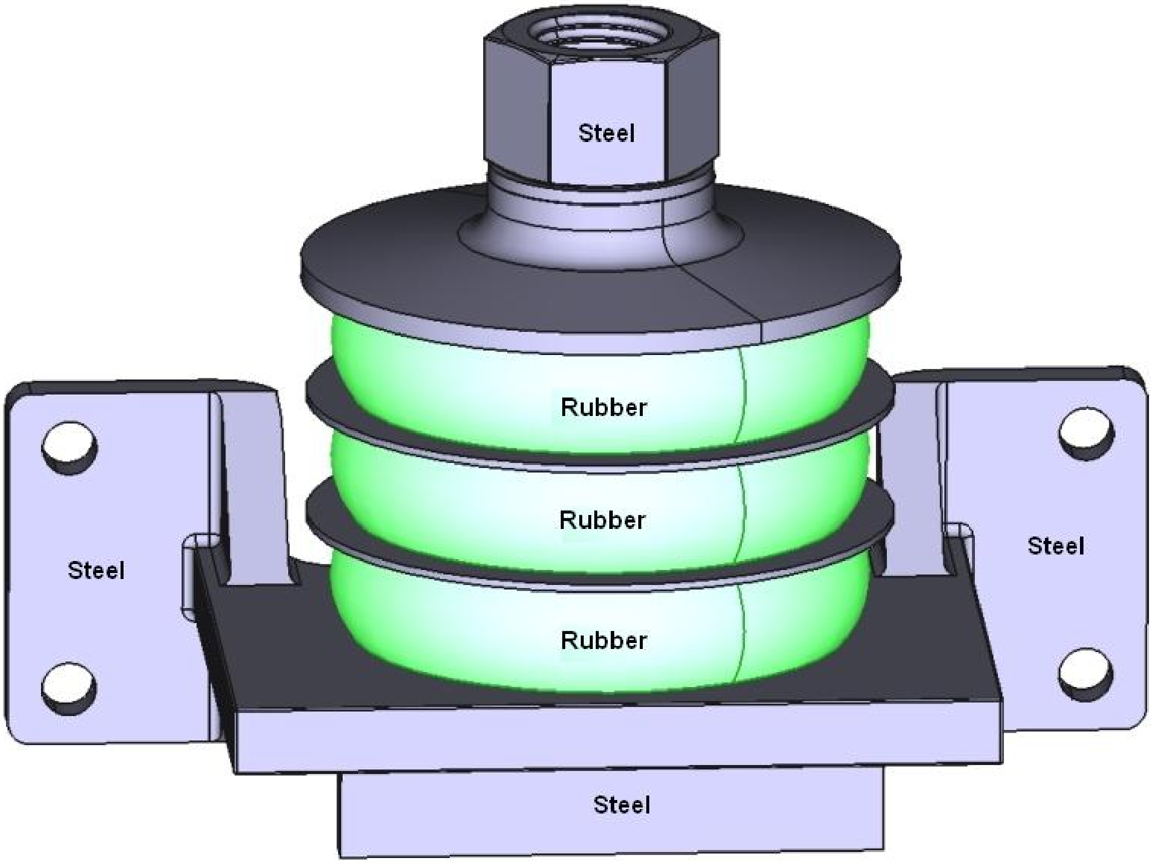
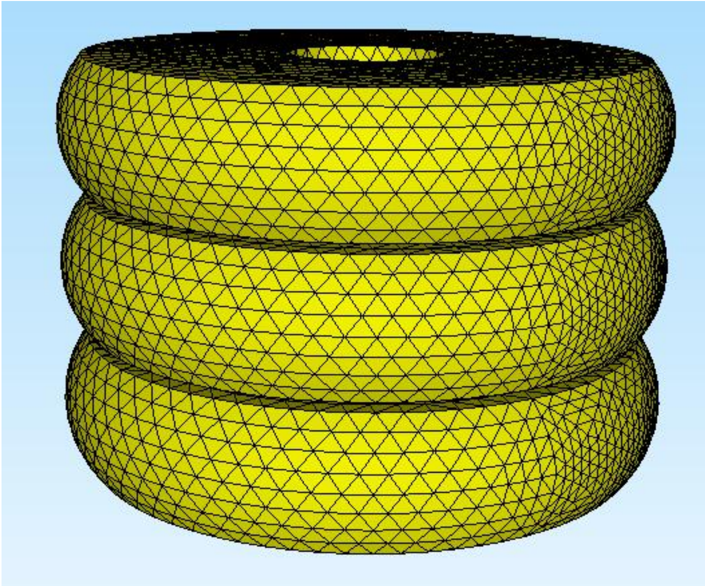
Part 27: CAD Assembly with Node Imprint calculate

Part 1: Steel-Rubber Buffer

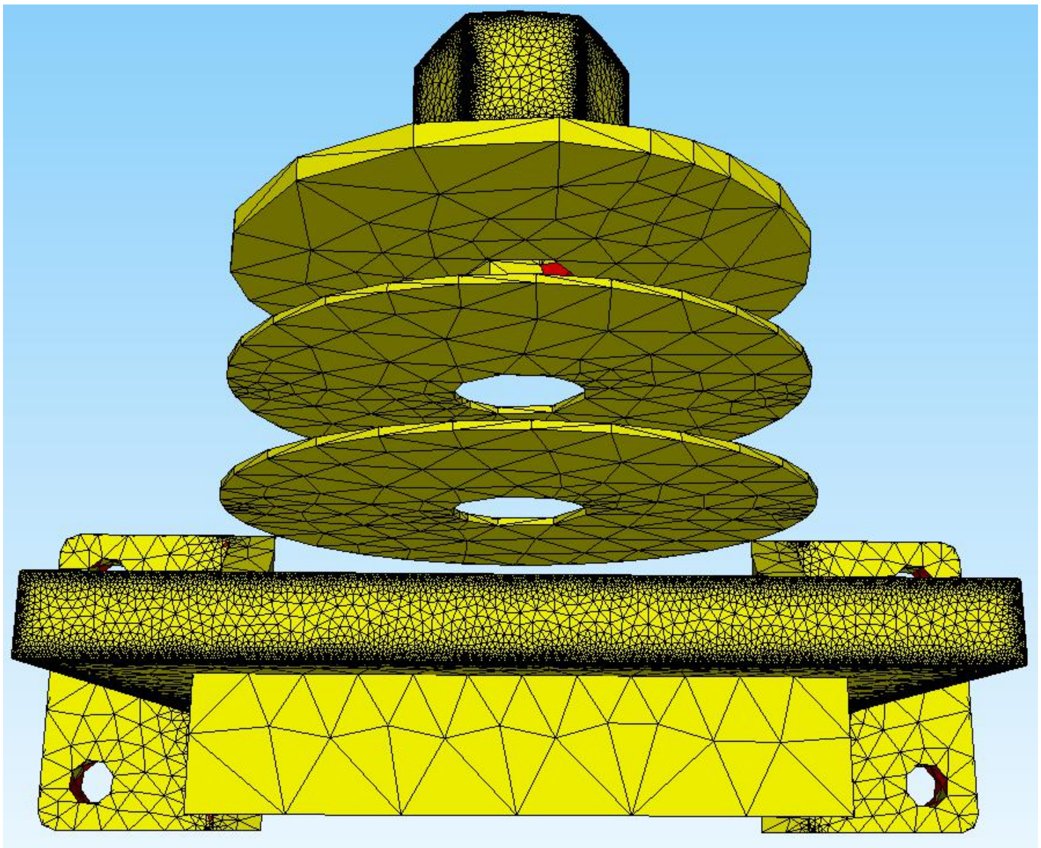
The CAD Assembly of a Steel-Bracket with an E-Modulus = 210 000 MPa and a Rubber-Buffer with an E-Modulus = 15 MPa cannot be meshed in one step because of the different materials.
With MEANS V12, such structures can be meshed with different materials by first separately meshing all components with uniform materials and reuniting them with a node imprint of the contacting mesh.
In this way, even complex welded constructions or thin container structures can be meshed up to a certain degree of difficulty.
Unfortunately, numerically unstable tetrahedra with a negative Jacobi determinant can also arise when generating the node imprint, but these can be corrected, displayed or deleted in MEANS V12.



