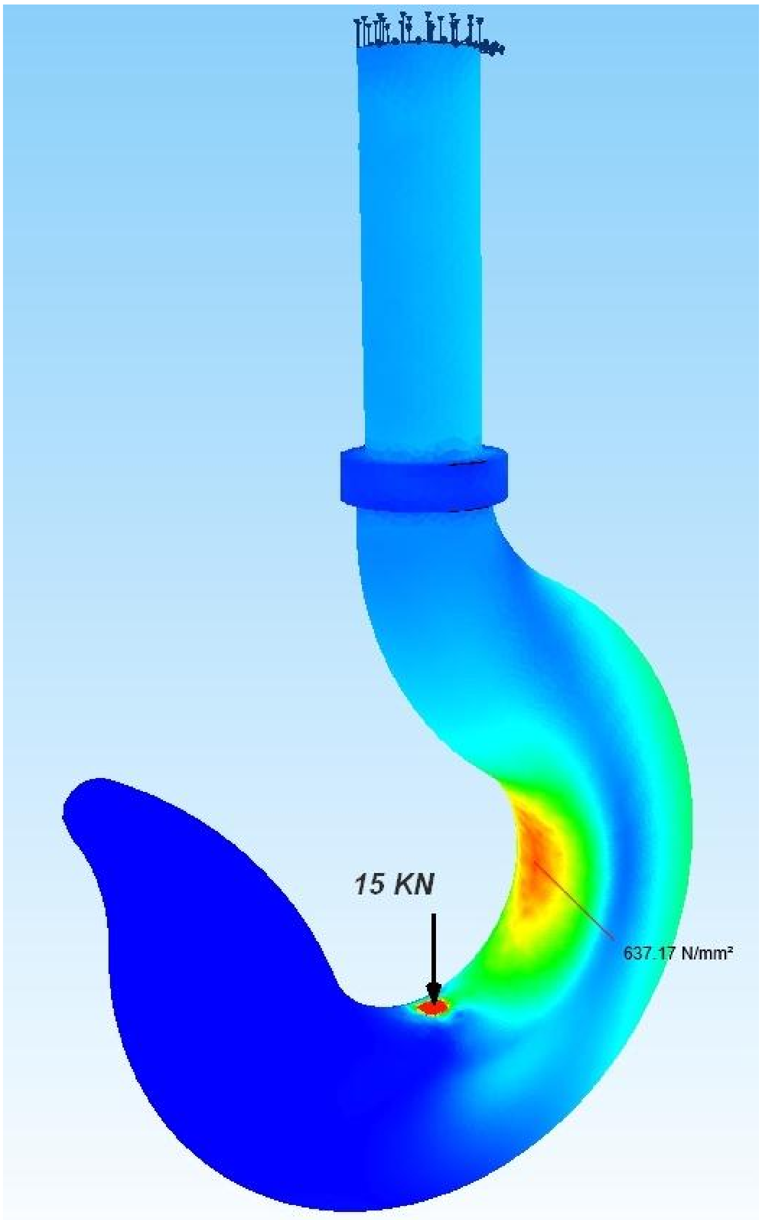
Ing.Büro HTA-Software
Germany-Rheinau

[*www.femcad.de*](http://www.femcad.de)

[*www.fem-infos.com*](http://www.fem-infos.com)

Part 18: Crane Hook with MEANS-LITE and Crane Hook Assembly with MEANS HIGH-END

Crane Hook Part: The Crane Hook is a single part with a point load of 15 KN. The FEM Model consists 92 343 Tetrahedral elements and the FEM calculation can therefore be done with **MEANS-LITE up to 100,000 elements**.

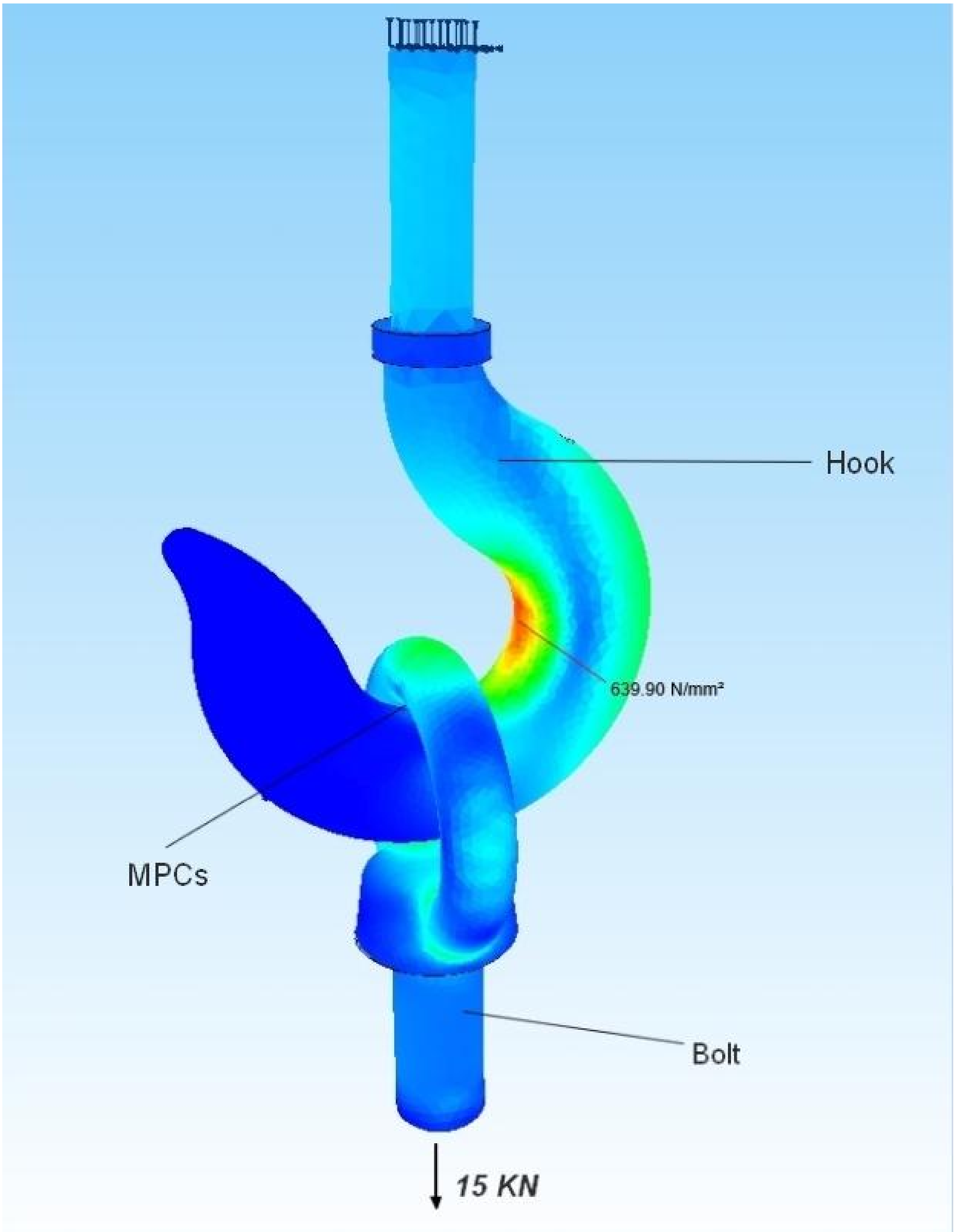


Material Datas:

The Crane Hook is loaded with a hook load of 15 KN and consists of the tempered steel 42CrMo4 with the following material datas:

