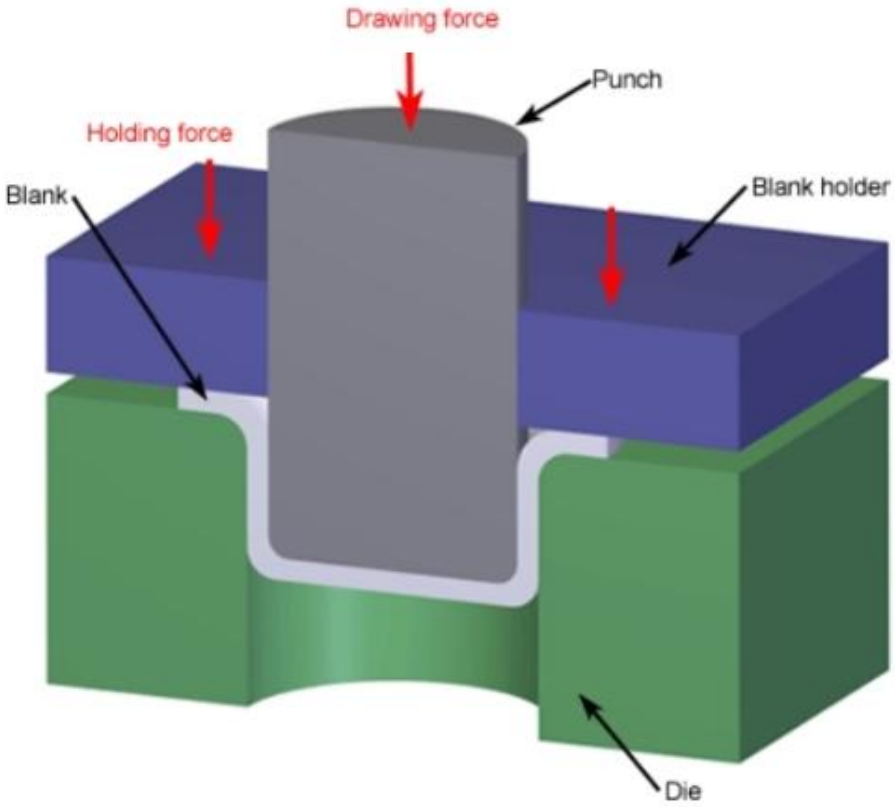
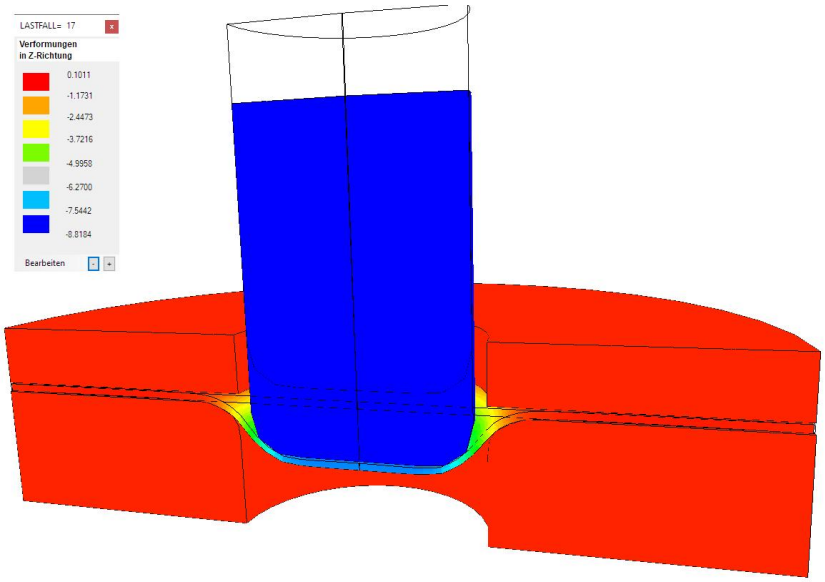
Part 43 - Deep-Drawing Simulation with the FEM System MEANS V14

A deep-drawing process is simulated using the FEM system MEANS V14 (www.fem-infos.com). Mesh generation and the evaluation of results are performed using the pre- and post-processor of MEANS V14, while the non-linear FEM analysis is carried out via the open-source solver CalculiX with an FEM-INP interface developed by HTA-Software.