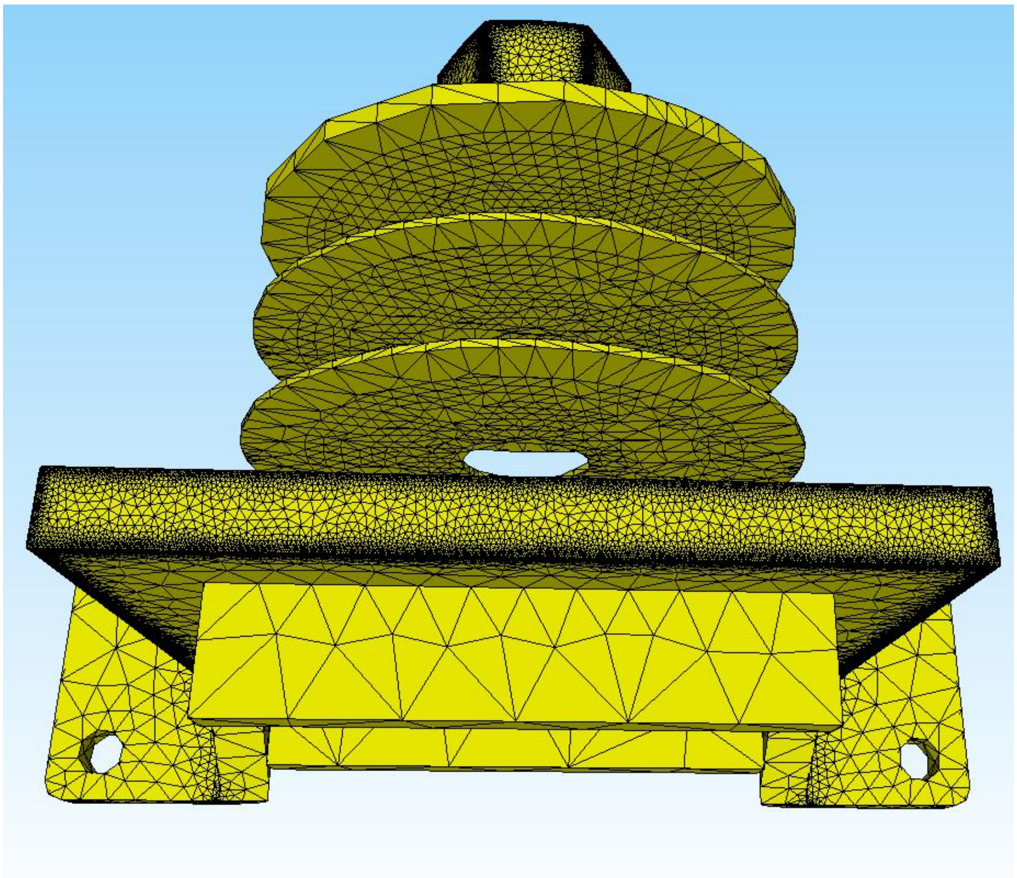
Rubber Buffer generated separatly with GMSH:



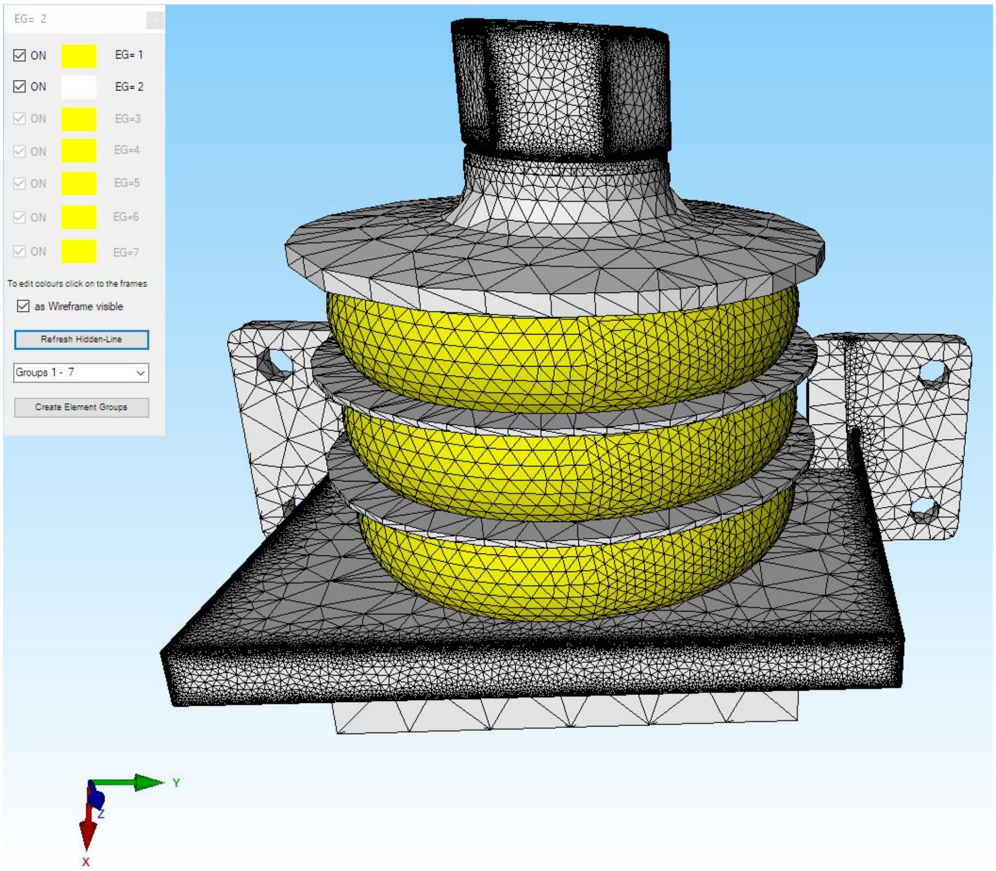
Steel Bracket generated separatly with NETGEN:



Steel Bracket with the Node Imprint of the Rubber Buffer:



FEM mesh combined with Steel Bracket and Rubber Buffer

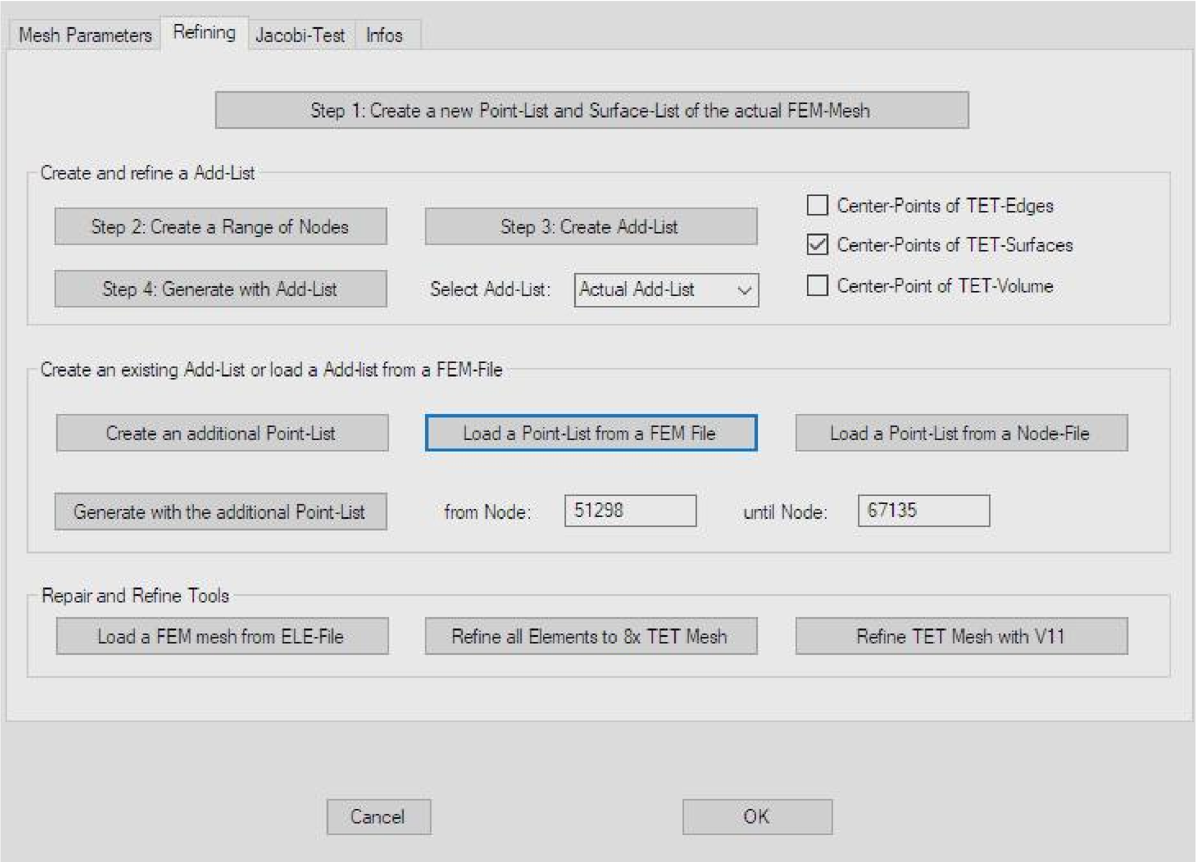


Generate Node Imprint

Either the Node Imprint of the rubber buffer can be inserted into the mesh of the steel bracket or the Node Imprint of the steel bracket can be inserted into the mesh of the rubber buffer. A FEM file with all node coordinates or a faster Add List with a range of nodes can be used to generate the Node Imprint.

Create the Node Imprint of the rubber buffer

First load the FEM mesh of the steel bracket and select the “Mesh generation” and “Local Refinement” tabs as well as menu “Step 1: Create a new Point-List and Surface-List of the actual FEM-Mesh“ to save the nodes of the actual mesh. Then select menu "Load a Point-List from a FEM-File“ to load the coordinates of the rubber buffer. Then select the menu "Generate with the additional Point-List“ to add the Add-List to the steel bracket mesh. The refined FEM mesh shown above is obtained.

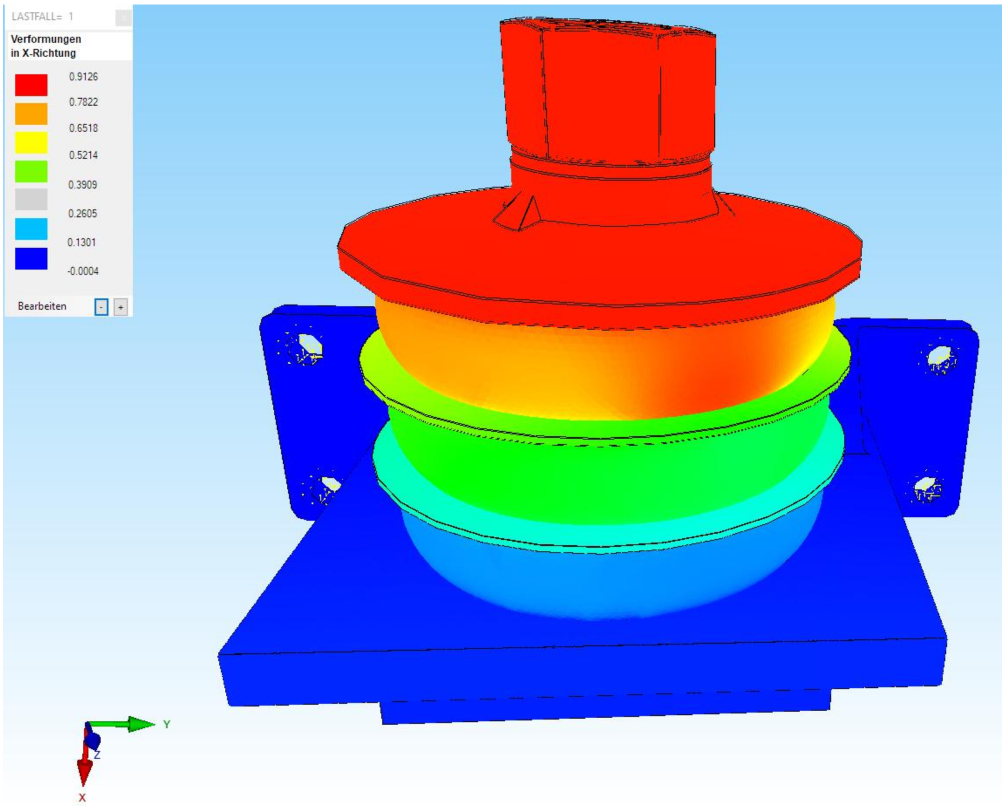


Download the Video-MP4, FEM-, INP- and FRD-File for the Steel-Rubber-Buffer

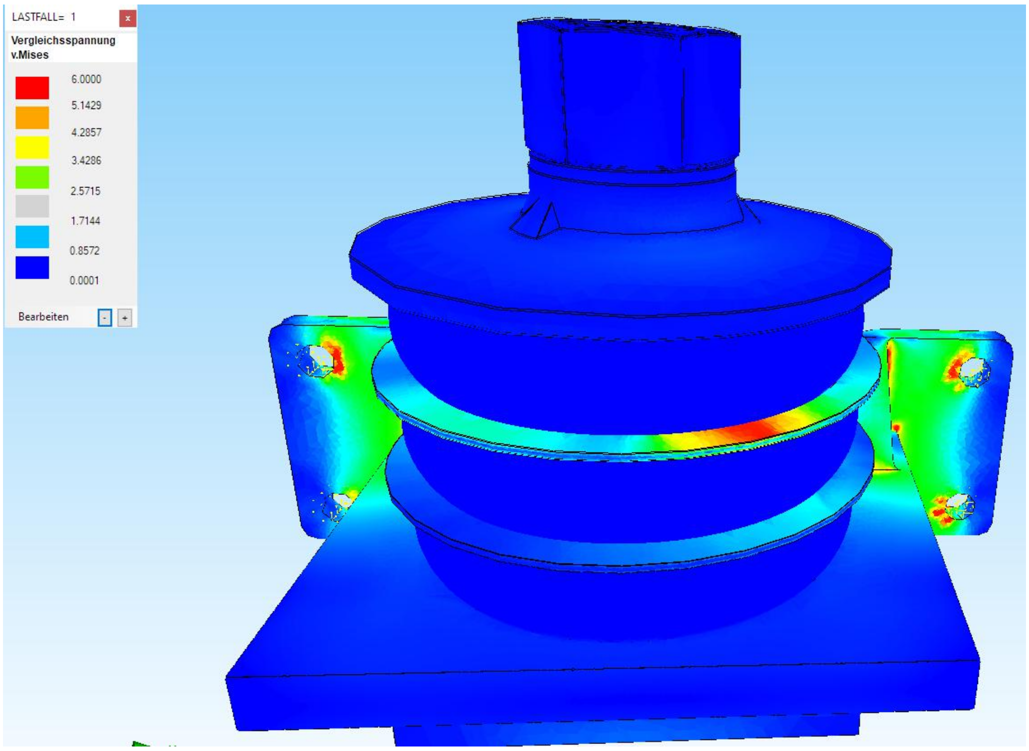
Postprocessing

With Register „Postprocessing“ and the Icon  the results of displacements and stresses can be evaluated.