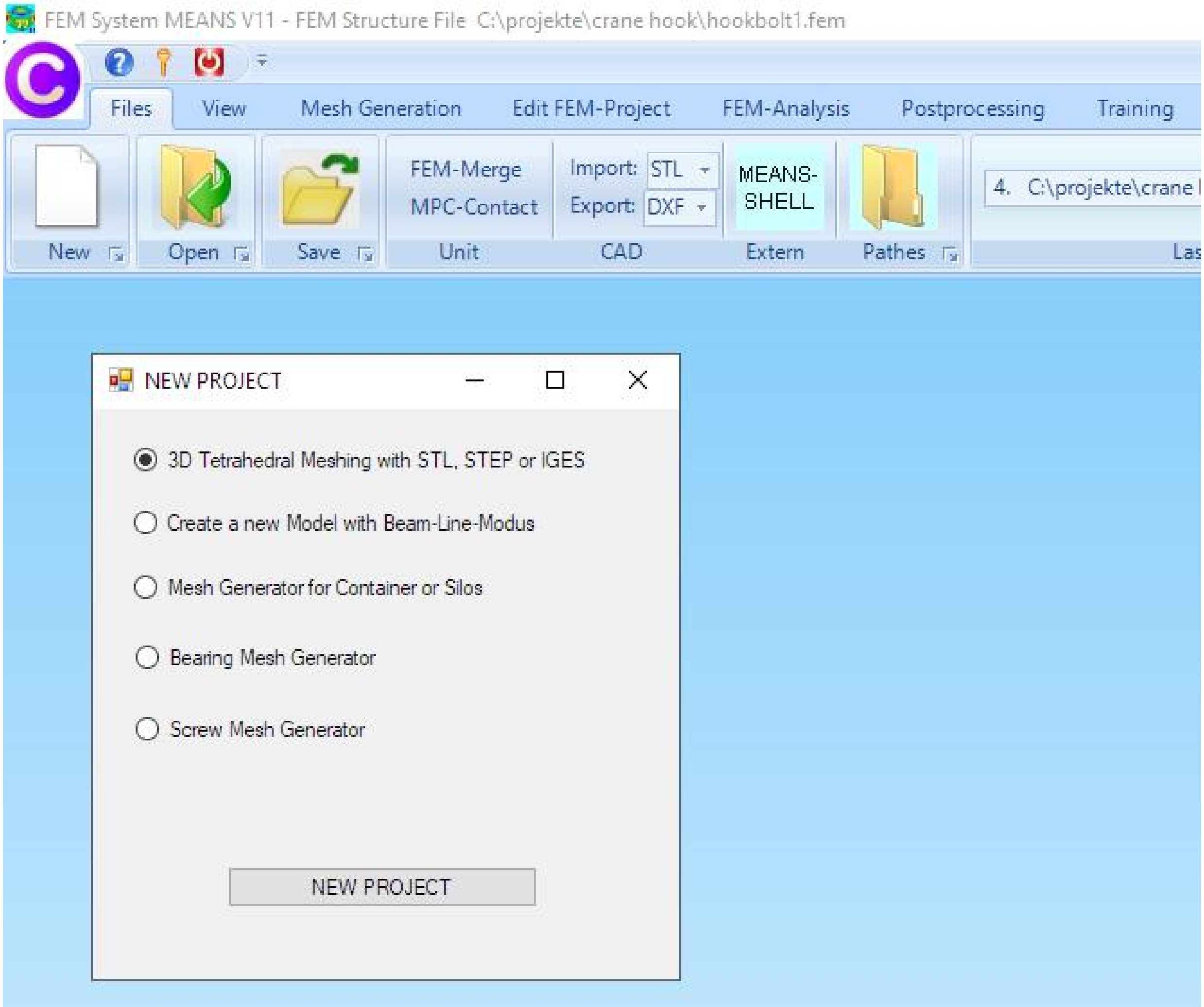
Tensile Strength Rm = 1100 MPa Yield Strength Re = 900 MPa
Young's modulus E = 210,000 MPa

Crane Hook Assembly: The Assembly consists of Hook and Bolt and is loaded at the bolt end with 15 KN. The Assembly is calculated using a Contact Analysis and consists 350,000 TET4 elements and 350 MPC elements and can only be calculated with **MEANS HIGH-END and Add On CONTACT.**



Crane hook with MEANS-LITE

First an FEM mesh with over 99,000 tetrahedral elements is generated from the crane hook CAD model and is calculated with the light version MEANS-LITE which is limited until 100,000 elements and nodes.



Select the "Files" tab and select "New" to create a new FEM project.

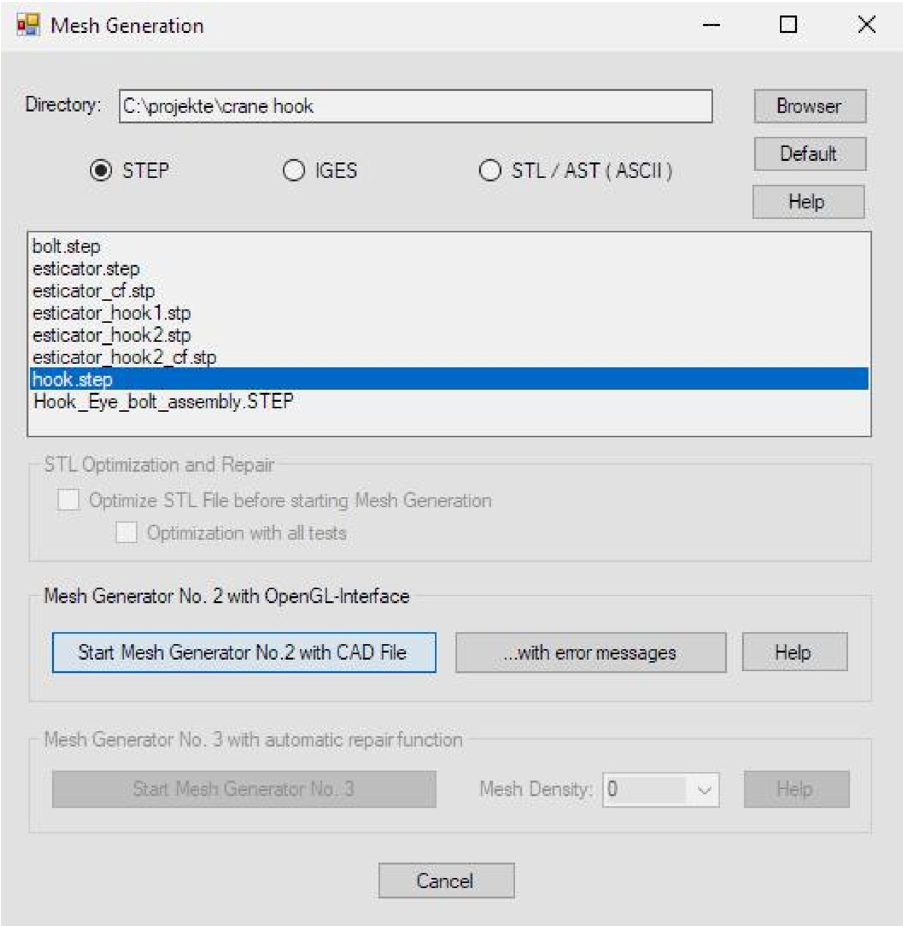
Select "3D Tetrahedral Meshing with STL, STEP or IGES for the following formats:

STL this 3D model consists of a triangular outer shell for the 3D Mesh generation, this flexibel format can also be imported and exported. STL Files with holes or gaps, it can also be repaired before meshing.

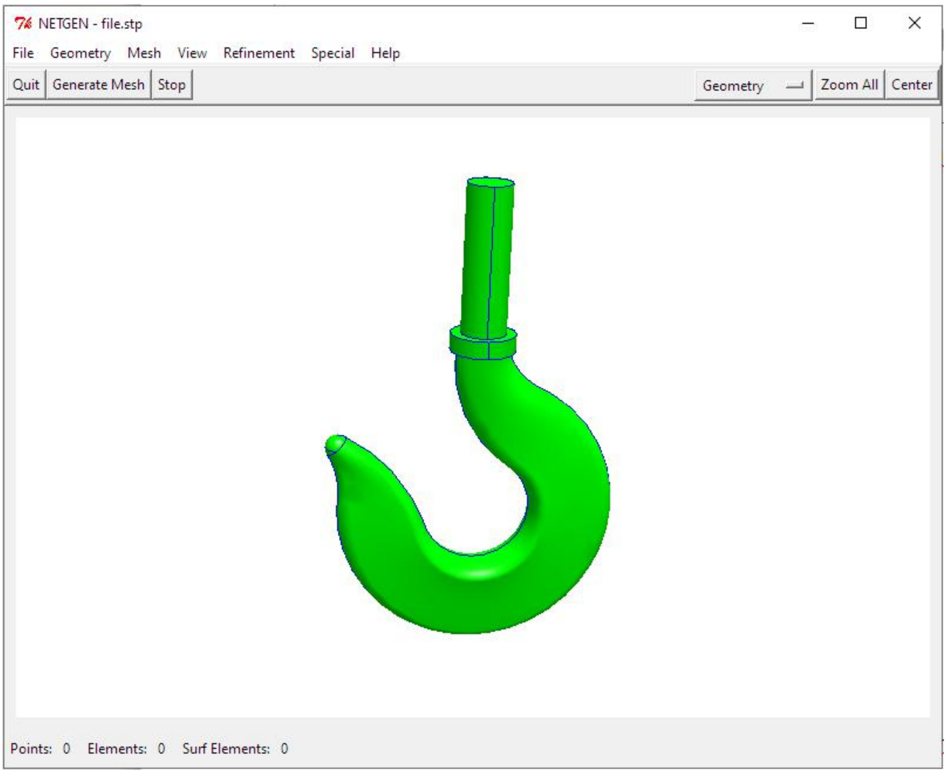
STEP today it is the Standard format, note that no CAD assemblies but only single parts can be meshed. Assemblies can be combined via the “Boolean operations” or with “Screw Models” into one part.

IGES like STEP format but is not as common anymore

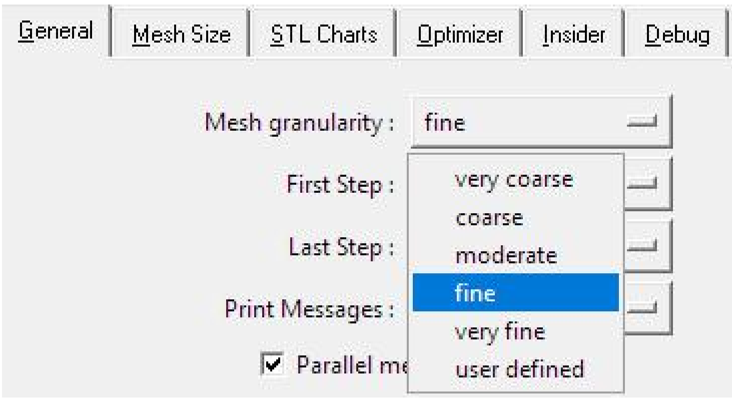
Use the "Browser" button to select the "hook.step" STEP file and click on menu "Start Mesh Generator No. 2 with CAD File" to display it in the mesh generator.