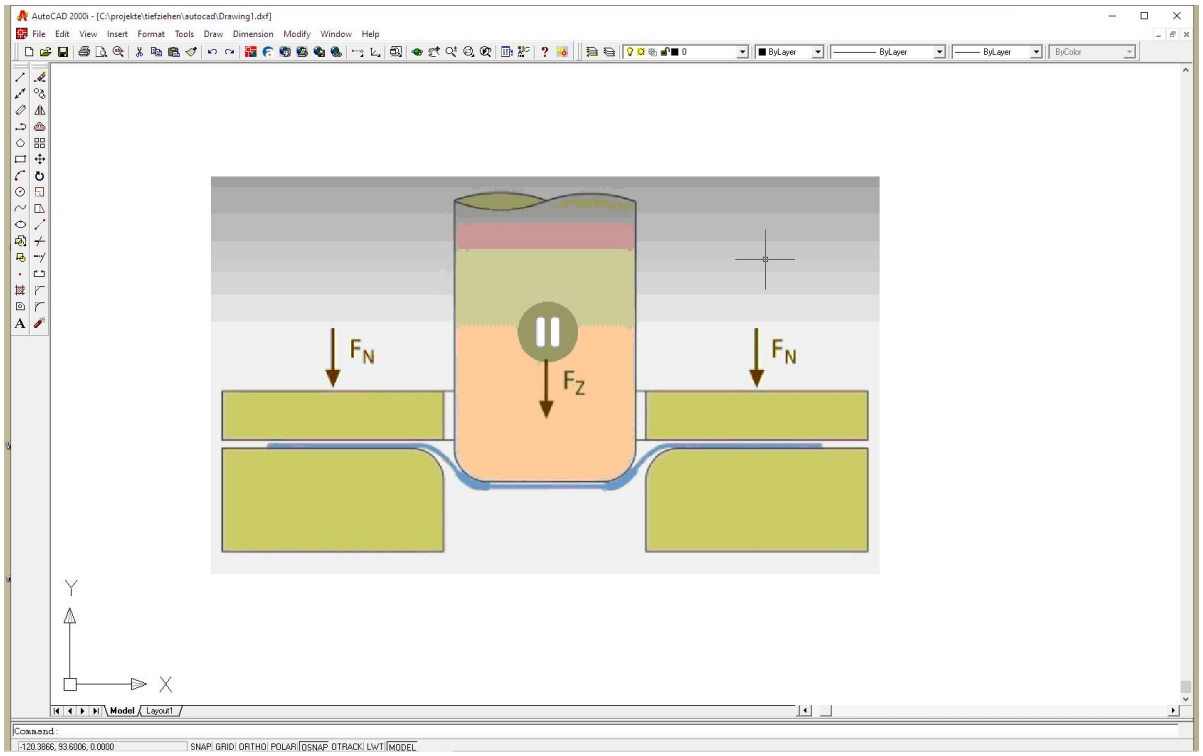


For non-linear contact analyses, the implicit solvers from Abaqus or Ansys are recommended. However, it is also possible to determine key parameters - such as radii, clearances or material properties - using a cost-effective FEM system like MEANS V14.

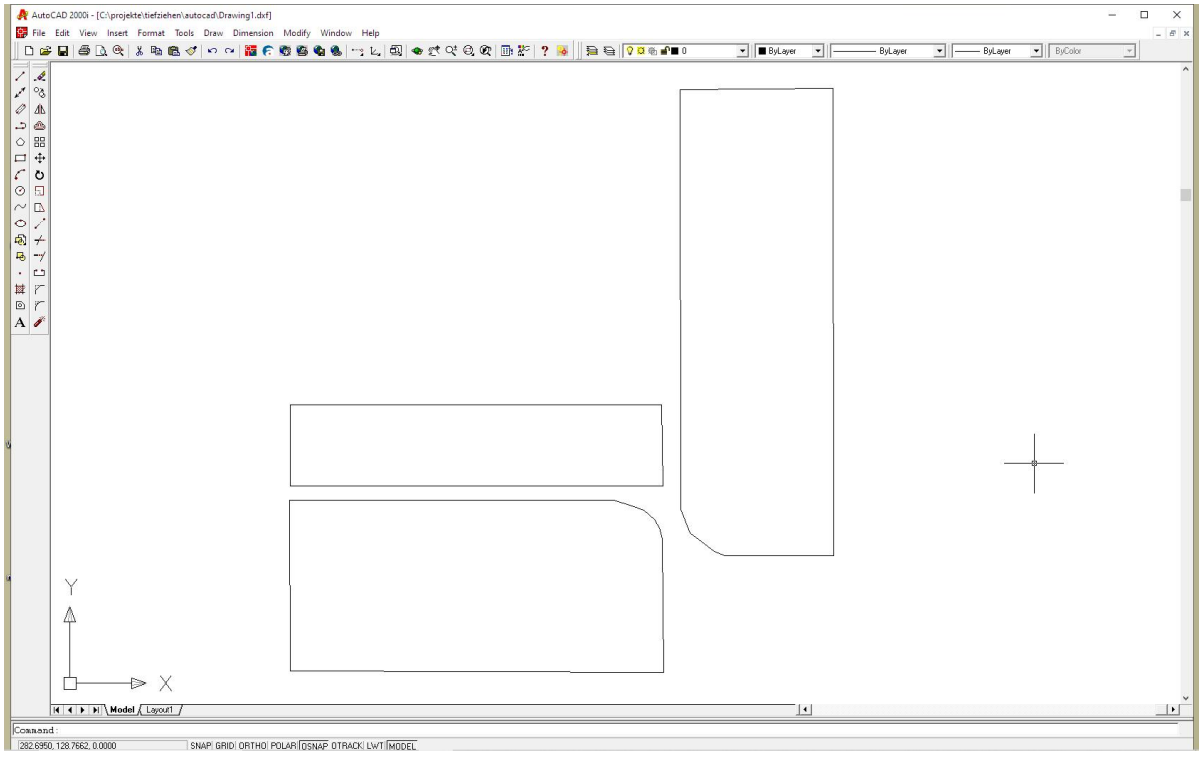


1. Creating a DXF Model

Launch AutoCAD and insert the deep-drawing image shown above into your AutoCAD drawing using the "Paste" command to serve as a template. Since the model is symmetrical, it is sufficient to trace only the left half of the contour using simple lines.



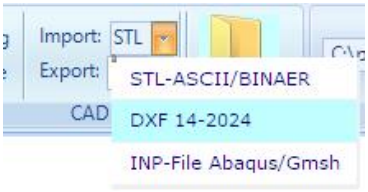
Finally, remove the image using the "Cut" command and save the contour as a DXF file for use with the MEANS DXF interface.



