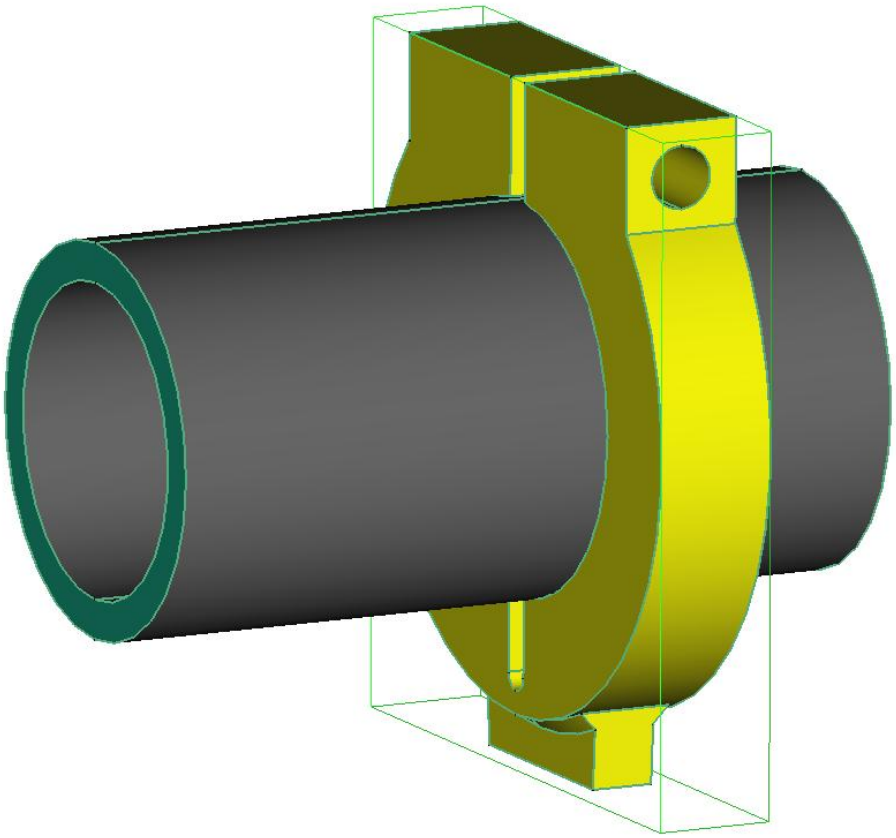
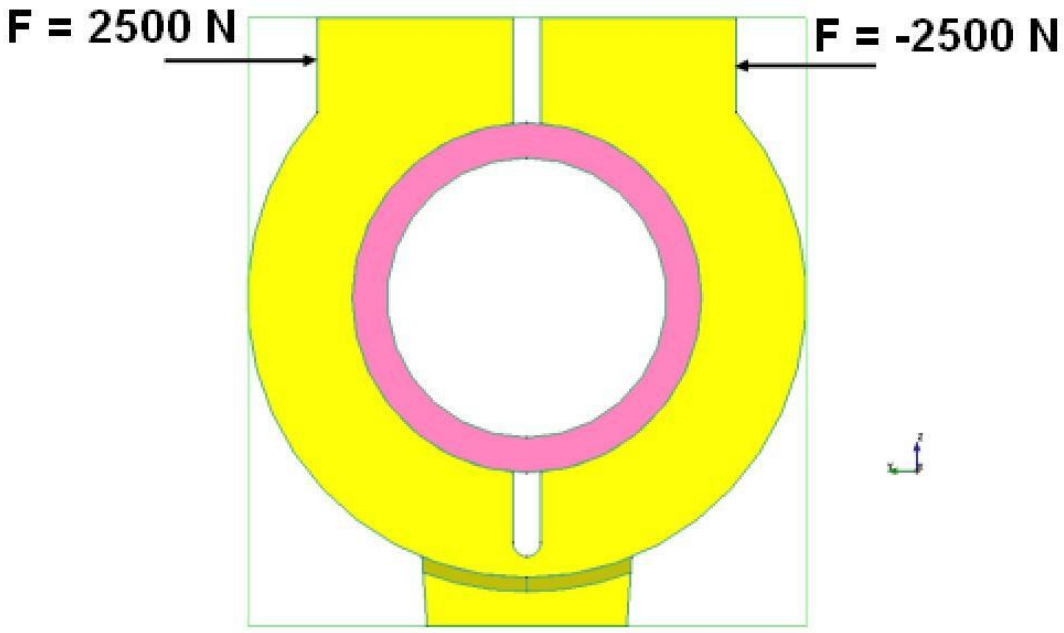
Part 36 - Contact-Analysis of a Clamp Connection with MEANS V13

The clamp connection CAD assembly consists of a clamp and a pipe section. The following calculations should be carried out:

1. How high is the contact stress on the pipe section when the clamp is firmly clamped with a screw tightening force of 2500 N.

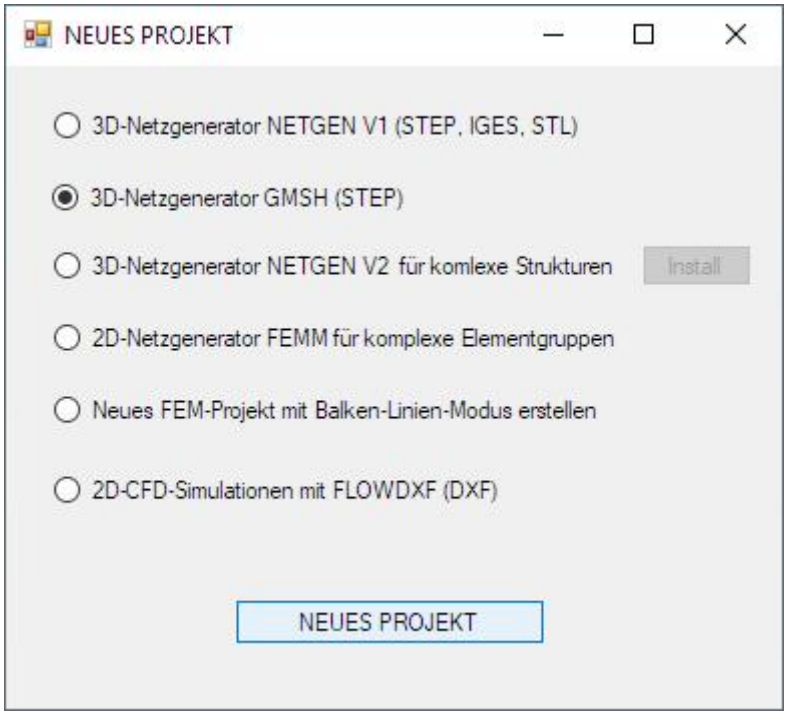
2. Calculate the contact stresses from the deformations calculated above as prescribed boundary conditions.