Max. Displacements in X Direction = 0.912 mm

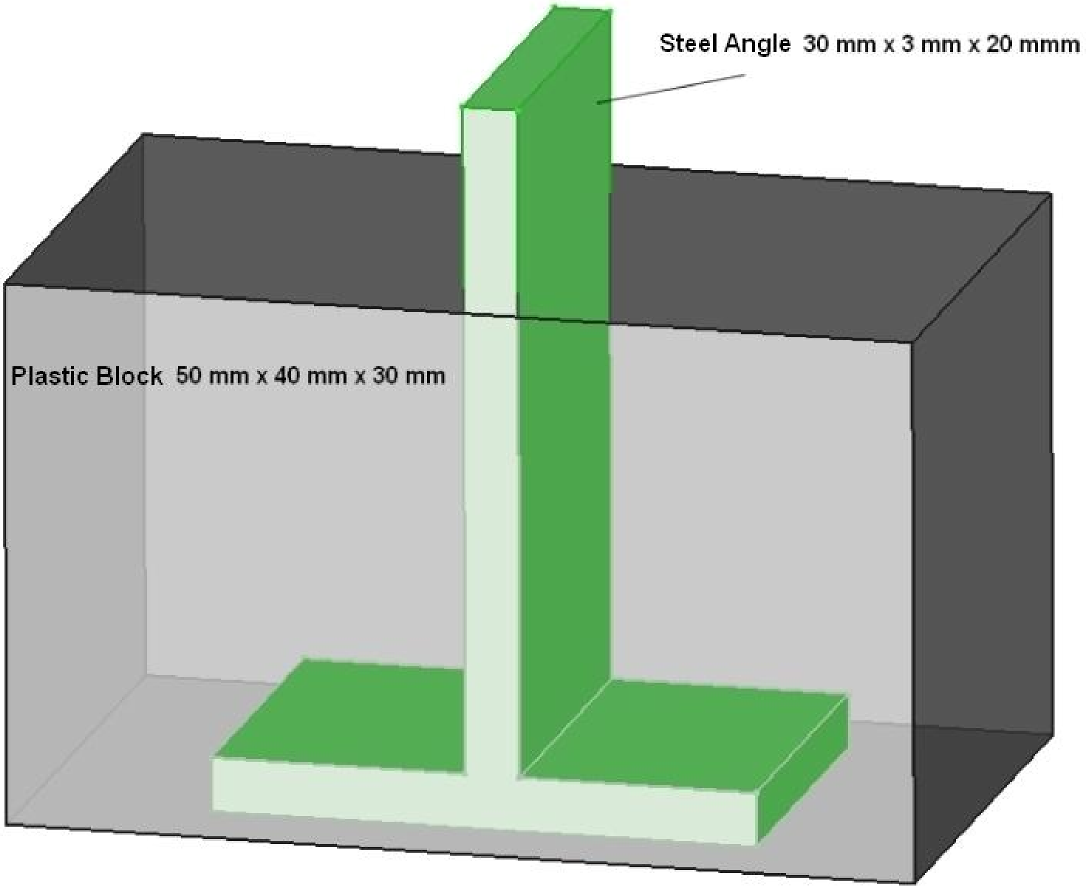


Max. v.Mises-Stresses = 6.51 MPa

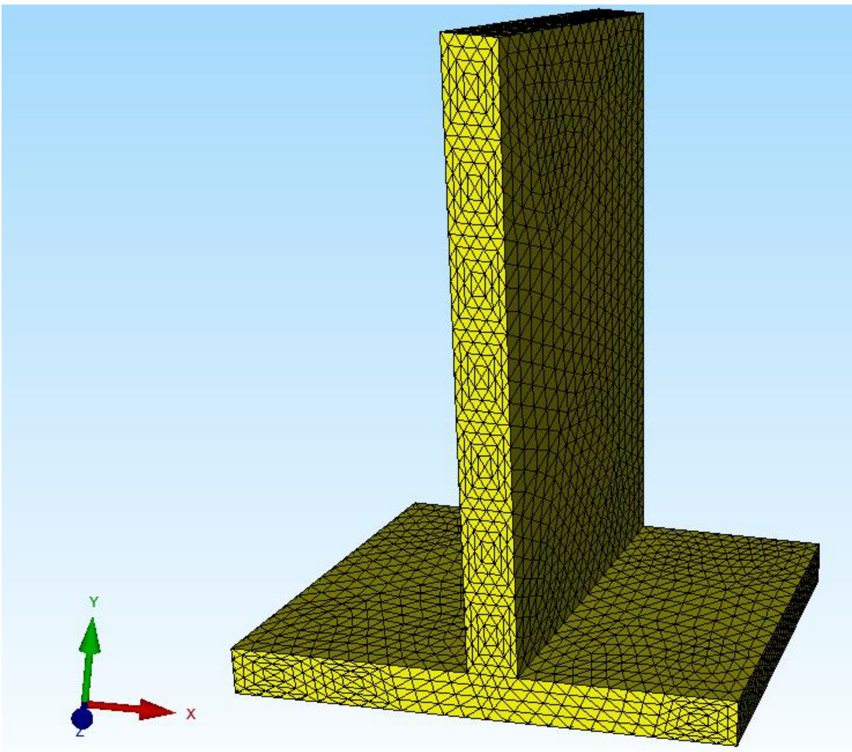


Part 2: Steel Angle with Plastic Block

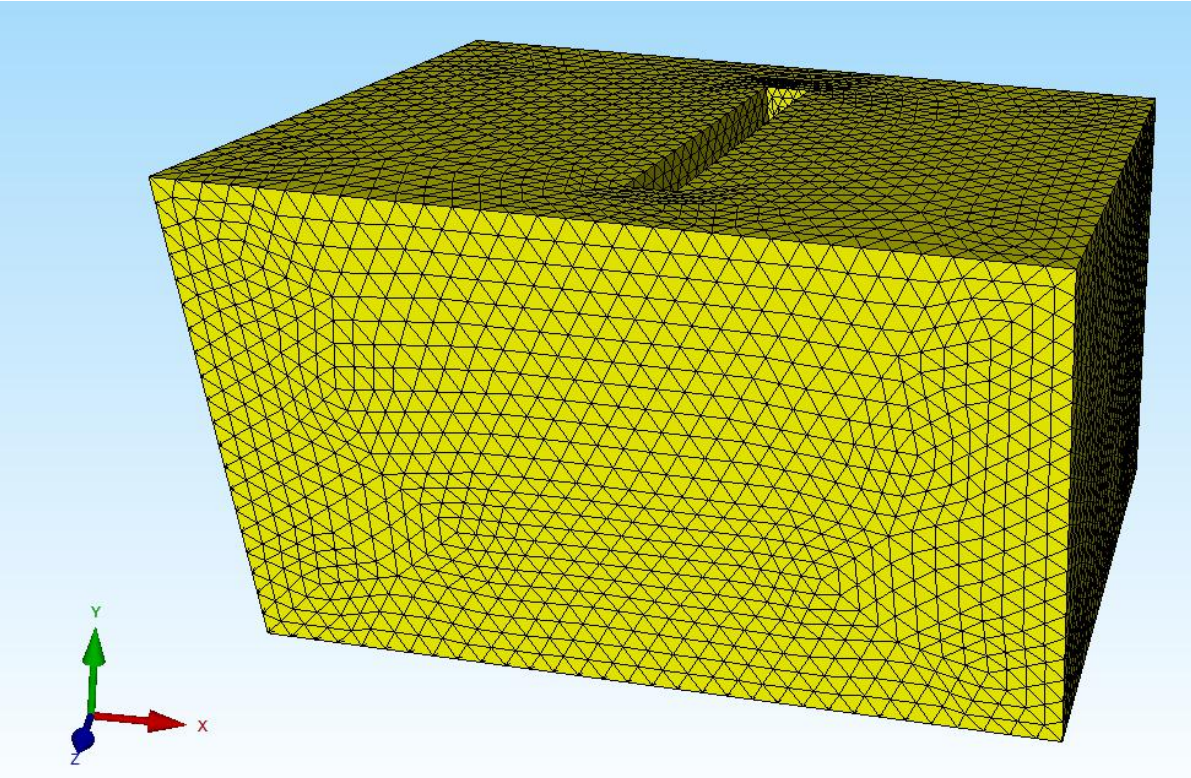
A CAD assembly consisting of a Steel Angle with an E-Modulus = 210 000 MPa that was