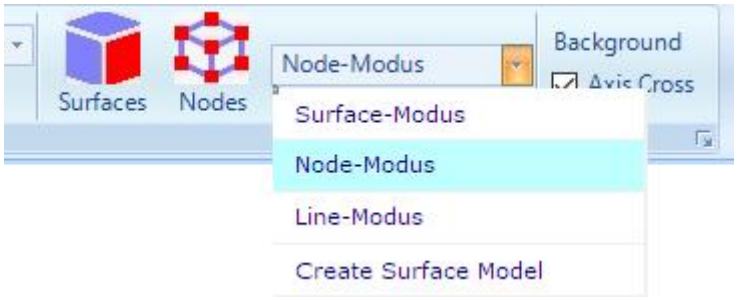
2. Importing the Sheet blank in Line Mode

Launch MEANS V14 using the Desktop icon  and follow these steps:

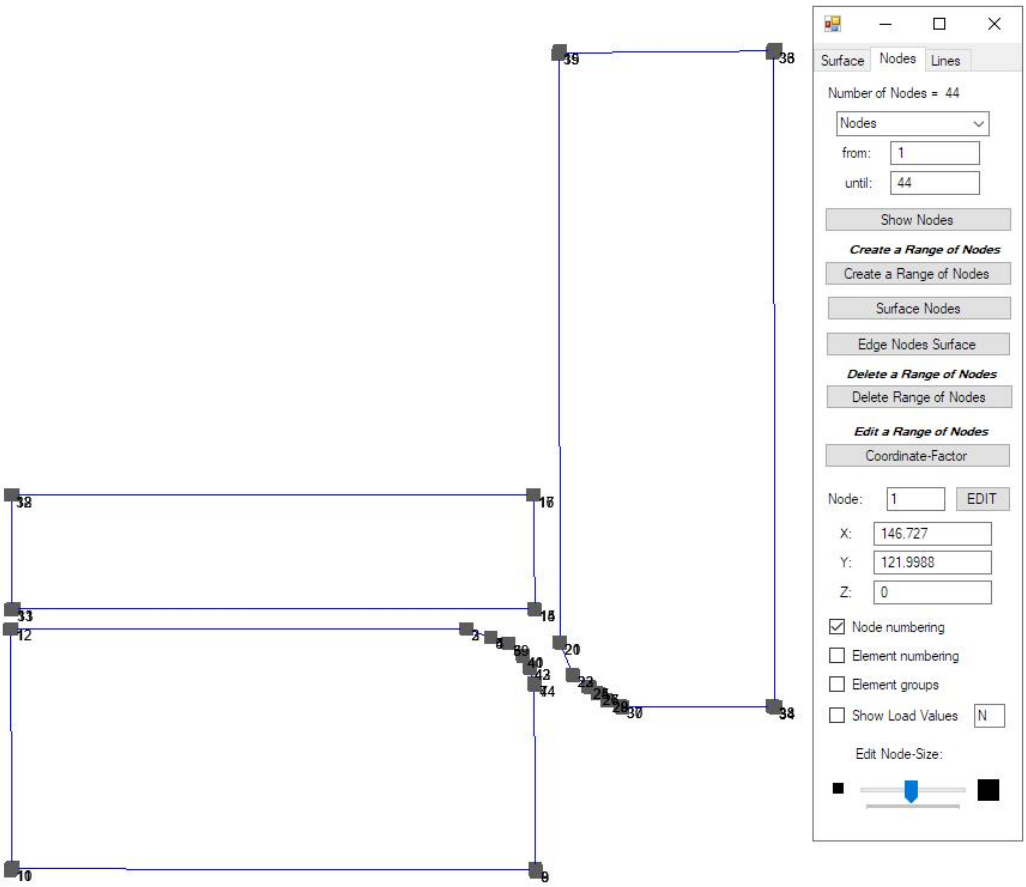
- Use the "New" tab to import the DXF file.



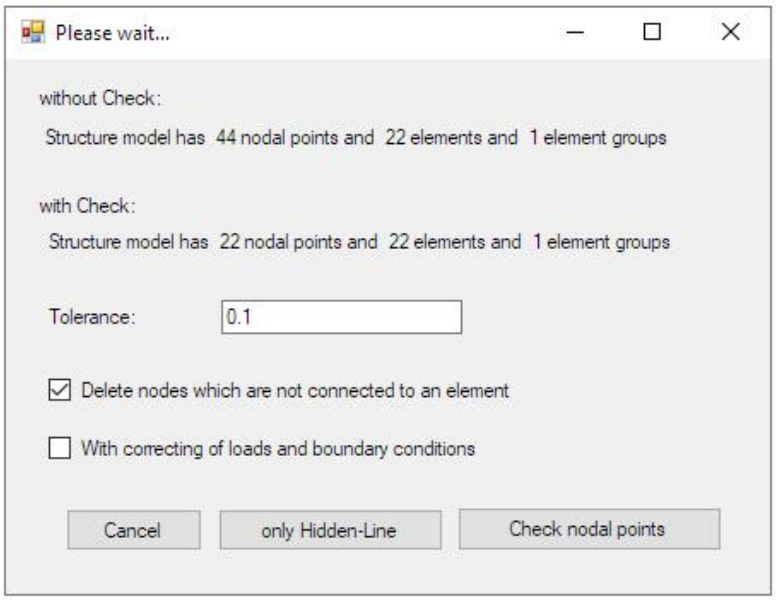
- Go to the "View" tab and select "Node Mode" to edit the nodes.