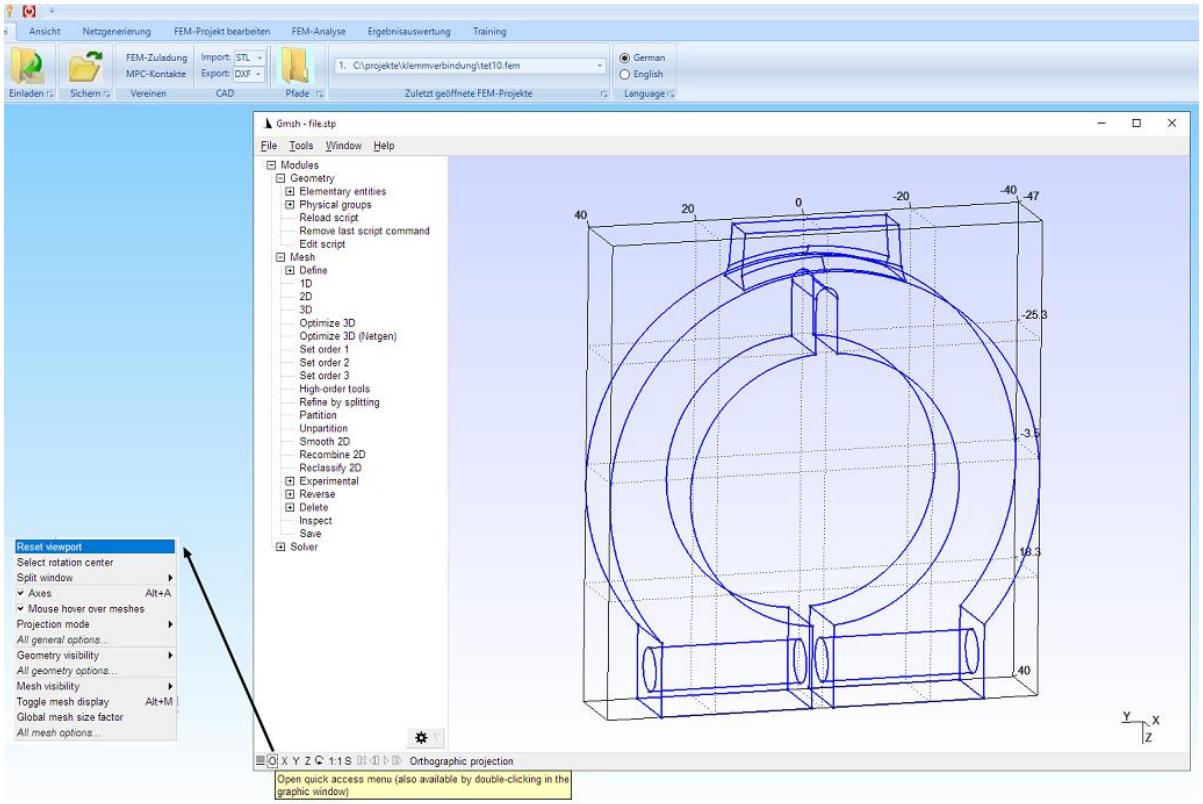


Generate FEM mesh

A very fine and uniform tetrahedral mesh is required for the contact analysis. This is why it is generated with the GMSH mesh generator, although in contrast to NETGEN, GMSH can only mesh relatively simple structures.

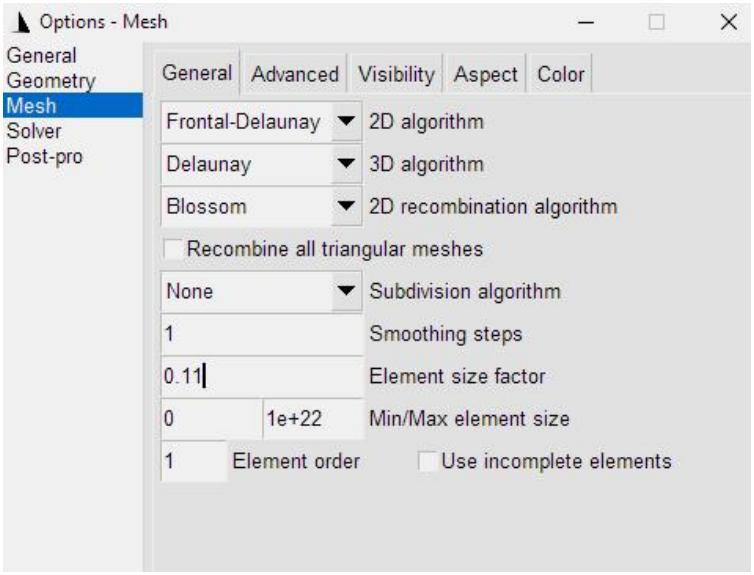


Select the “File” and “New” tab and “3D mesh generator GMSH (STEP)” and select the STEP file “Clamp.STP” as the new FEM project.