embedded in a Plastic Block with an E-Modulus = 1200 MPa cannot be meshed in one step because of the different materials.

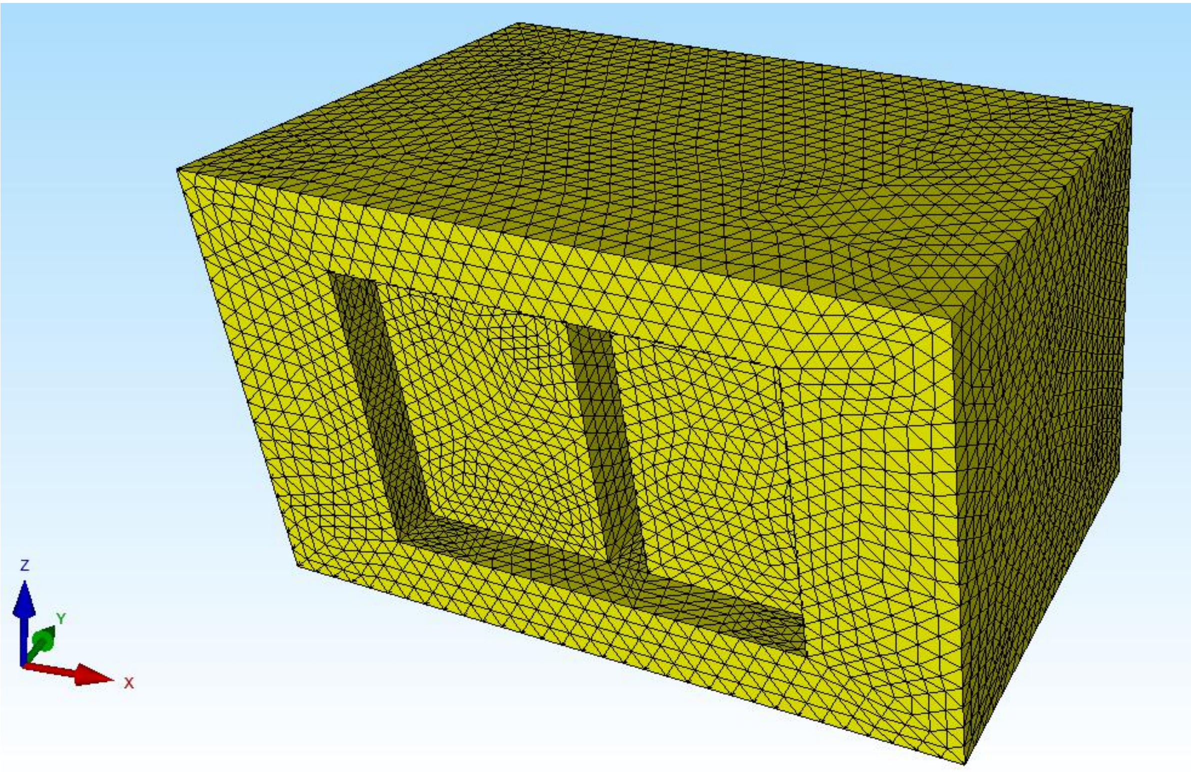


The Steel Angle and Plastic Block must first be meshed separately with GMSH:



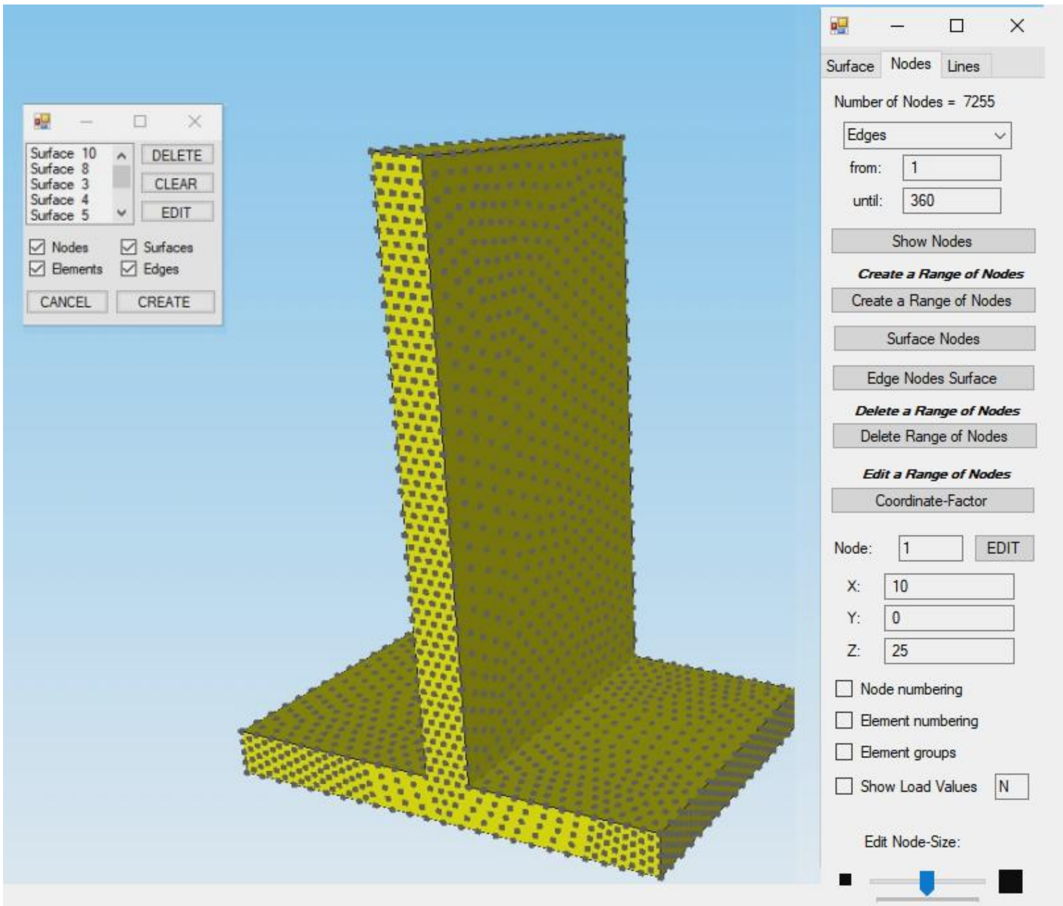
FEM-Mesh Plastic Block from above and below:



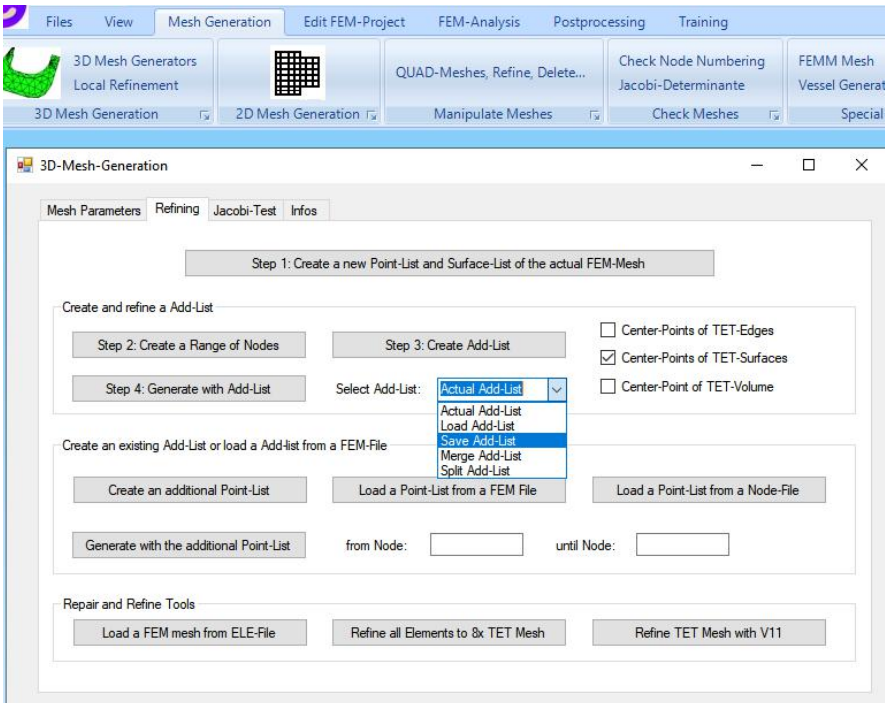


Create a Range of Nodes of the Steel Angle

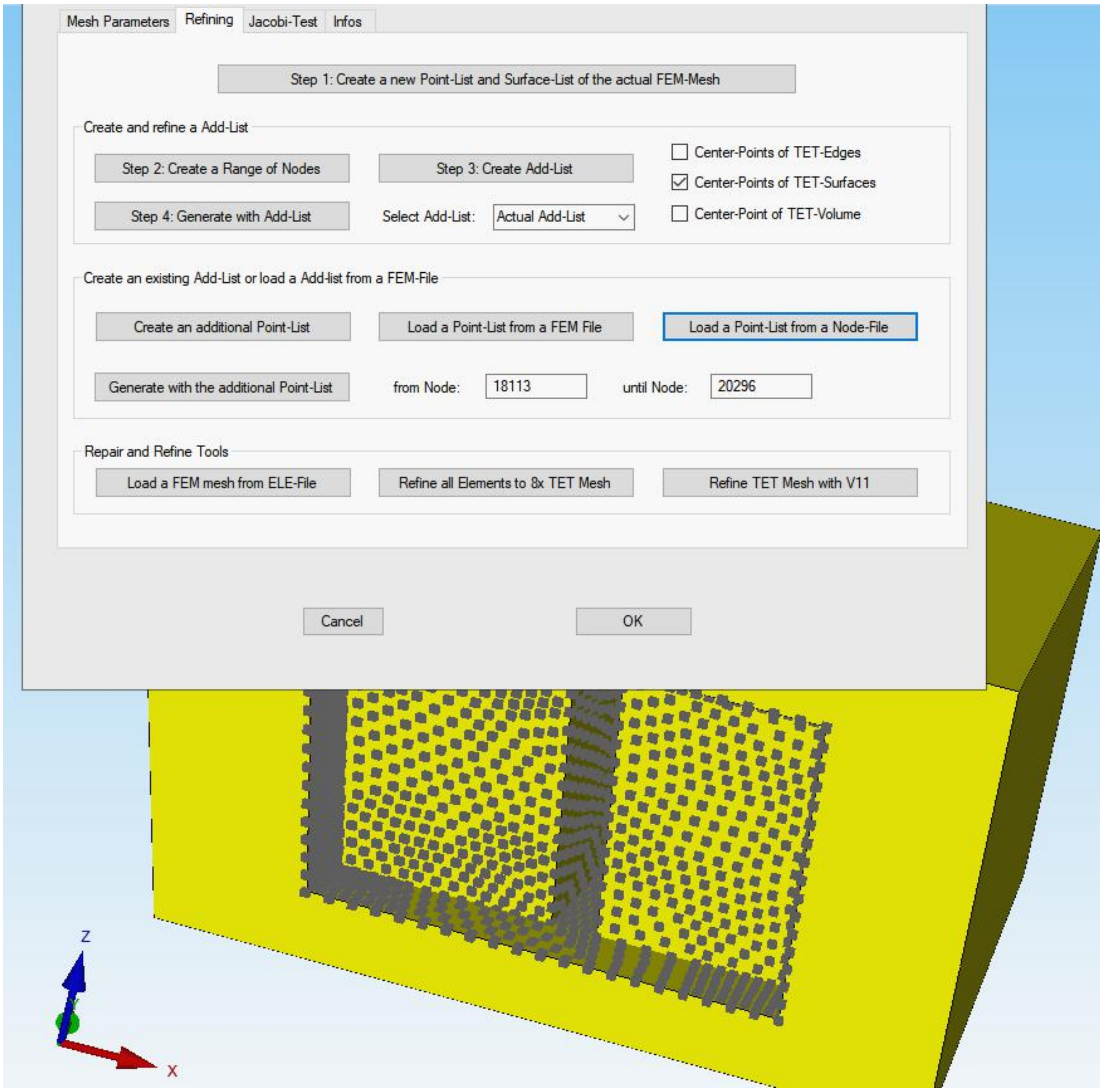
Load the FEM mesh of the Steel Angle and create a Range of Nodes of all surfaces that contact the Plastic Block with the Register "View" and "Node-Modus" as well as the menu "Surface Nodes" except the upper and lower ones.