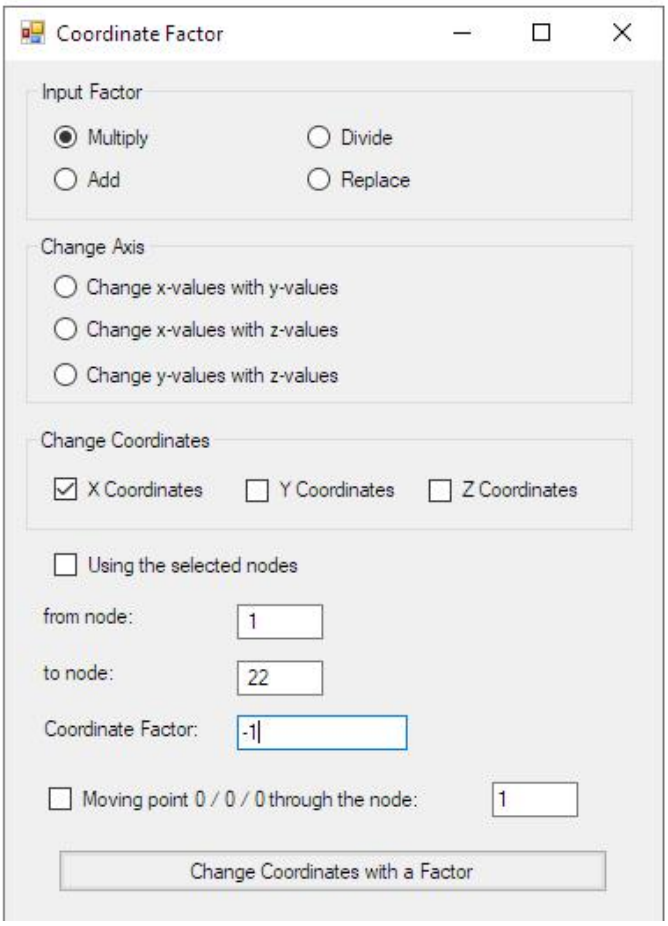
- Use the „Node-Size“ and „Number Nodes“ options to display all 44 nodes



- While in "Line Mode," use the "Check Nodes" function with a capture radius of 0.1 to remove any overlapping nodes, leaving a total of 22 nodes.