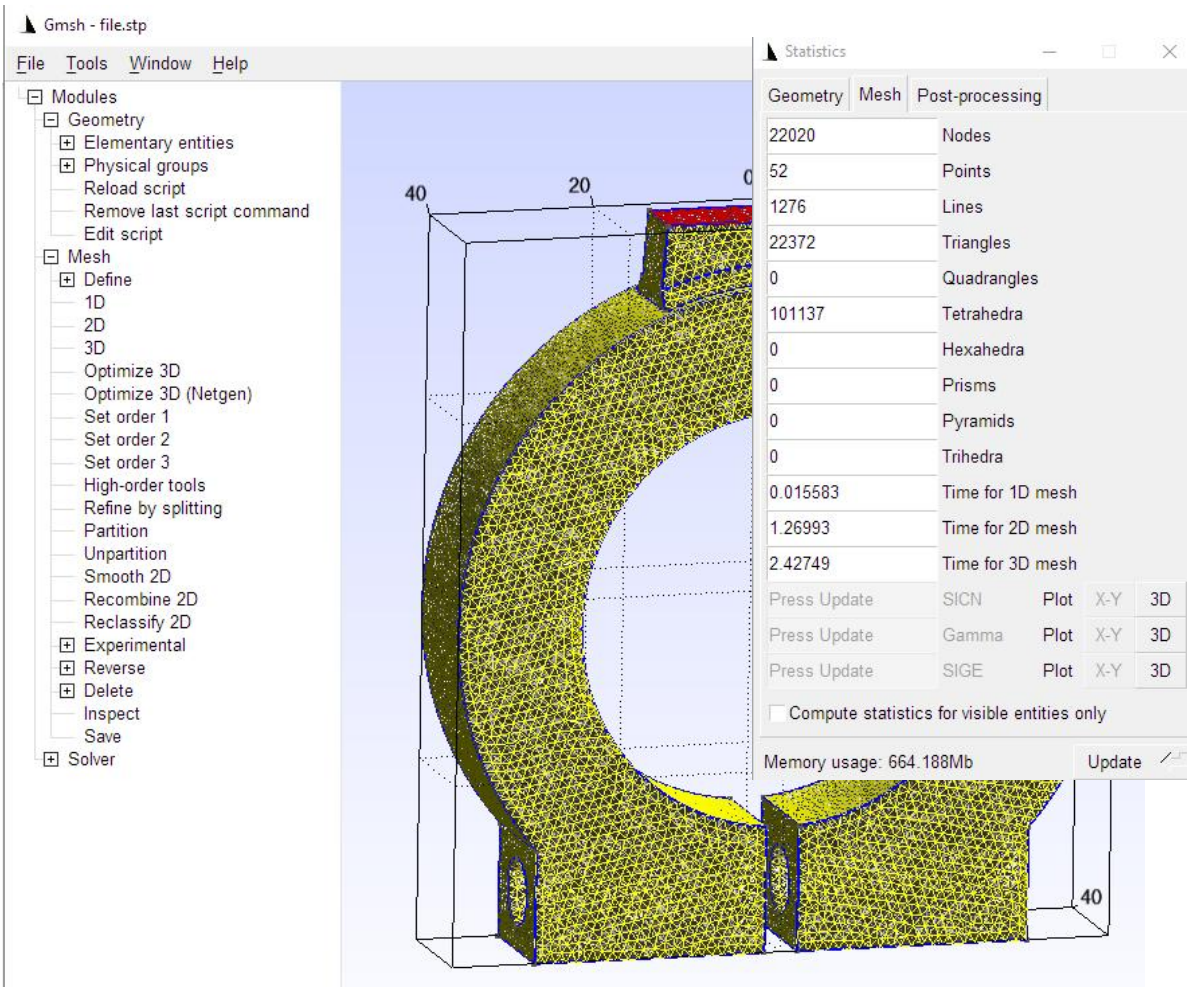


Set mesh density

Select the “Mesh” menu and the “General” tab to set the “Element size factor” to “0.11”, which allows you to generate a tetrahedral mesh with 101137 elements using the “1D” and “3D” menus.