The model can now be seen in the mesh generator and can be rotated as required.

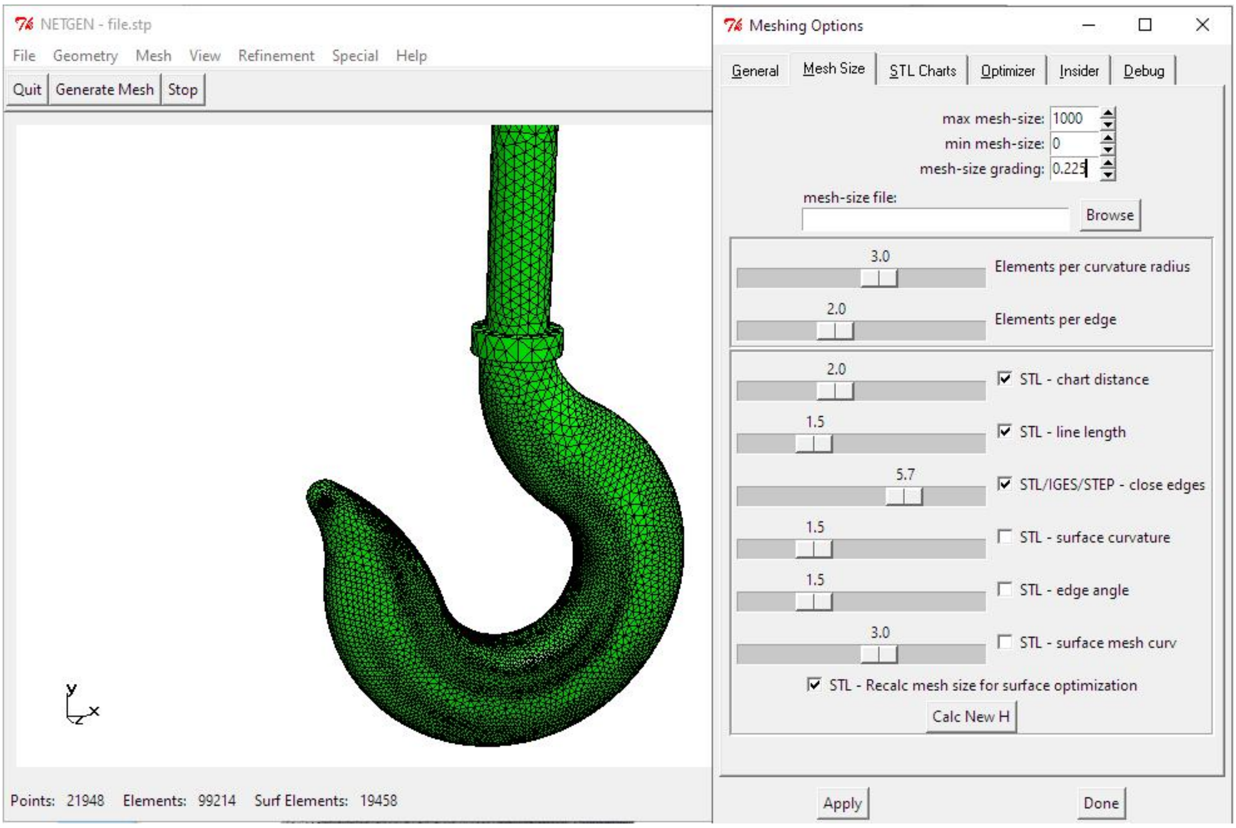


First select the "Mesh" menu and the "General" tab and select the mesh density "fine".



Mesh size grading

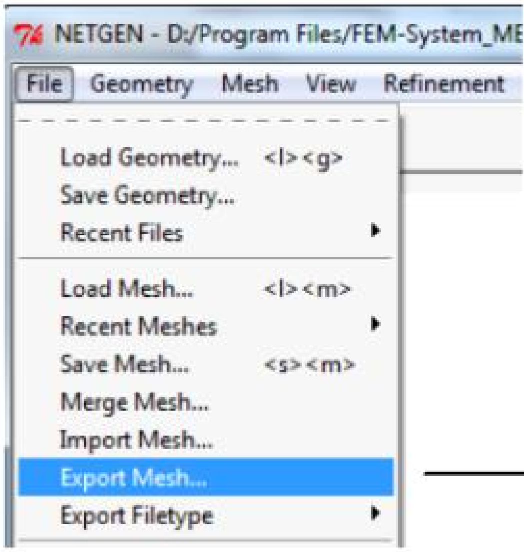
Then select the “Mesh Size” tab and generate with the mesh-size-grading = 0.225 and the main menu "Generate Mesh" an FEM mesh with 99214 tetrahedral elements so that the number of elements is just below the MEANS-Lite limit of 100,000 elements.



Export Mesh

After generating the mesh, the FEM mesh with the name "test.fem" must be exported to MEANS V11. Select the menu "File" and "Export Mesh" and save the mesh "test.fem" in the specified debug mesh path.

.



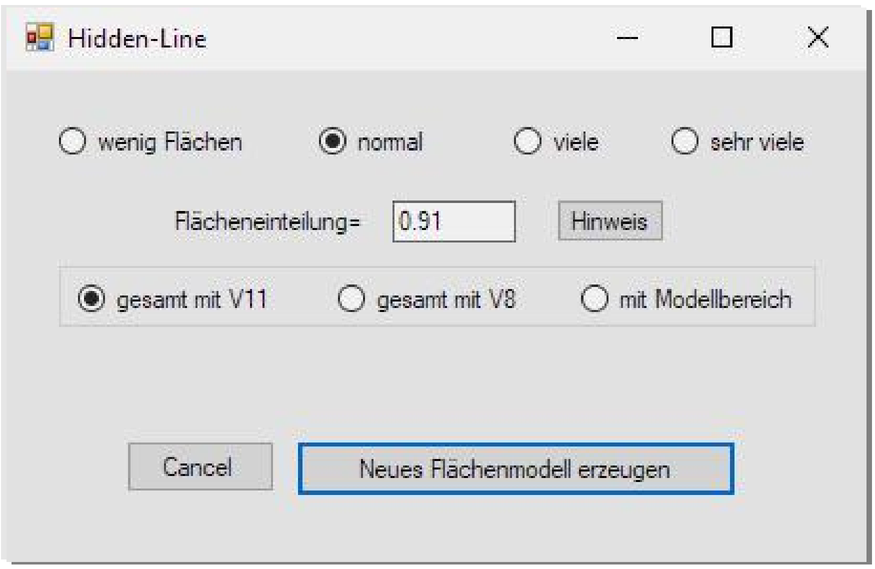
Select the menu "File" and "Export Mesh" and save the mesh "test.fem" in the specified debug mesh path.

Generate Surface Model

After exporting "test.fem", MEANS V11 is started automatically and first creates the Surface Model so that surfaces, edges and nodes can be selected for loads, boundary conditions or element groups.

.