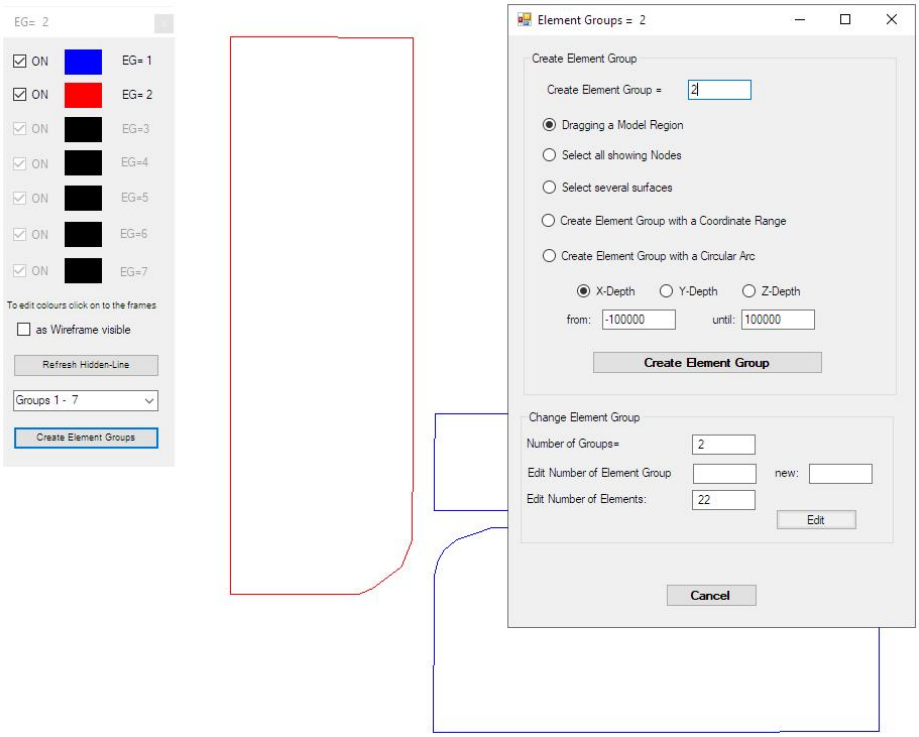


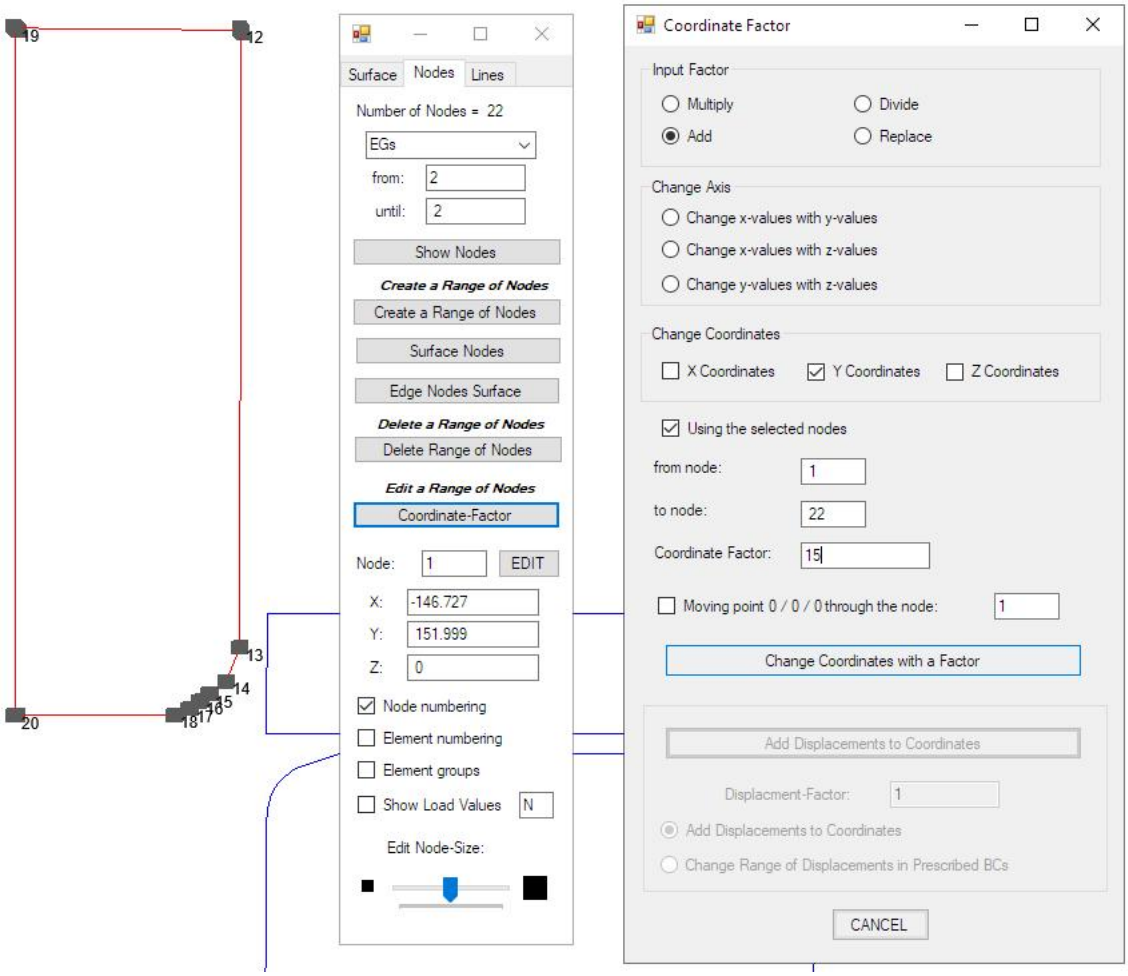
- Use the "Coordinate Factor" option to multiply the X-coordinates by -1



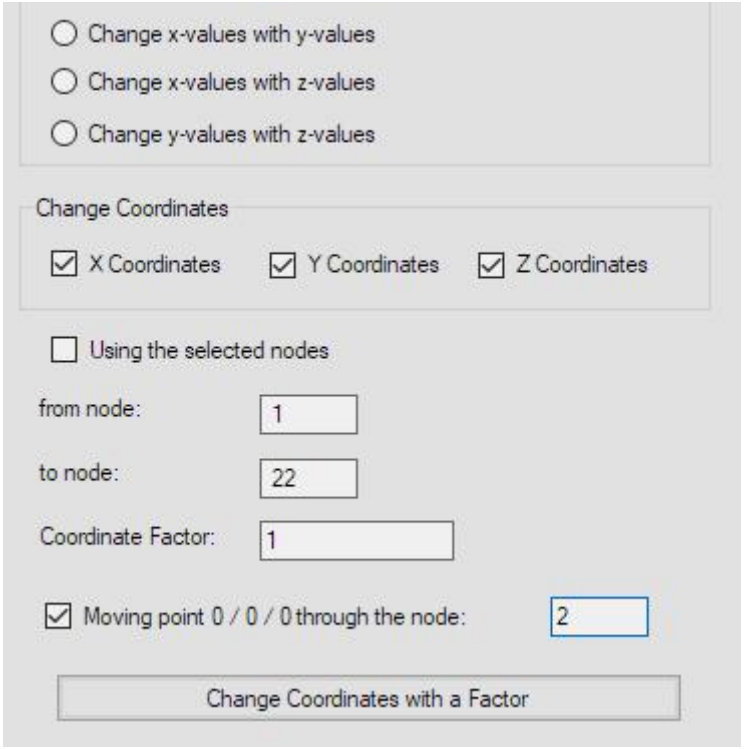
- To shift the punch 15 mm upward, select the "Edit FEM Project" tab. Use the "Element Groups" menu and drag a rectangle over the punch to assign it to Element Group 2 (EG 2); then, click the color box to set the color of EG 2 to red.

- To move the stamp upwards by 15 mm, select the "Edit FEM Project" tab and, using the "Element Groups" menu and the "Draw Rectangle" tool, convert the stamp to element group 2. Then, click the color box to assign the color red to element group 2.