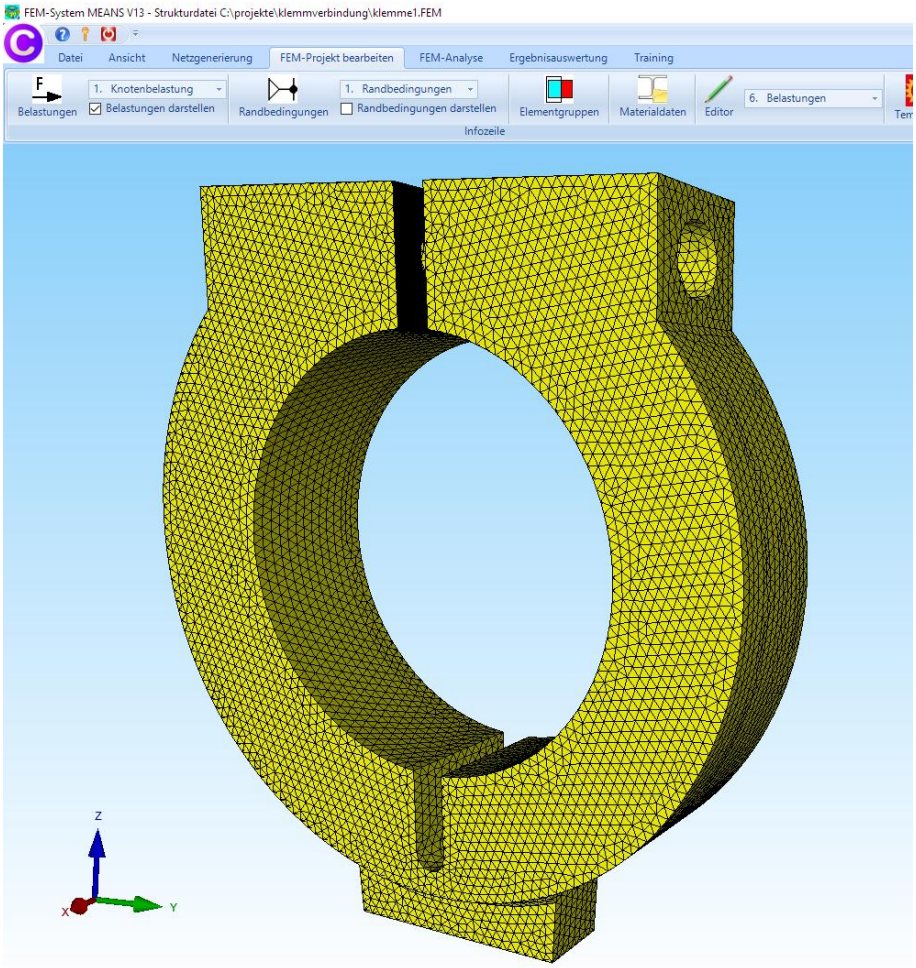


If the model is not visible at first, select **“Reset viewport”** or double-click on the screen to recalculate the overall view.



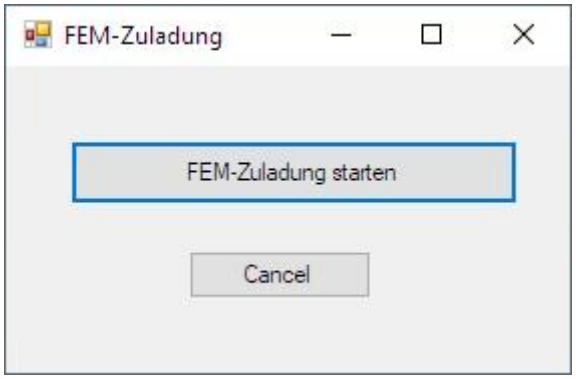
Export in Abaqus format (INP)

Select the menu "Tools" and "Statistics" to check the key data of the mesh generation, then select "File" and "Export" and export the mesh in Abaqus format (INP) and import it into MEANS via the INP interface. Then generate the pipe section with "0.12" and 116,494 TET4 elements.



Load the clamp and pipe section together in MEANS V13

The two FEM meshes can be loaded together using the "File" tab and the "FEM load" menu. The new FEM model now consists of 217,631 elements, 49,012 nodes and 2 element groups.



Defining the contact surfaces

The following 3 load cases must be created for a contact analysis:

Load case 1: Load on the component. If there are several load cases, all load cases must be added to load case 1.

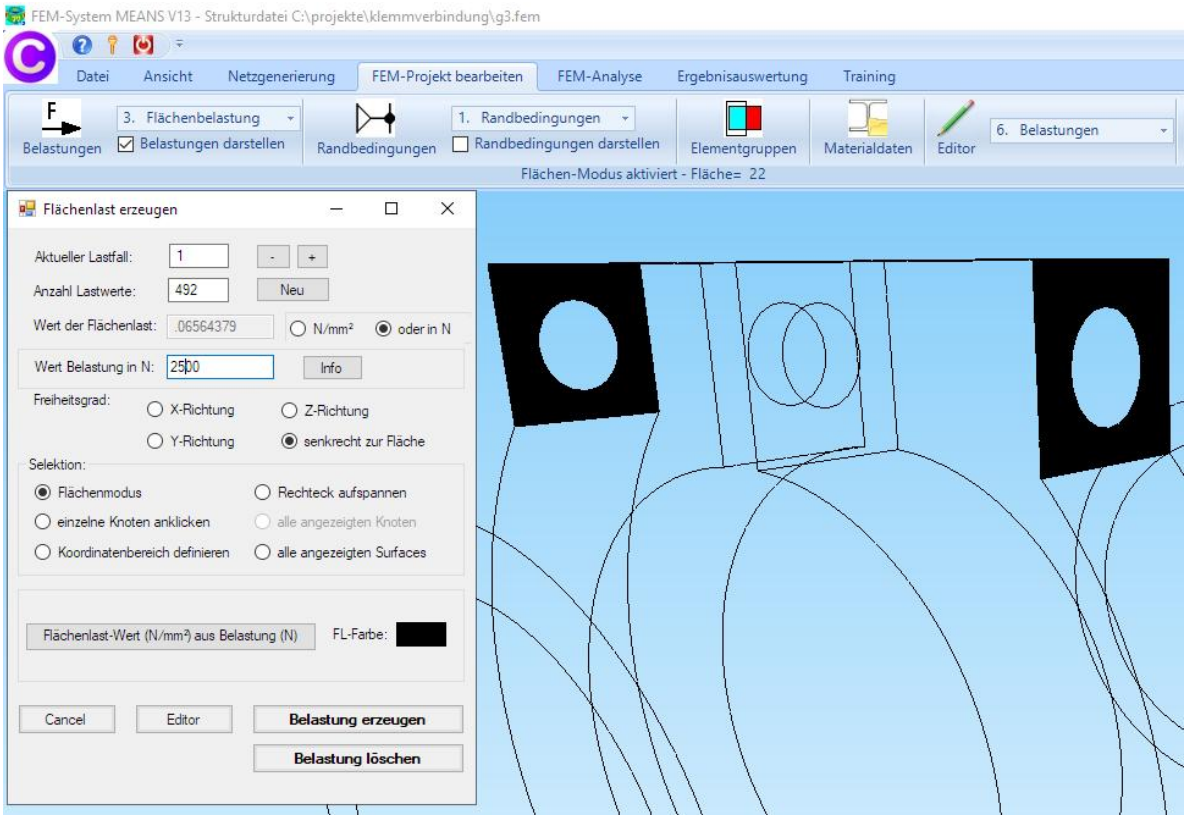
Load case 2: Define master contact surface with a surface load

Load case 3: Define slave contact surface with a surface load

Node contacts can be converted into a surface load by selecting "all displayed nodes".