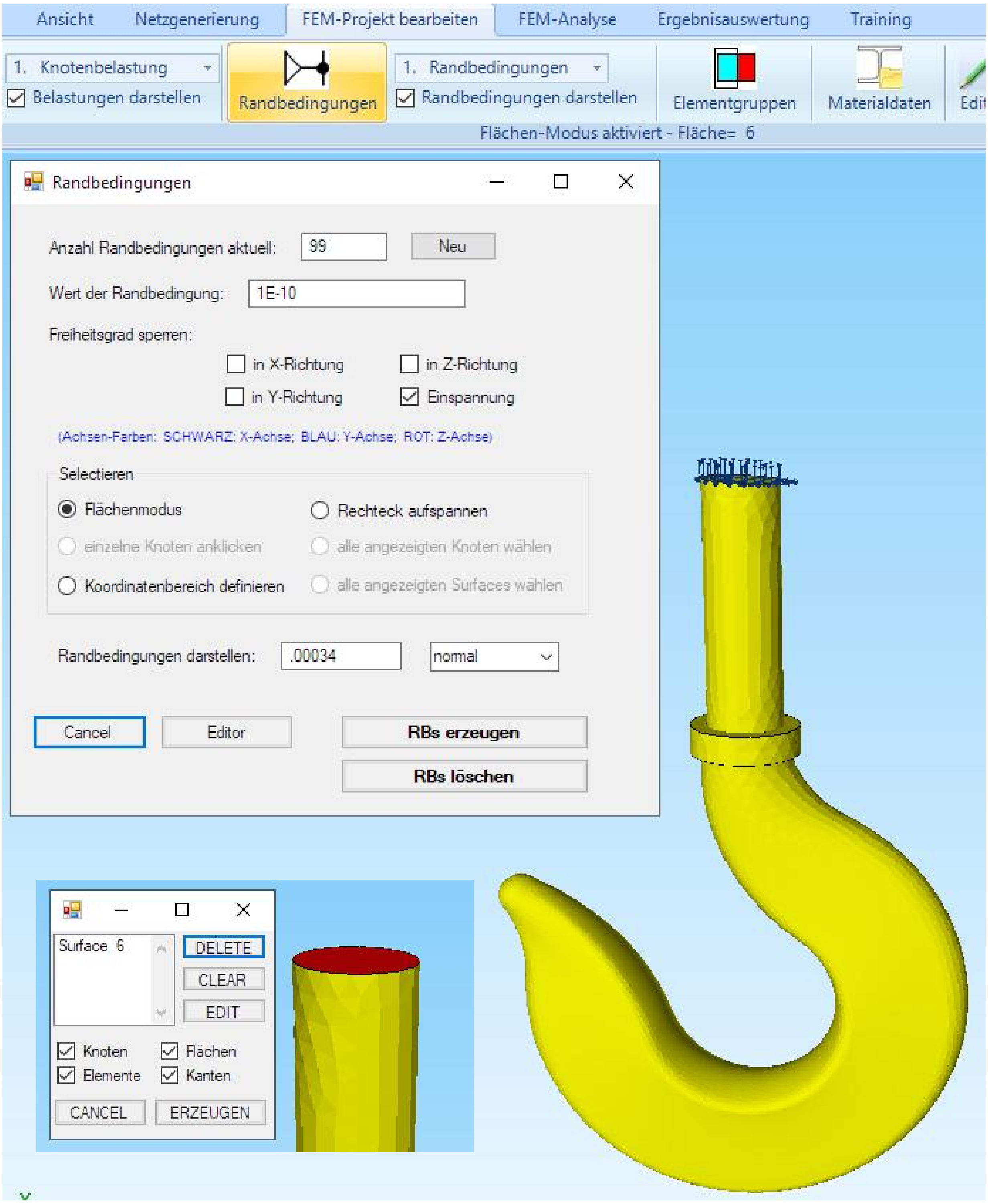
.



Crane hook clampled fixed

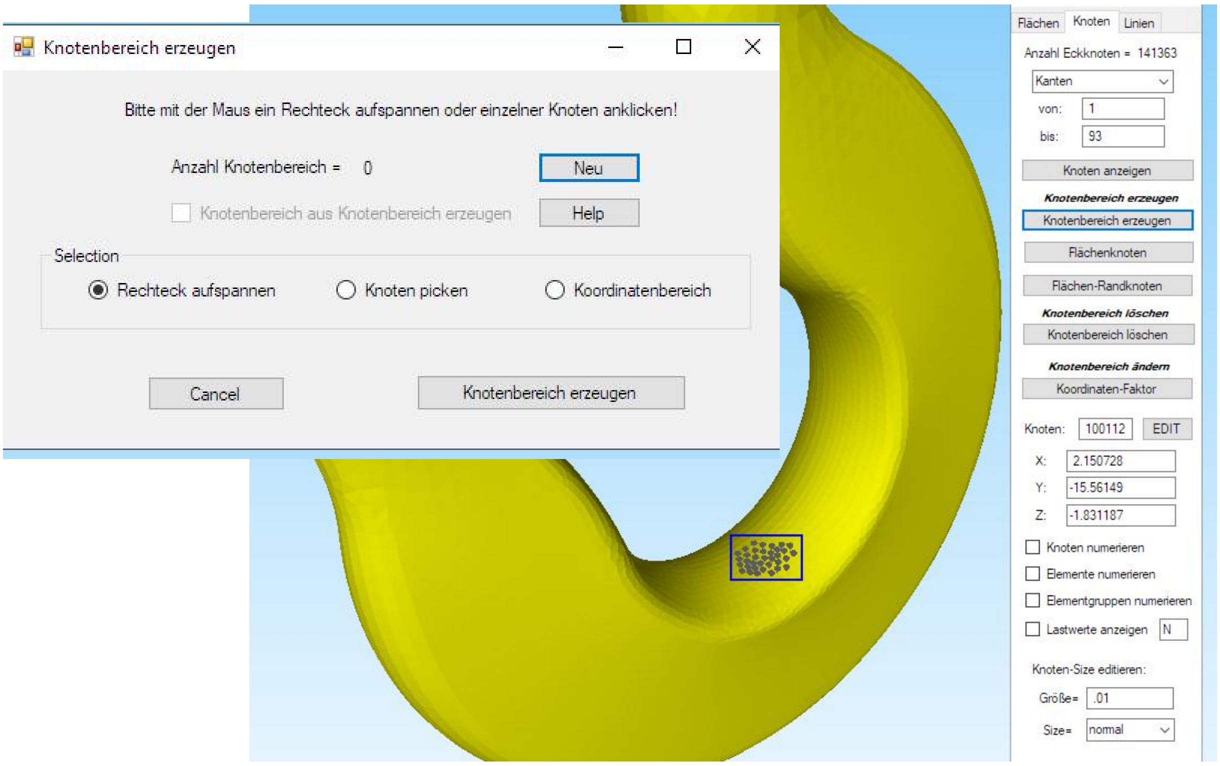
The crane hook is firmly clamped at the top. To do this, select the "Edit FEM project" tab and "Boundary conditions".

Choose “Select Surface” and "Create RBs" button and click on surface 6 to display it in the select box. The clamping is created there with “Generate”.

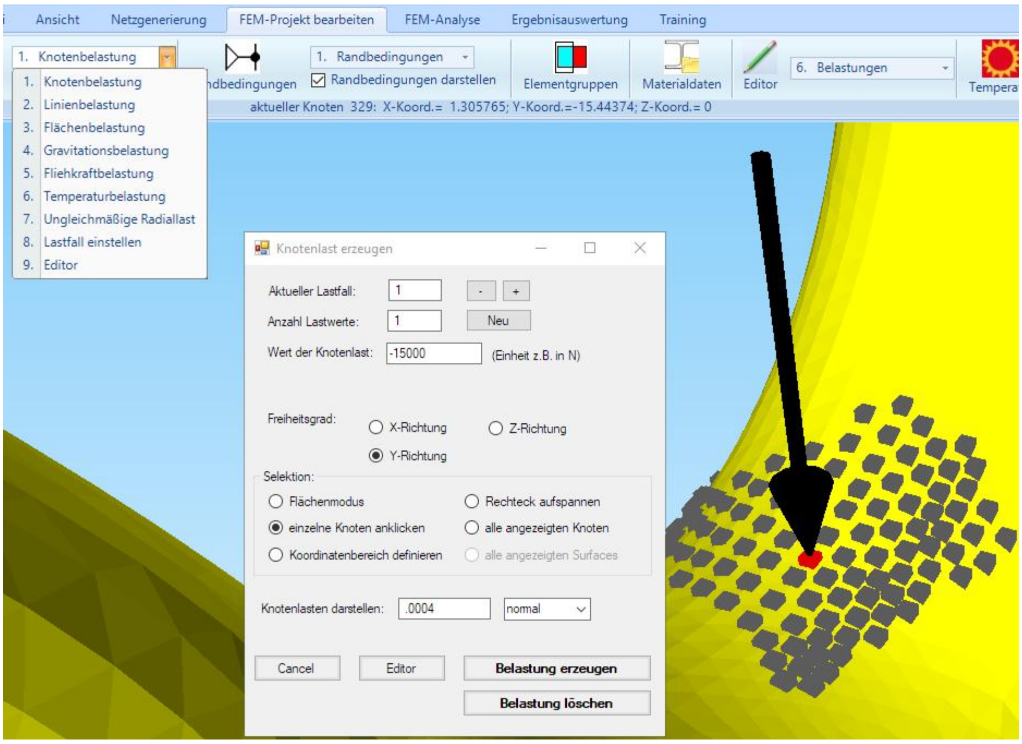


Hook loaded with a Point Load

The hook is loaded with 15 KN in the Y direction. First, a selectable node area must be created in the Node-Modus by spanning a rectangle over the load. Now the model is rotated and zoomed so that the node is not covered and can be clicked on.