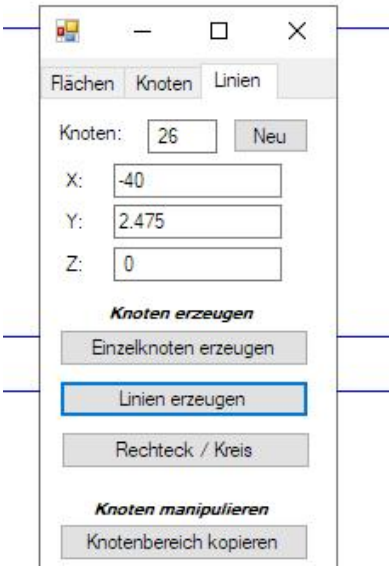


- In "Node Mode," display all nodes belonging to Element Group 2, then use the "Coordinate Factor" function to add "15" to the Y-coordinates of these nodes.
- Use the "Coordinate Factor" function to set the origin (zero point) at Node 2, ensuring the blank is positioned precisely between 0.3 to 2.175.



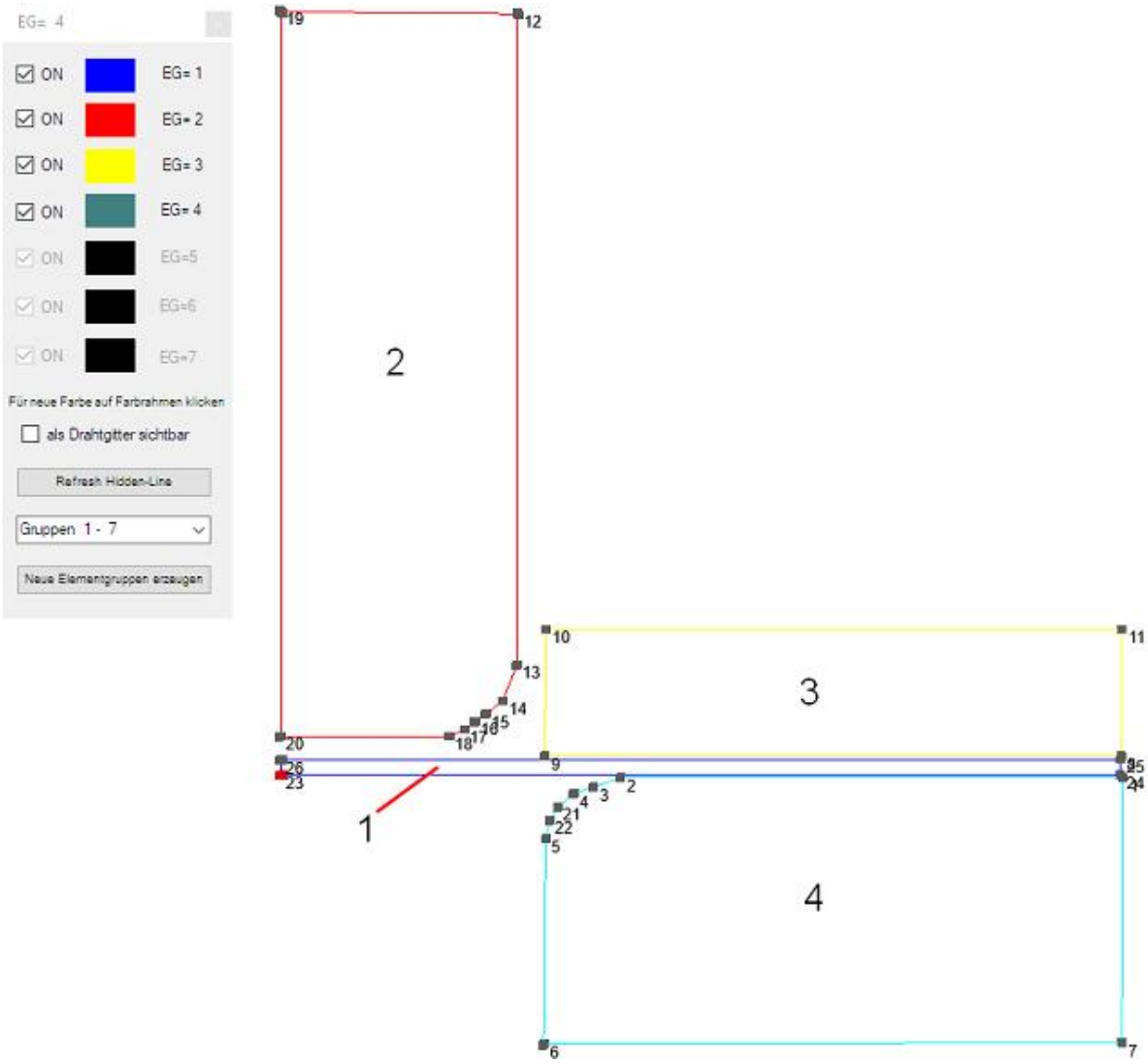


- In line mode, select "New" to display a new node (node 23), enter the coordinates -40/0.3, and select "Create single node." Repeat this process for node 24 (59/0.3), node 25 (59/2.175), and node 26 (-40/2.175)



- Then, use the "Create lines" menu to connect the four nodes: 23–24, 24–25, 25–26, and 26–23

- Create all four element groups by "drawing a rectangle" so that the entire structure can be generated in one step.