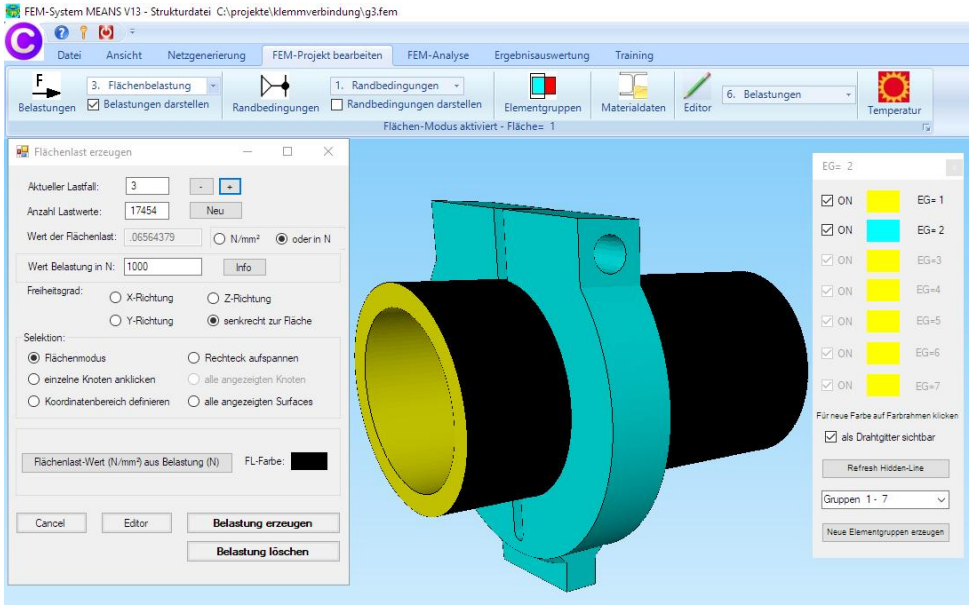
Load case 1 with Screw Force

Select the "Edit FEM project" tab and the "Surface load" menu and create load case 1 on surfaces 22 and 23 with a surface load of 2500 N "vertical to surface" on both sides.



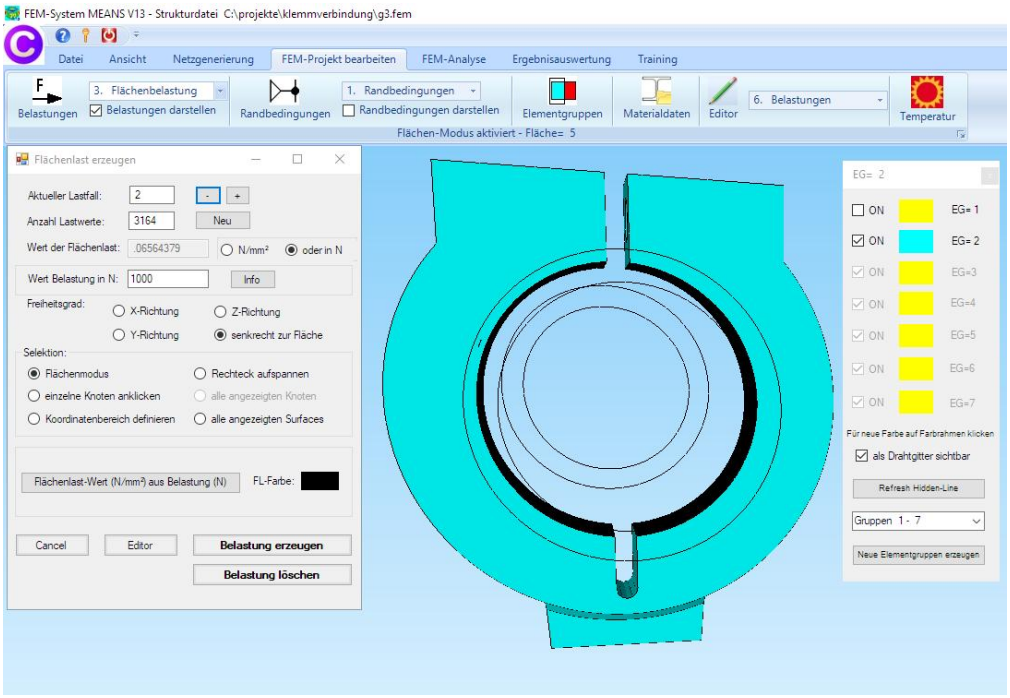
Load case 2 with Master Contact Surface

In order for the outer surface of the pipe section to be selectable again, element group 2 must be displayed.
Select the “Edit FEM project” tab and the “Surface load” menu and create load case 2 on the outer surface 1 with a surface load “vertical to surface”.
Since this surface load defines the master contact nodes, no load value is required.