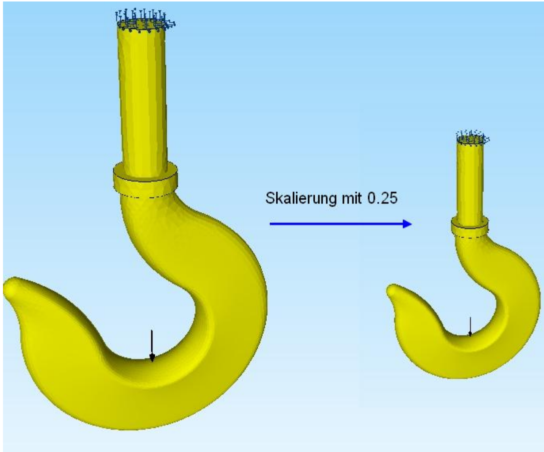


Now you can click on node "Edit FEM project" and "Point Load" of node 329 with a load of -15000 N in the Y direction

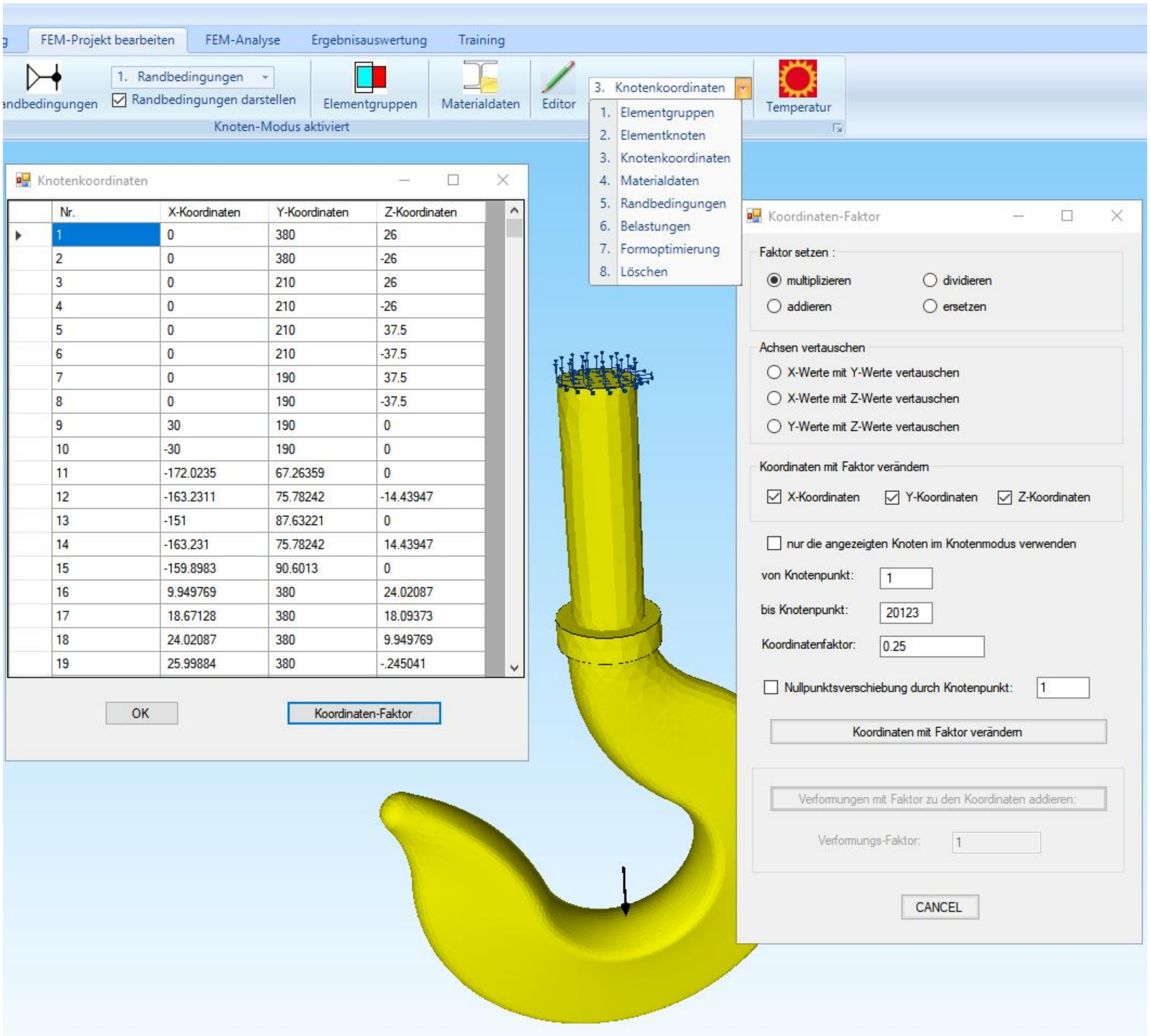


Model scaled

So that the Hook is comparable to another FEM model, the node coordinates should be multiplied by a factor of 0.25.

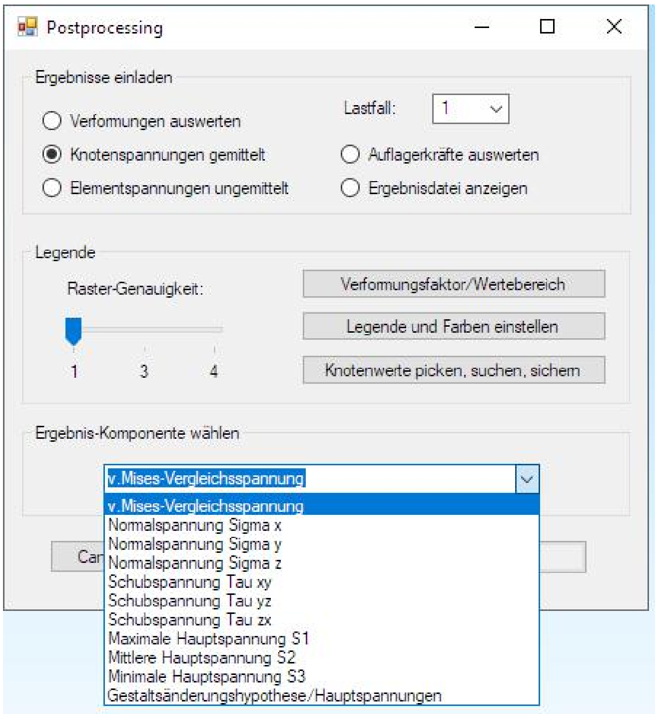


Select the tab “Edit FEM Project” and “Node coordinates” and multiply the coordinates by a coordinate factor of “0.25”. The node check can be carried out.

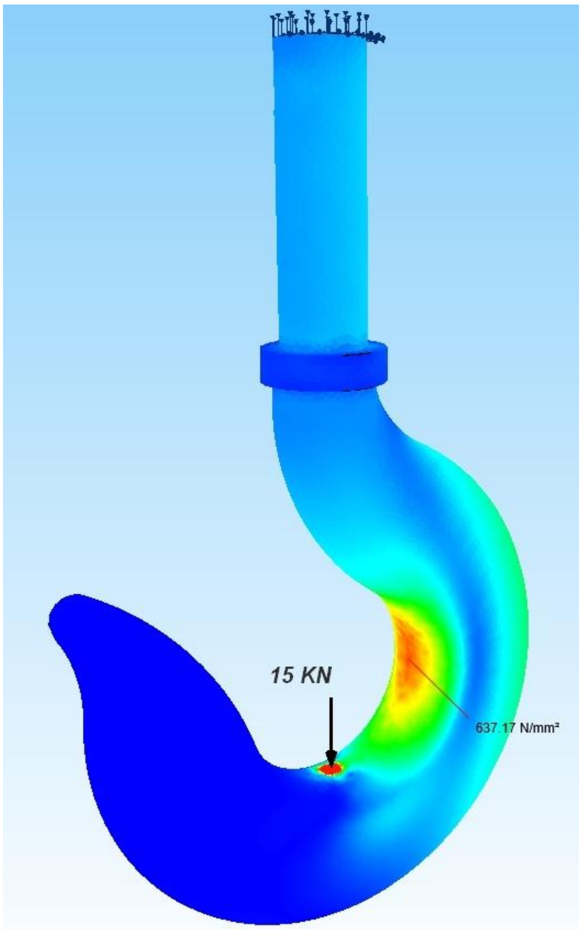


Postprocessing

After the FEM analysis the node stresses can be evaluated with the “Postprocessing” tab.

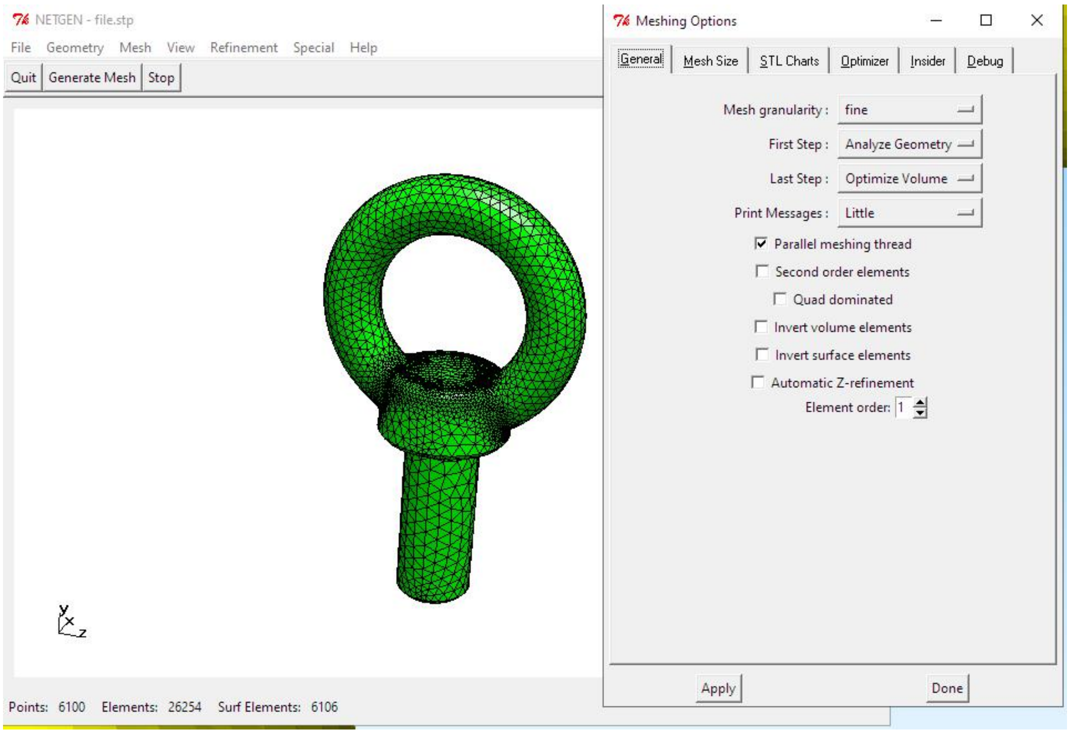


The max. v.Mises-Stress is 637.17 MPa.