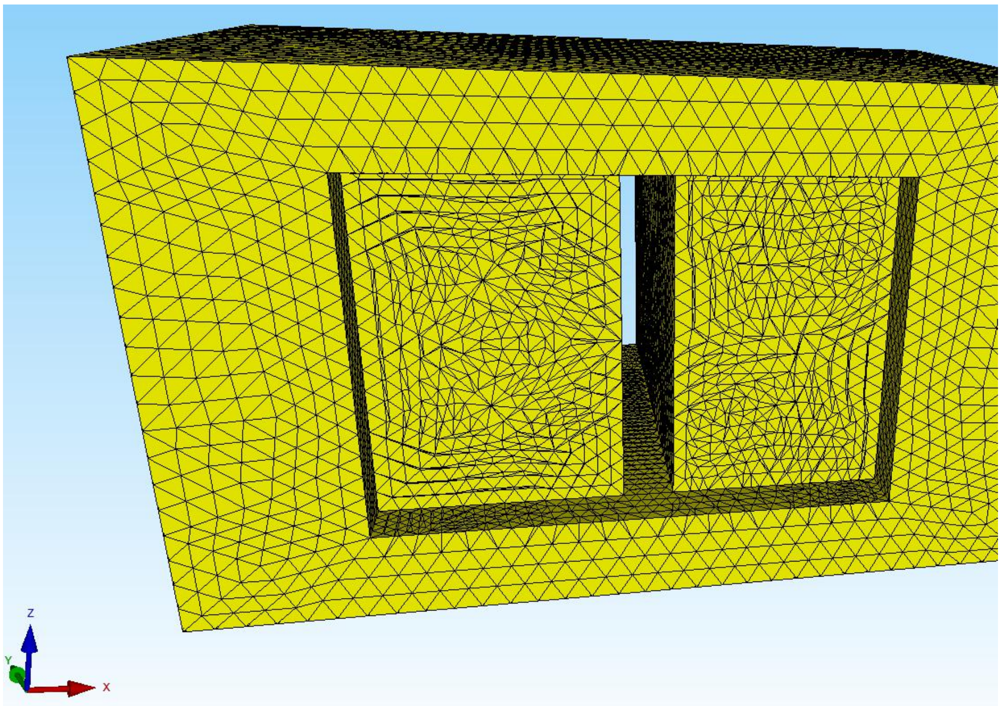
Then you create an Add List with the Register "3D Mesh Generators" and "Local Refinement" as well as menu "Save Add-List" or "Create an additional Point-List".



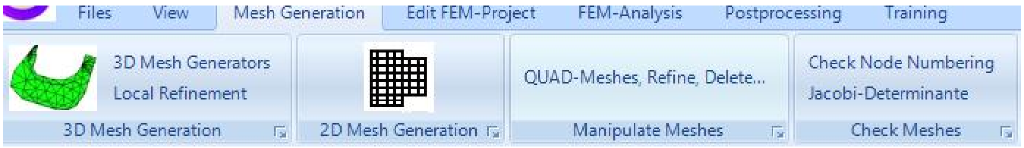
Load the FEM mesh of the Plastic Block. Select in the same dialog box menu "Step 1: Create a new Point-List and Surface-List of the actual FEM-Mesh“ and then select the menu "Load a Point-List from a Node-File“ to load the Add-List of the Steel Angle and check it on the model.
Then select the menu "Generate with the additional Point-List" to create a Node Imprint of the Steel Angle in the Plastic Block.

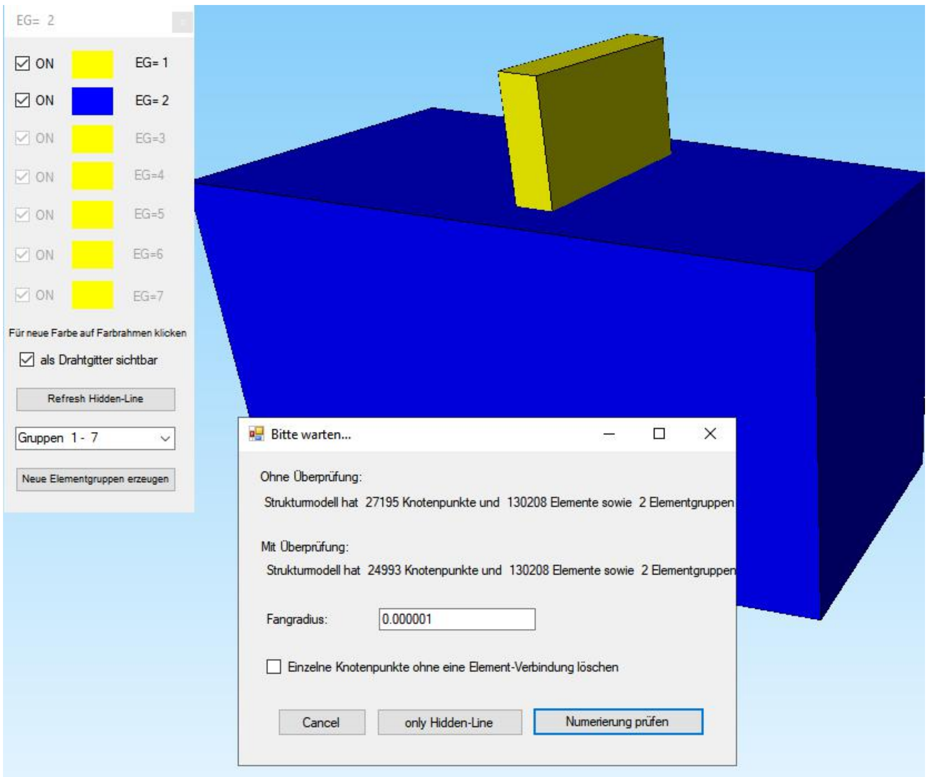


Then the following FEM mesh is obtained with the Node Imprint of the Steel Angle:



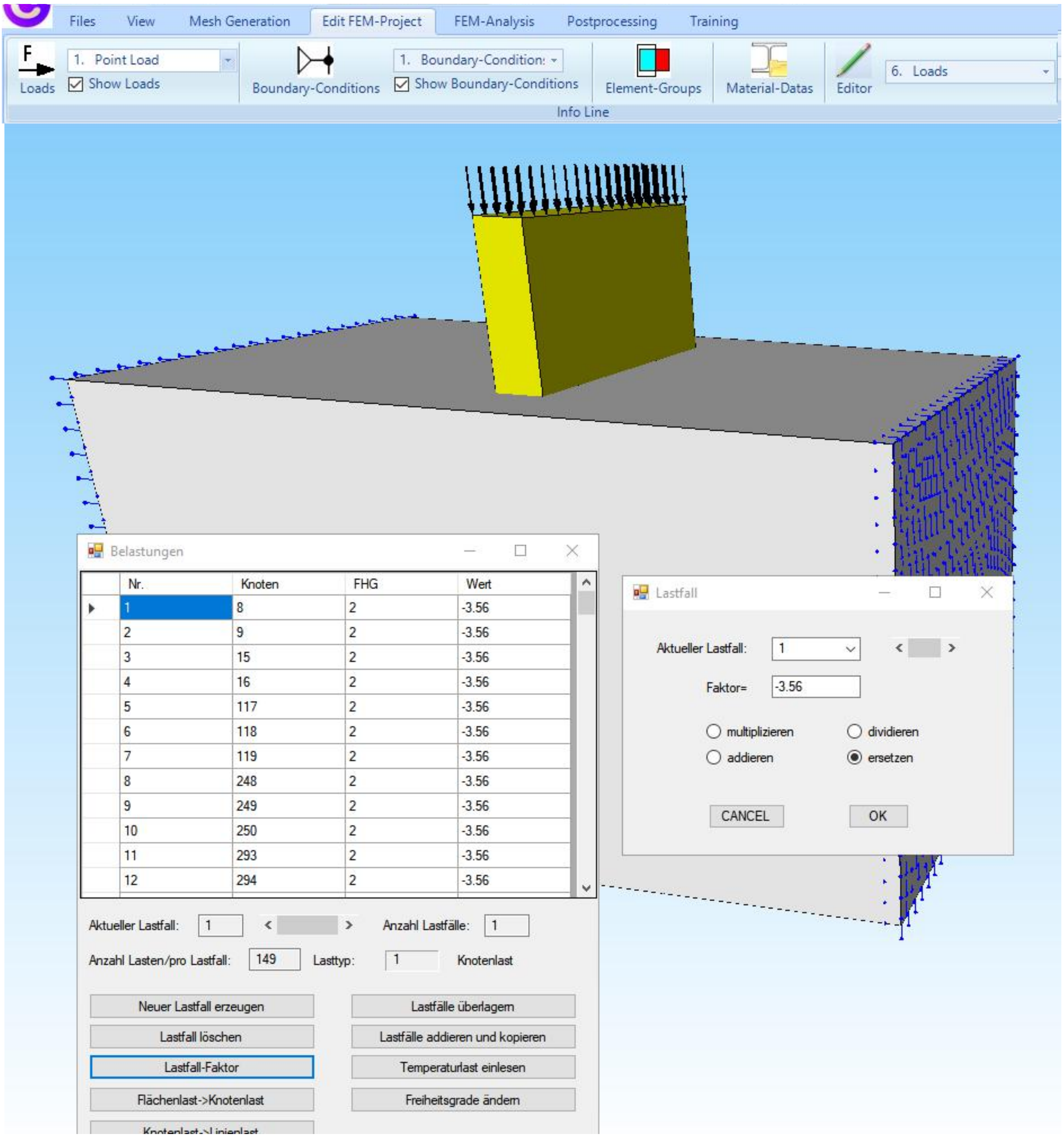
This is followed by an FEM Merge of the Steel Angle with Register “File” and menu “FEM-Merge” as well as a Node Check with Register “Mesh Generation” and menu “Check Node Numbering” to delete the 2202 node overlay.