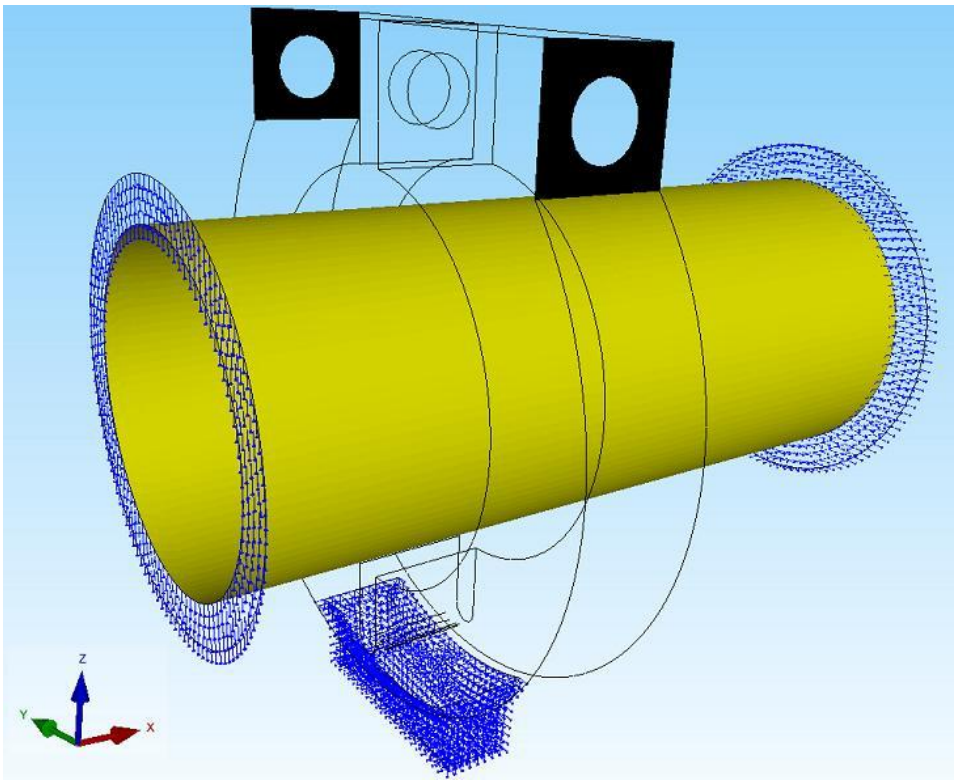
Load case 2 with Slave Contact Surface

In order to select the inner surface of the clamp, the pipe with element group 1 must first be hidden.
Select the "Edit FEM project" tab and the "Surface load" menu and create load case 2 on the inner surfaces 5 and 6 with a surface load "vertical to surface".
Since this surface load defines the slave contact nodes, no load value is required.



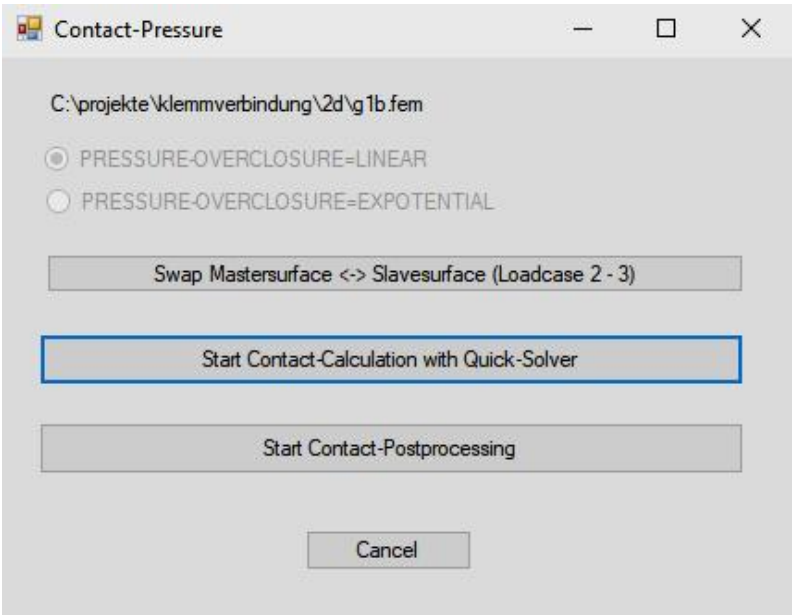
Create clamping

Using the "Edit FEM project" tab and the "Boundary conditions" menu, the lower part of the clamp is firmly clamped in the x, y and z directions. To prevent the pipe from moving, the edge on the left must be held in the y direction and the right in the xz direction.



FEM Analysis

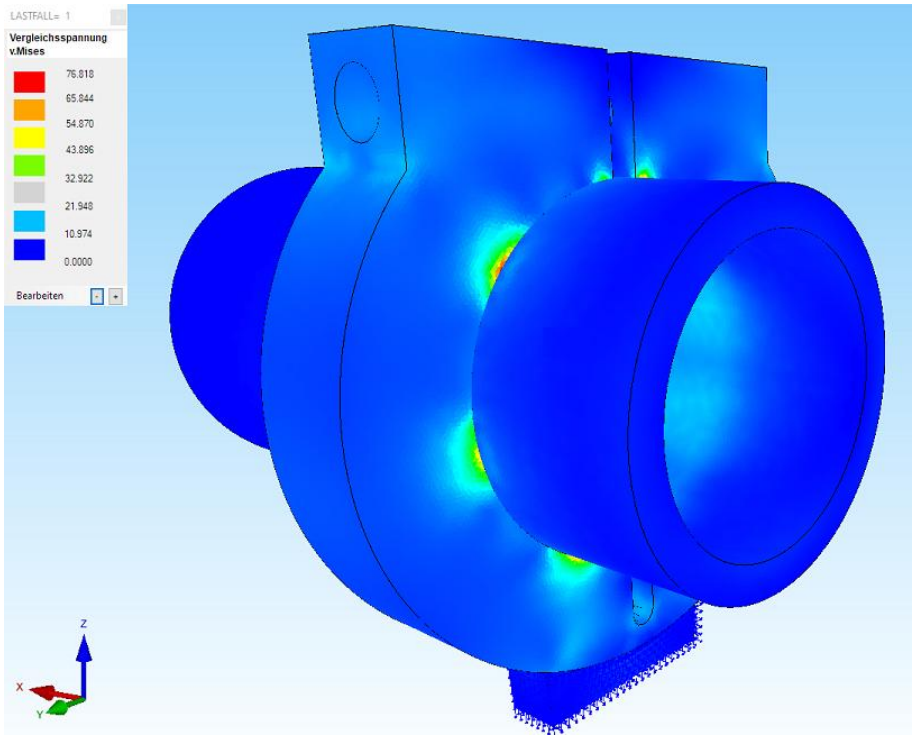
Using the "FEM Analysis" tab and the "Contact-Analysis" menu, the contact stresses are calculated with the quick solver after around 10 iterations. The master and slave contact surfaces can also swaped.