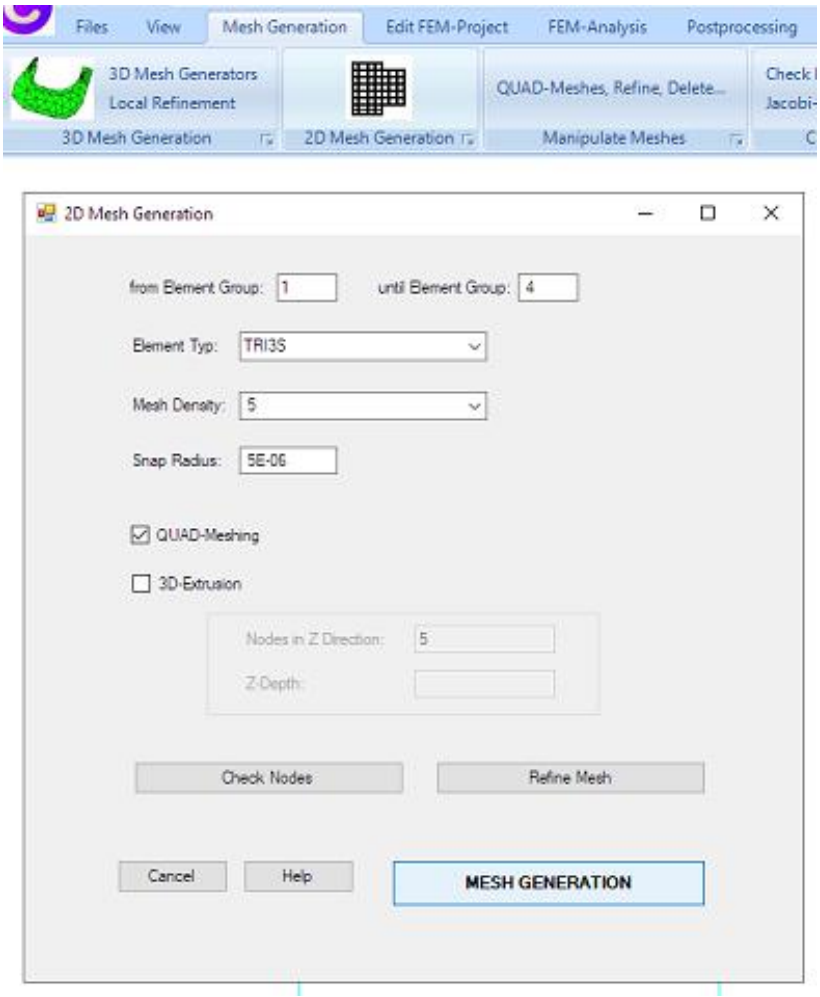


- Use the “Coordinate Factor” menu to set the origin at node 23, ensuring that the axis of rotation lies at X=0.

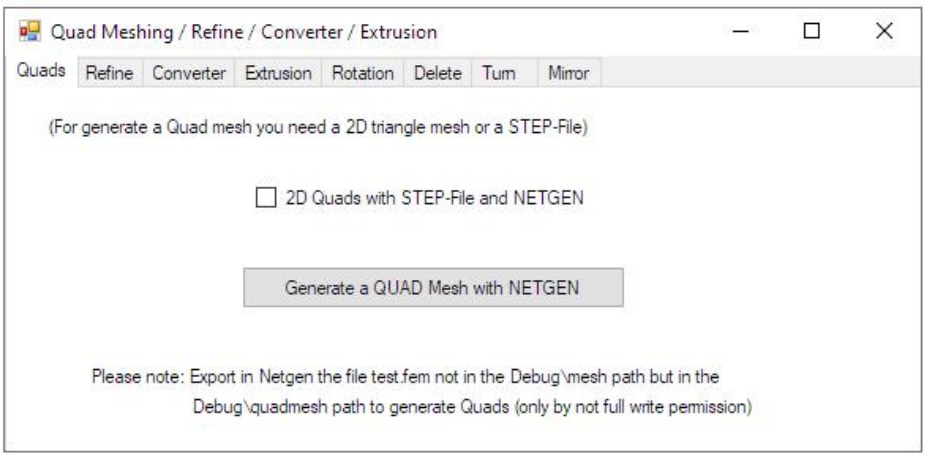
- Save the line model to the hard drive under a name of your choice.

3. Triangular and Quadrilateral Meshing

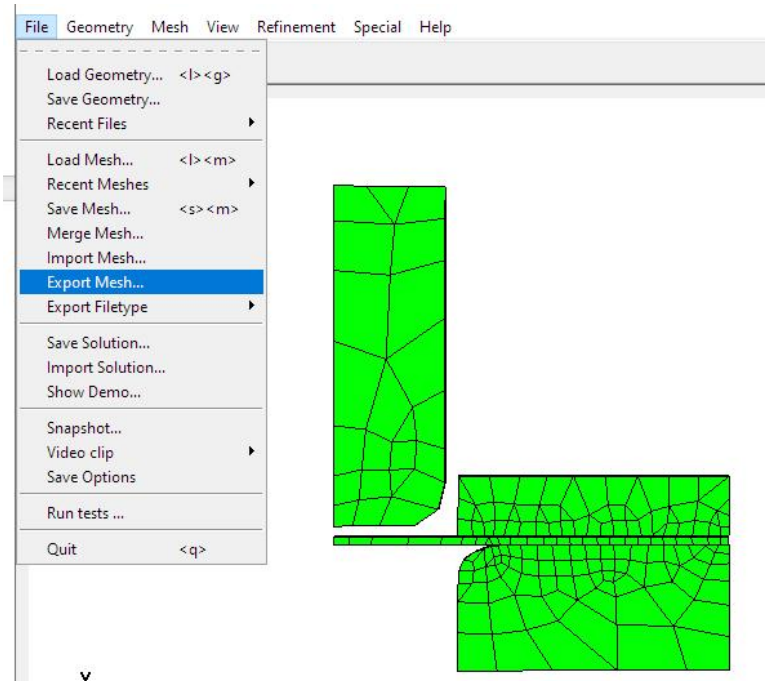
- A triangular mesh is created first, serving as a template for generating a quadrilateral mesh using NETGEN.
- Select the "Mesh Generation" and "2D Mesh Generator" tabs to mesh element groups 1 through 4 with an element density of "5," and ensure the "QUAD-Quadrilateral Mesh" option is checked.