Connect the Crane Hook Assembly

The crane hook assembly consists of the previous Hook and Bolt, the latter is meshing with “fine” with 26 254 tetrahedradral elements and saved as Bolt.FEM.



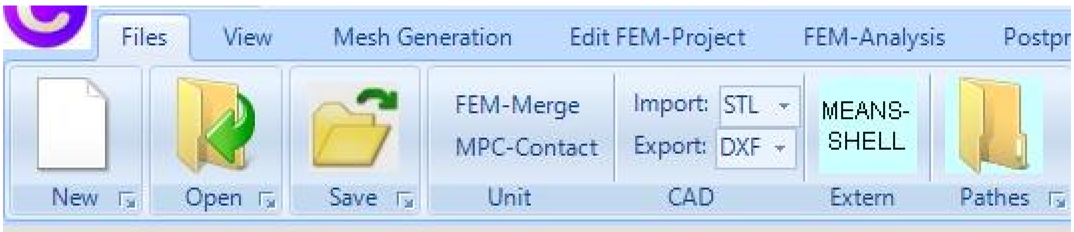
Merge Hook and Bolt Mesh to a single Part

The following two FEM meshes are now combined with an FEM-Merge:

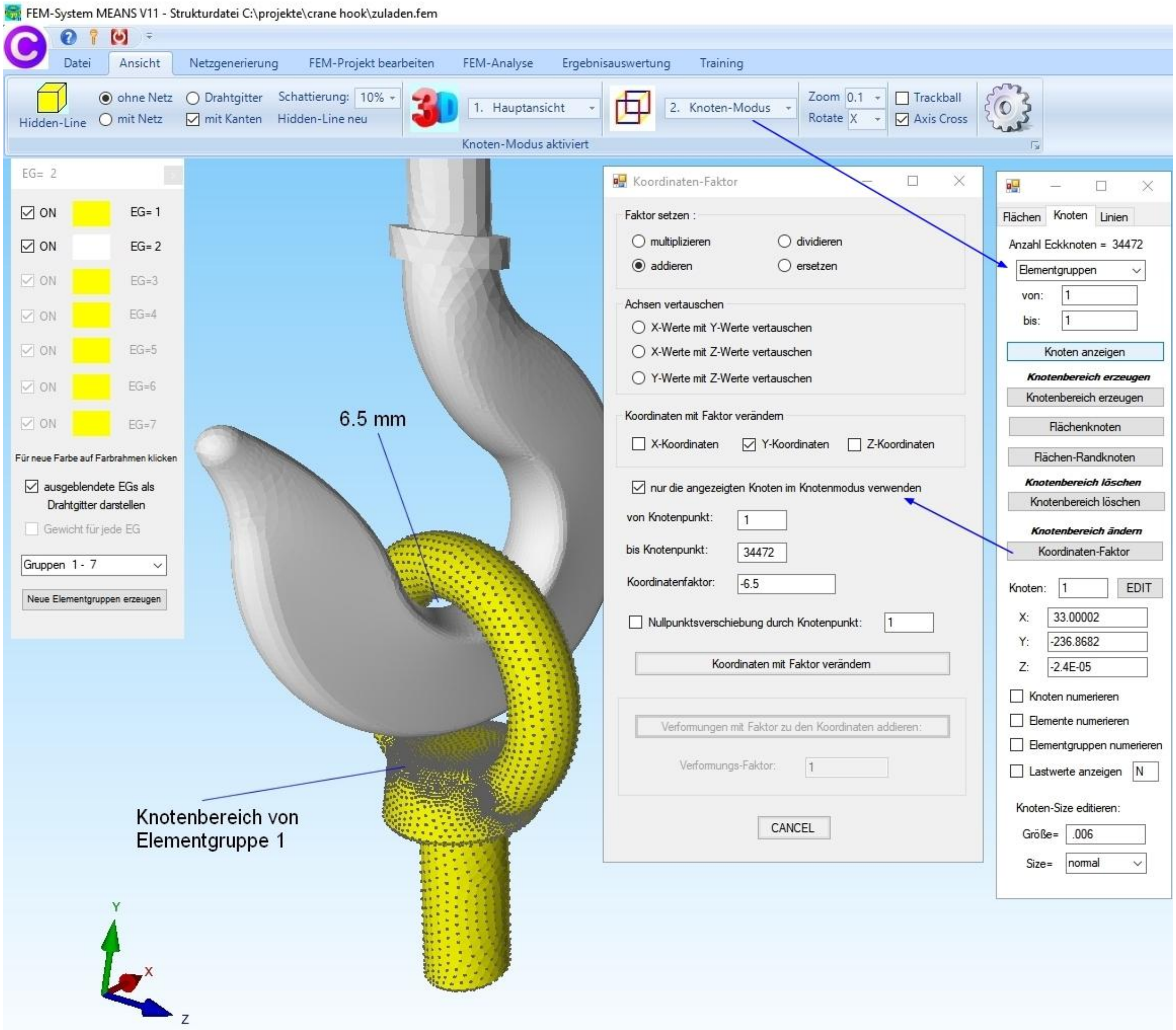
Mesh 1: Hook.FEM with 21 948 nodes and 99 214 tetrahedral elements

Mesh 2: Bolt.FEM with 6100 nodes and 26 254 tetrahedral elements

First load the larger FEM model Hook.FEM into MEANS V11, then add the second mesh using the "File" and "FEM-Merge" tabs.



The hook and bolt are now combined to a single FEM model, but unfortunately the two parts do not touch, so the bolt must be moved by -6.5 mm without scaling or -1.62 m with scaling in the Y direction.
Use the “View” and “Node Modus” tabs to set the node mode menu and show all nodes of element group 1. Select “Coordinate factor” and add the Y coordinates of the displayed ones Nodes with a coordinate factor of “-6.5” mm.