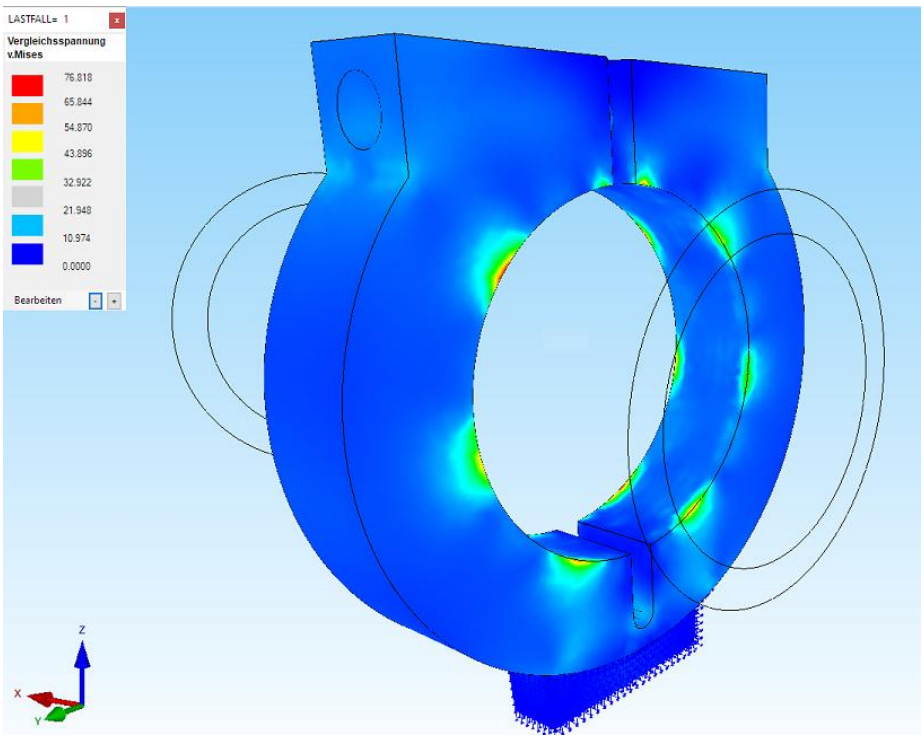
Postprocessing

Select the "Postprocessing" tab and the icon  and in the next post-processing dialog box "Node Stresses" and "v.Mises Stresses" to evaluate the following stress distributions.

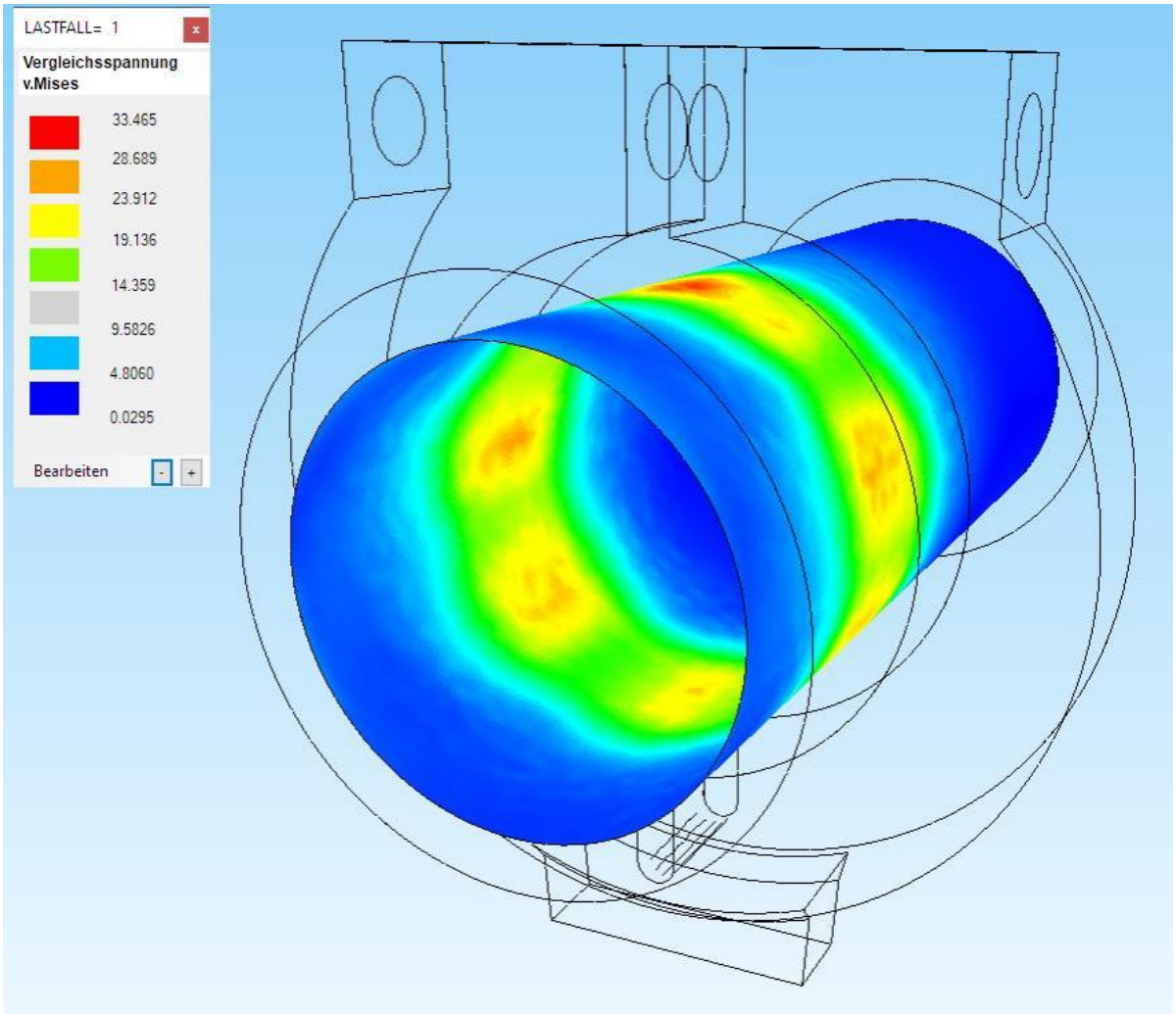
Contact stress on clamp and pipe section = 76 N/mm²



Contact Stress on the Clamp = 76 N/mm²



Contact stress on the inside of the pipe section = 33.4 N/mm²



Hide an Show Surfaces

To display only the inside surface of the pipe section, select the "View" and "Surfaces" tabs and in the right-hand side menu select "Show Surfaces" and click on surface 2.