Create a Point Load and Boundary Conditions

The Steel Angle is loaded with -500 N in Y Direction. Select the Register "Edit FEM-Project" and menu "Point Load" and load the upper surface with -1 N. Then enter a load value of -3.56 N with "Editor" which is calculated from 500/149.



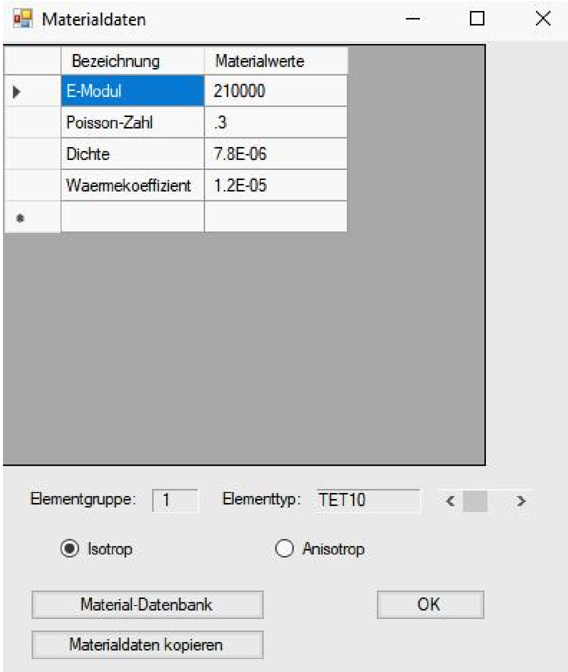
The Plastic Block is fixed clamped on the left and right side. Select "Boundary Conditions" to create the BCs.

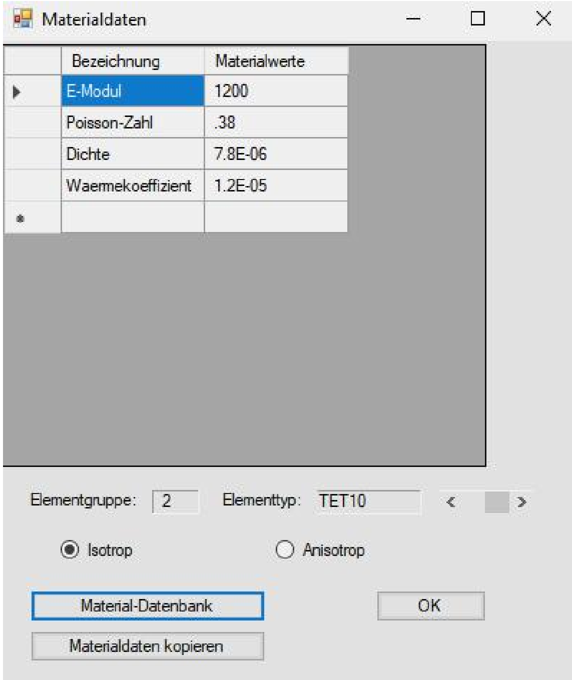
Enter the Material Datas

Select Register "Edit FEM project" and menu "Material-Datas" and enter the following material datas:

Element Group 1 - Steel Angle: E-Modulus = 210,000 MPa, Poisson's Value = 0.3

Element Group 2 - Plastic Block: E-Modulus = 1,200 MPa, Poisson's Value= 0.38





FEM Analysis

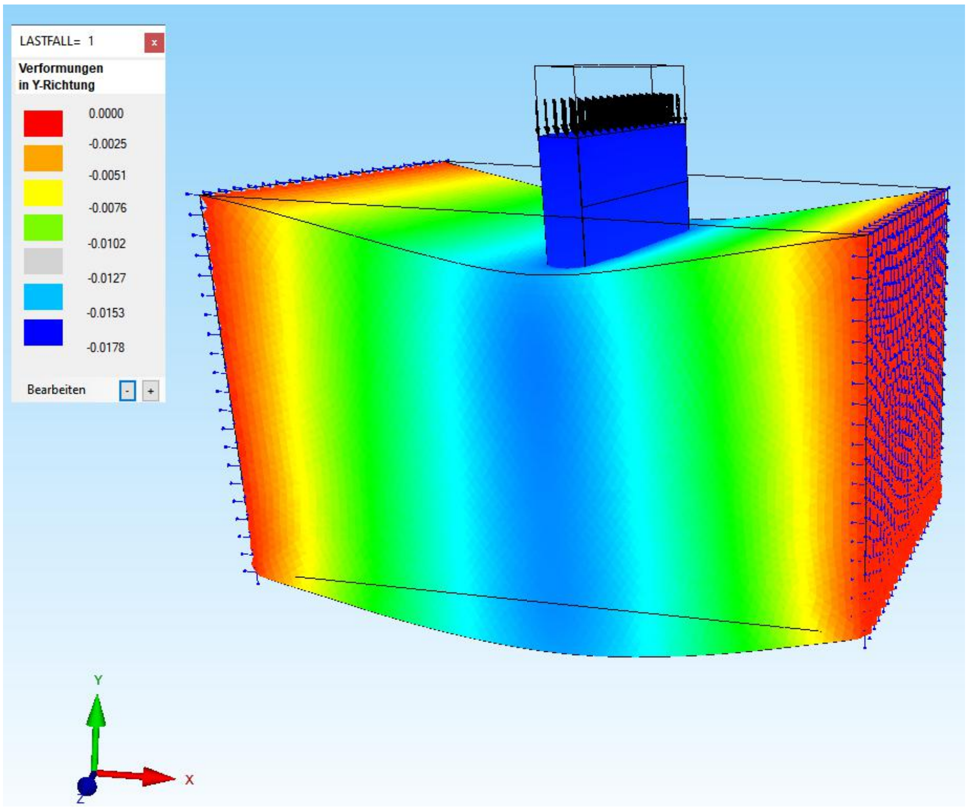
Before starting the FEM calculation, save the FEM model under the name “Steelangle-Plasticblock.FEM” and select Register “FEM Analysis” and start the Quick Solver to calculate the displacements and stresses.

[Download Video-MP4, FEM, INP, FRD Files for the Steel-Angel and Plastic-Box]

Postprocessing

With Register „Postprocessing“ and the Icon  the results of displacements and stresses can be evaluated.

Max. Displacements in Y Direction = - 0.0178 mm



Max. Normal Stresses SIGxx of the Steel Angle = 26.6 MPa
Min. Normal Stresses SIGxx of the Steel Angle = - 31 Mpa