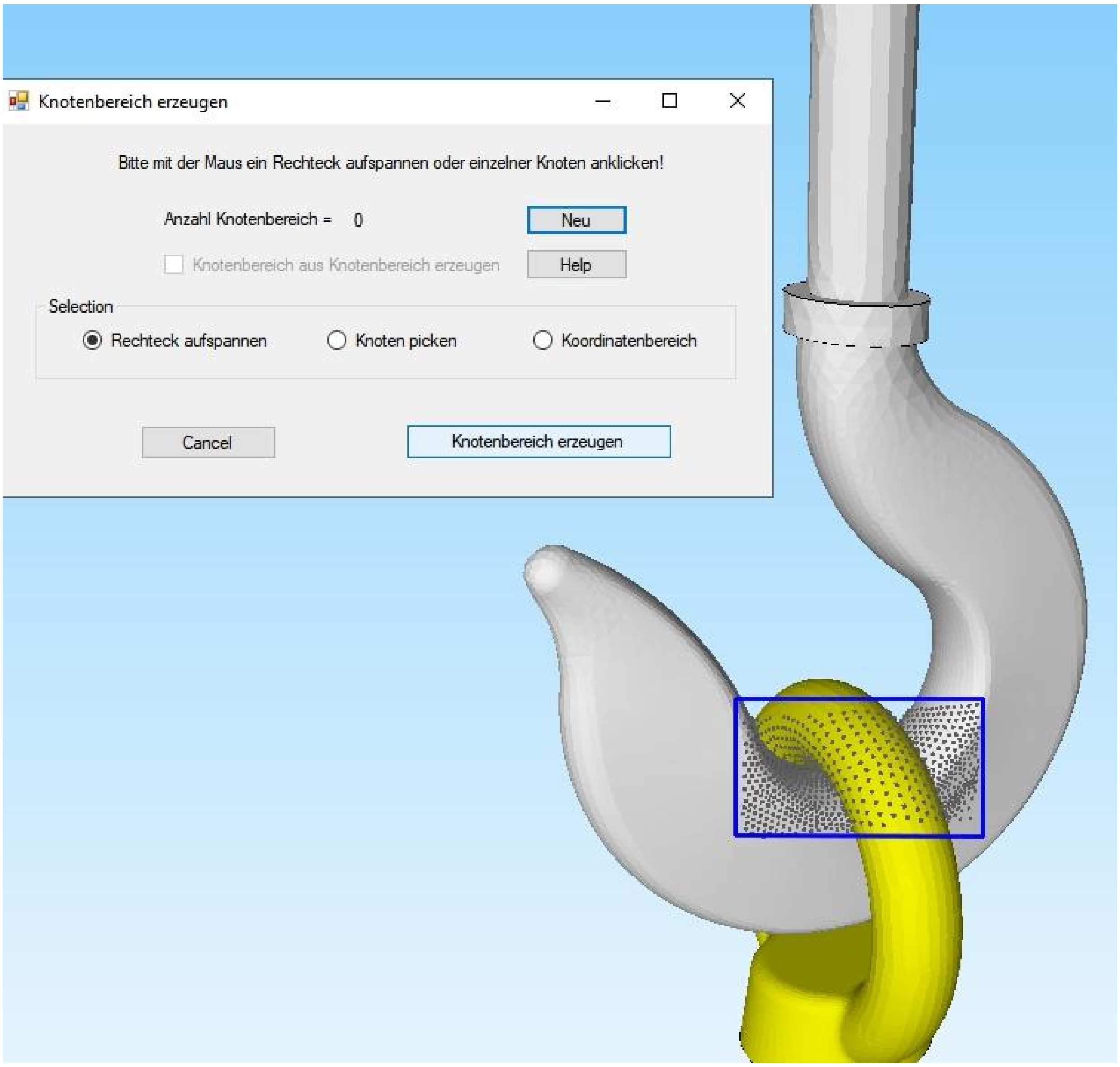


MPC-Contact

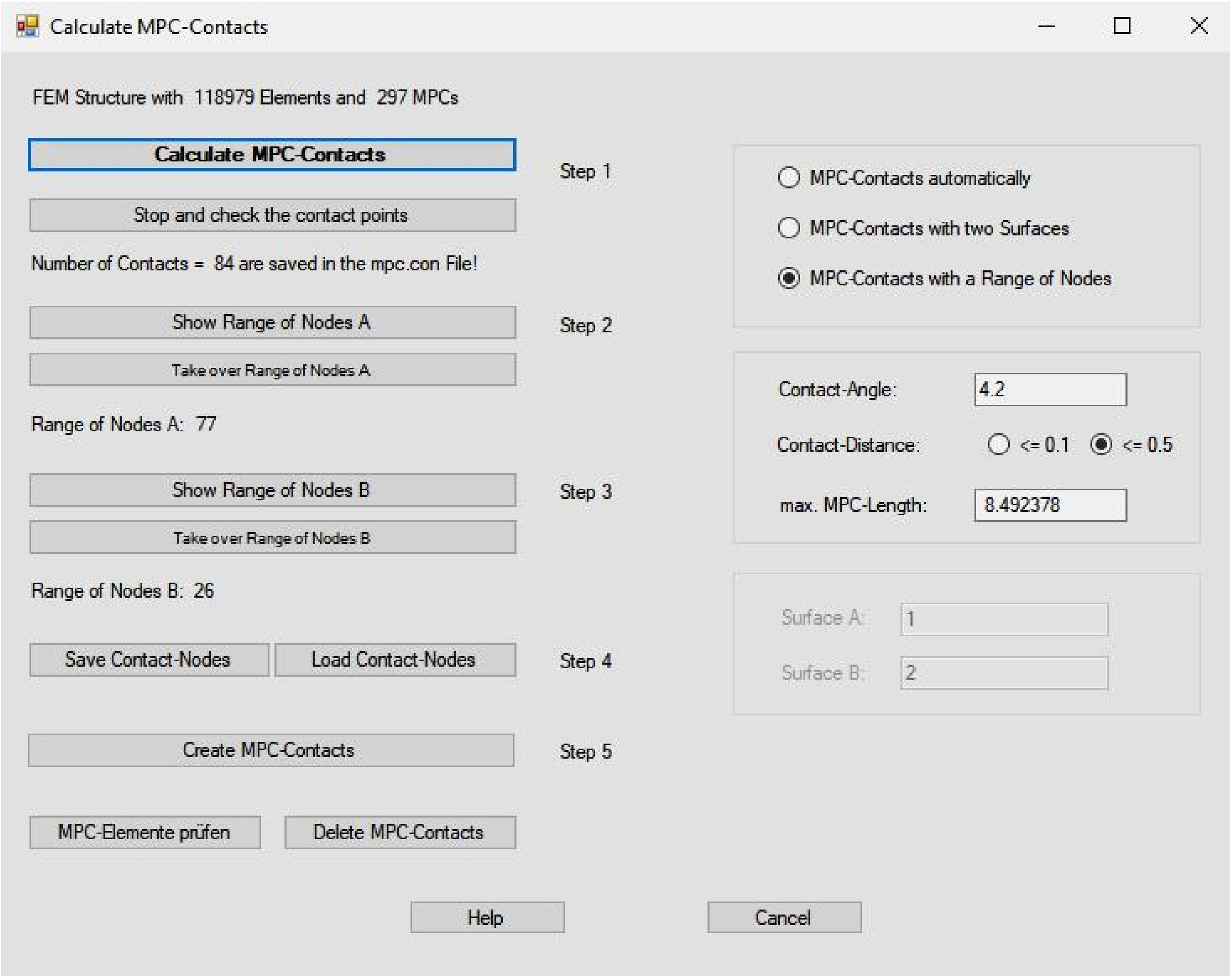
The two parts are now combined in a single FEM file, but they must also be connected at the point of contact via MPC elements (MPC elements are discussed in detail in Chapter 10 - MPC Analysis with MEANS V11).

Use the “View” and “Node Modus” and “Create a Range of Nodes” tabs to create a rectangle over the two contact areas.

Double-click on the blue marking rectangle to display all nodes in this area.

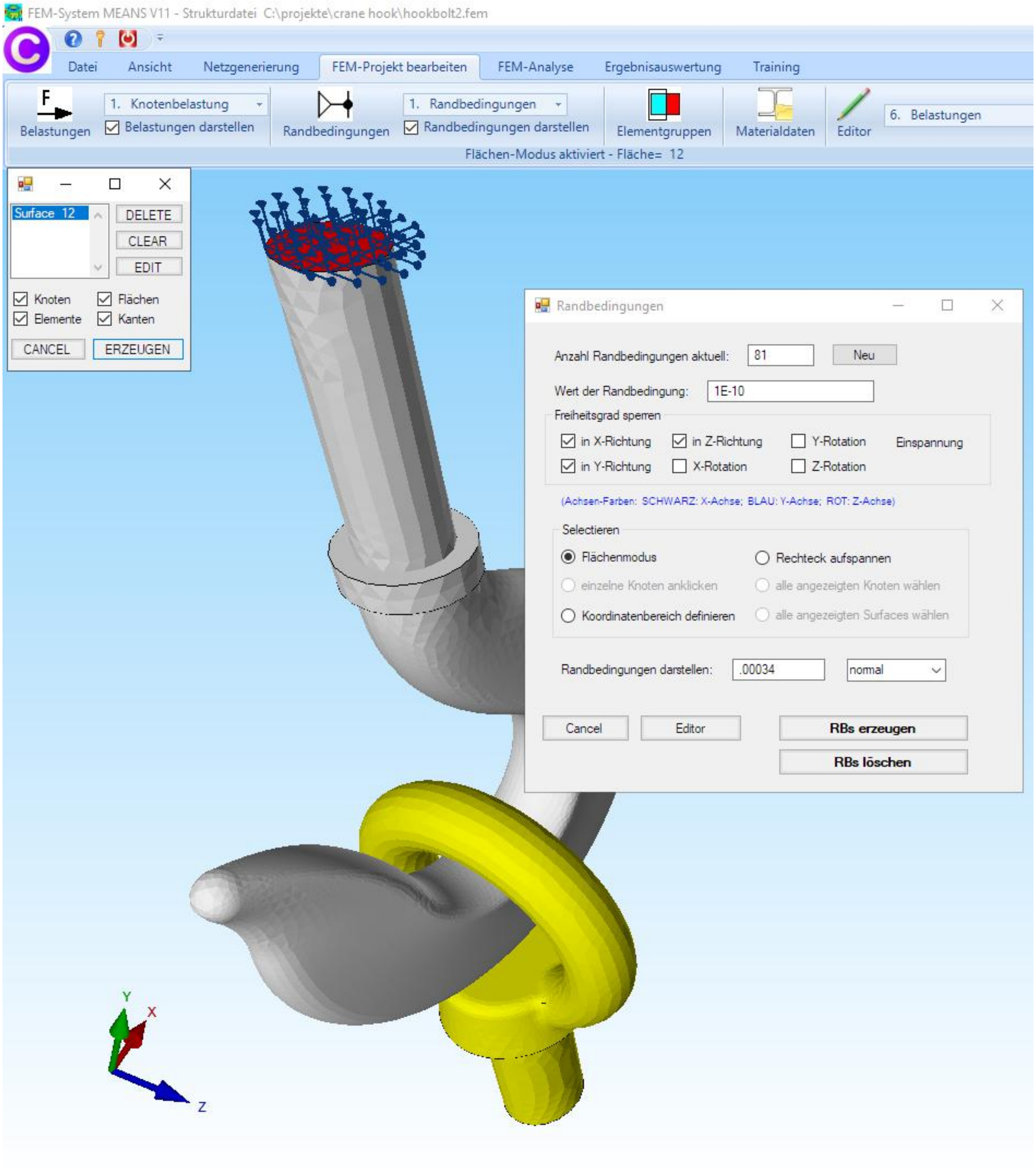


Then select the “File” and “MPC Contacts” tab and start the contact calculation to automatically generate 297 MPC elements with the node area above.