Evaluation of results

The result for the stresses is v.Mises equivalent stresses of approx. 30 N/mm². For this example task, this means that the screw force is harmless because the yield strength of round steel hollow sections is between 235 MPa and 355 MPa.

Generate prescribed boundary conditions

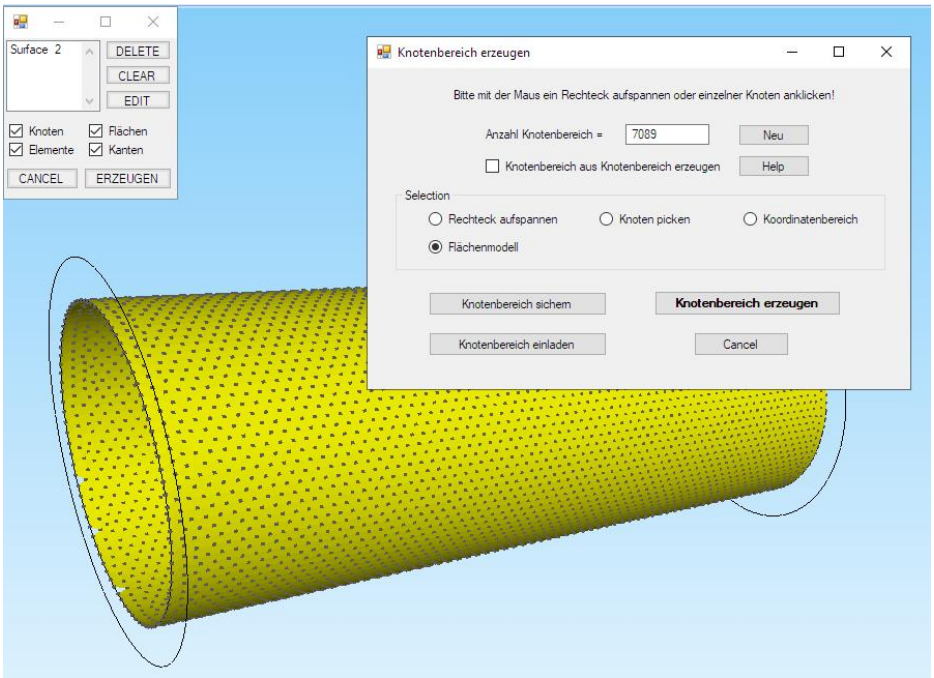
If the deformations are known, the contact stresses can also be calculated using prescribed boundary conditions in the following 4 steps.

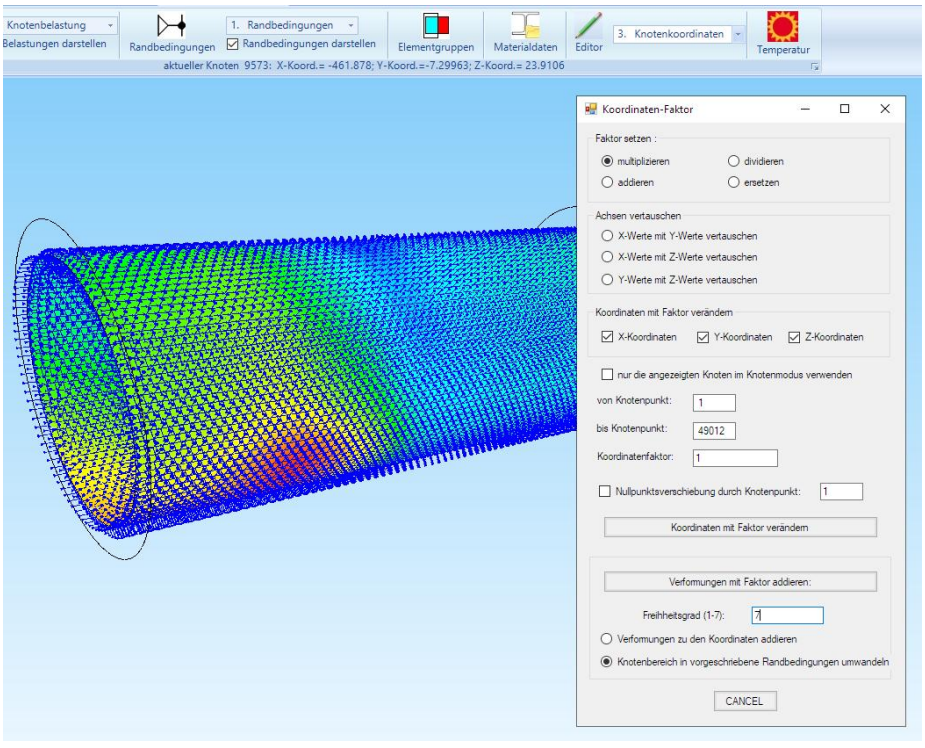
Step 1: Generate a node area from the inner surface 2

Step 2: Display a stress or deformation distribution

Step 3: Use the coordinate factor to convert the deformations of the node area into prescribed boundary conditions with “7”

Step 4: Calculate contact stresses using a normal linear statics analysis (no contact analysis) and without loading





The result is the same contact stresses of 33.7 MPa as previously determined with the contact analysis.