Hook clamping

The hook is firmly clamped at the top, select the "Edit FEM project" tab and "Boundary conditions" to clamp surface 12 in the X, Y and Z directions.

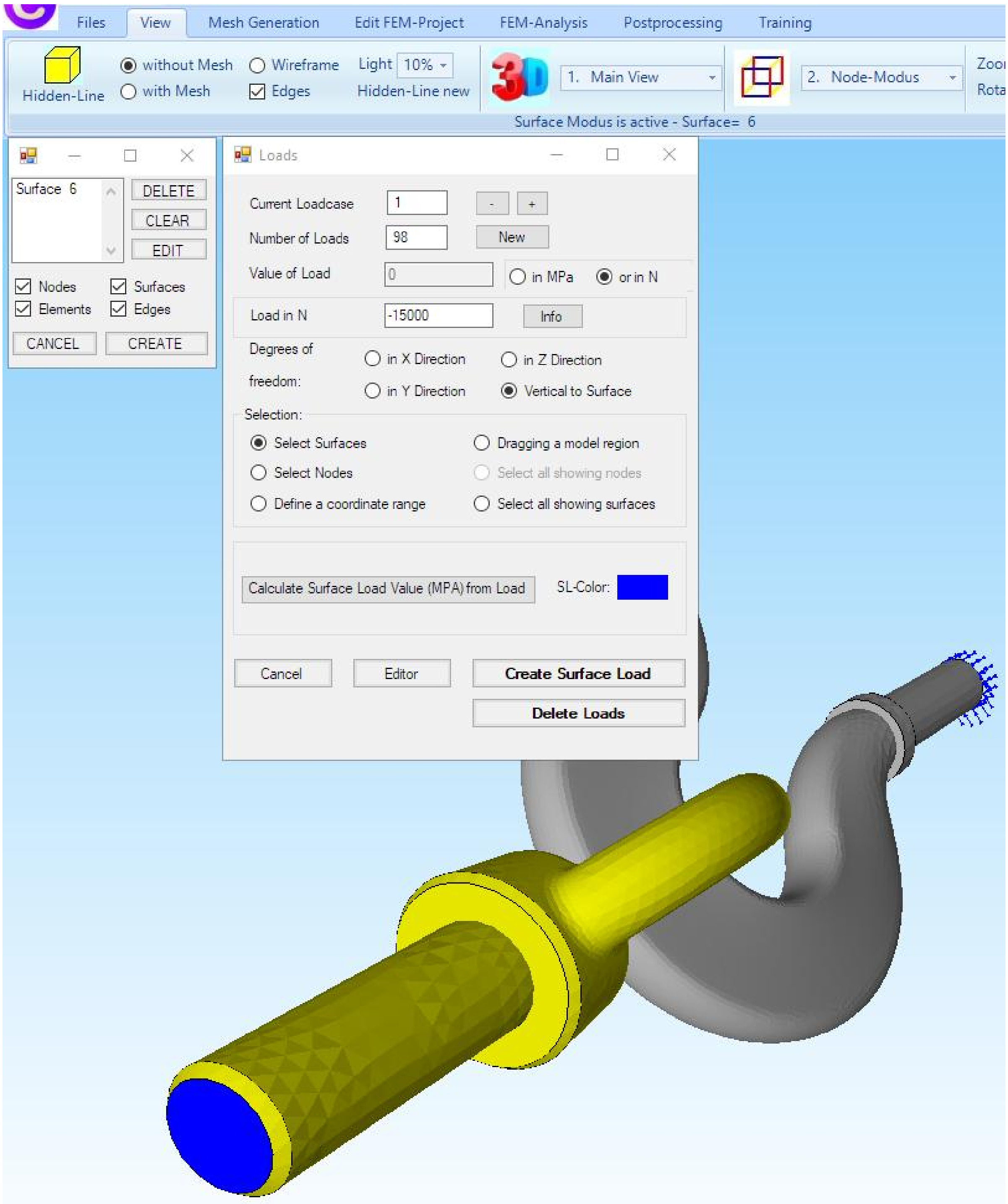


Load the bolt end with surface load

At the end, the bolt is loaded with -15,000 N in Y direction. Select the tab "Edit FEM Project" and "Surface Load" to load Surface 6 with -15000 N vertical to the surface.

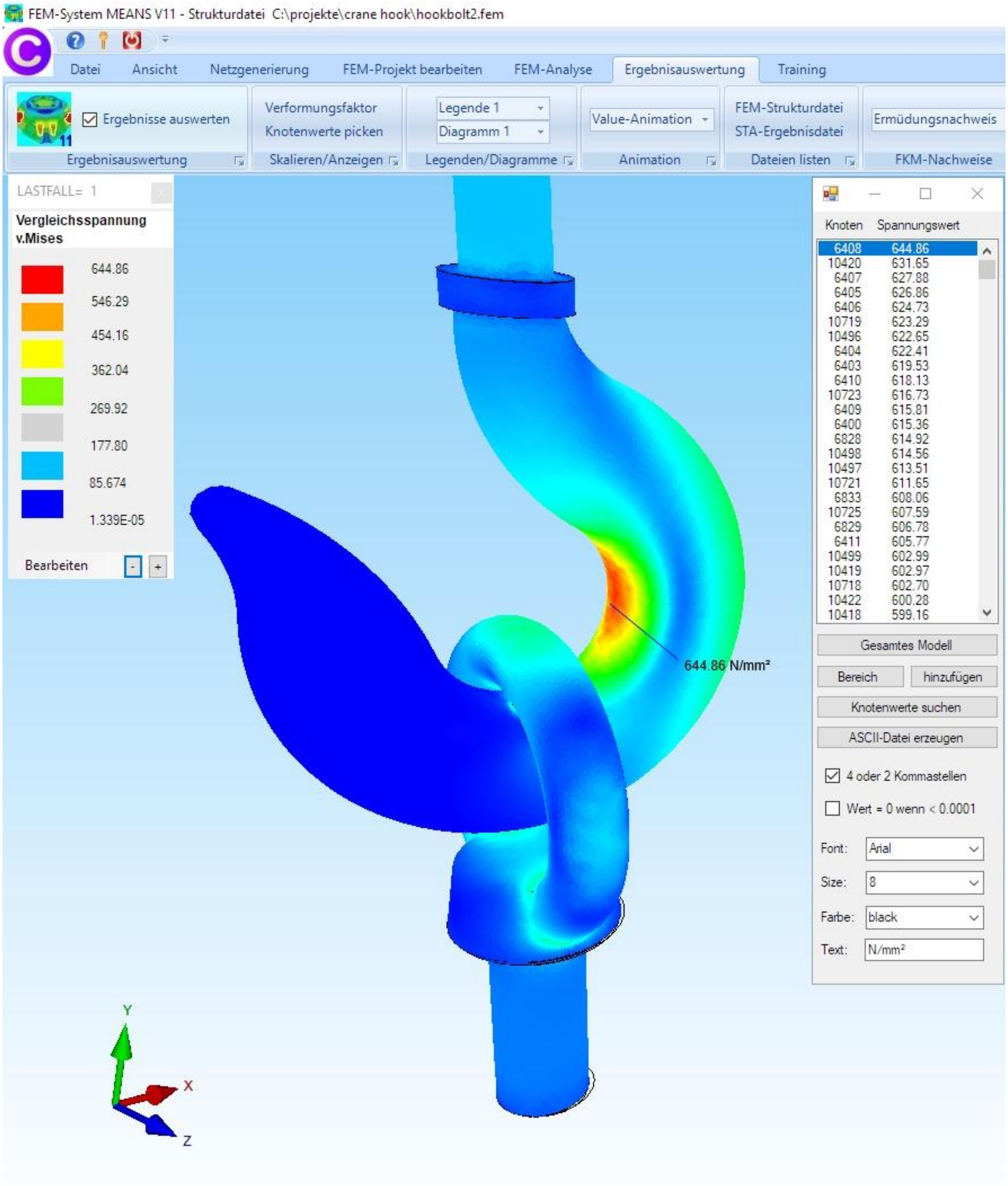
As of the July 2020 Update, the surface load can either be entered in MPa or in N, with the latter, the value is automatically calculated from the selected areas and divided by the N value:

$$\textbf{Surface Load (MPa)} = \frac{\textbf{Load (N)}}{\textbf{Area (mm}^2\textbf{)}}$$



Postprocessing

After the FEM analysis with the quick solver, the node stresses are evaluated.



The max. v.Mises Stress of the Assembly is 644 MPa and almost exactly matches the v.Mises Stress of the part with 637 MPa.

Bolt evaluation

Select “Edit FEM project” and “Element groups” and only show element group 1. Switch back to postprocessing to display the 1st principal Stresses S1 and the 3rd principle Stresses S3 with the maximal tensile and minimal compressive stresses.