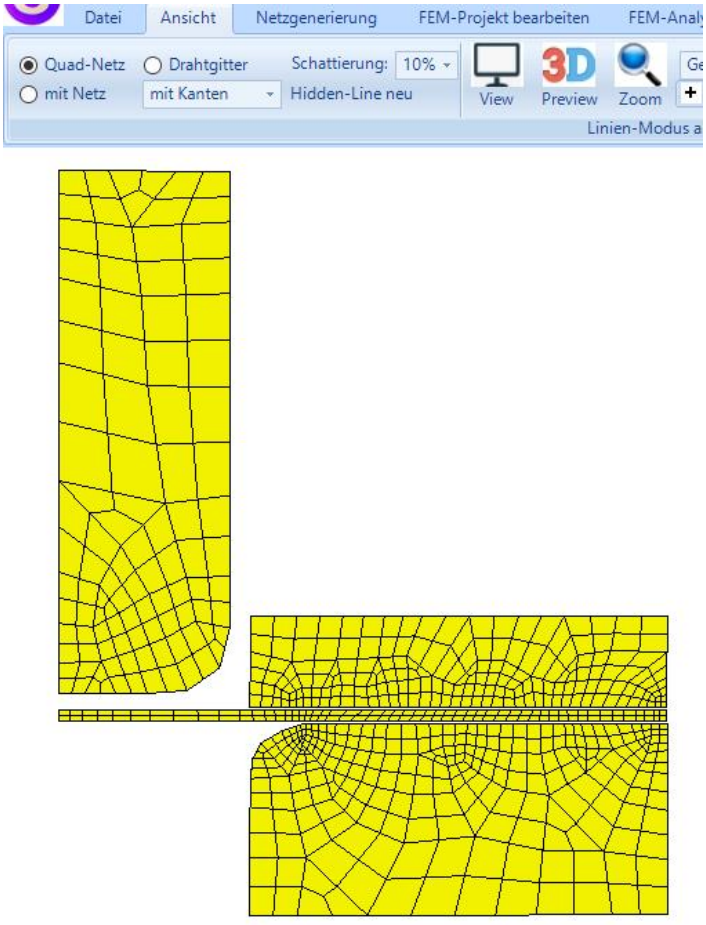
- After the triangular meshing process, another dialog box will appear; select the option "Generate QUAD mesh with NETGEN" from the menu.



- In NETGEN, the triangular mesh is displayed in blue; use "Generate Mesh" to create a QUAD mesh, and then use "Export" to save the mesh as "test.fem" in the `debug/quadmesh` directory. Do not perform any mesh refinement, as this may cause errors when importing the file into MEANS.



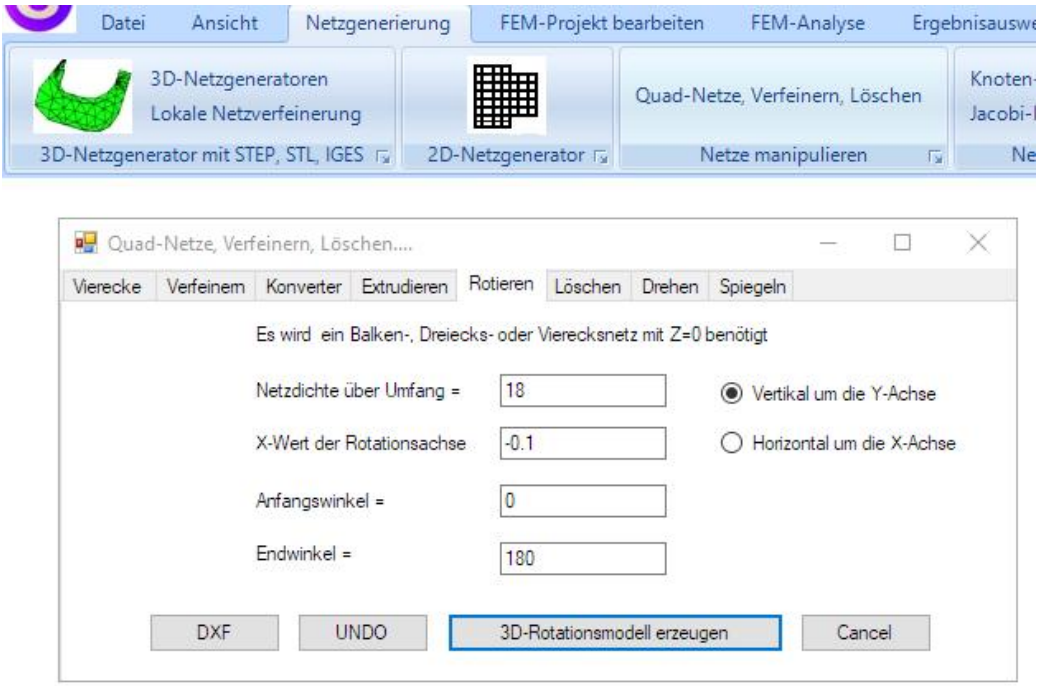
- Once the model has been verified, the QUAD mesh will be displayed in MEANS.



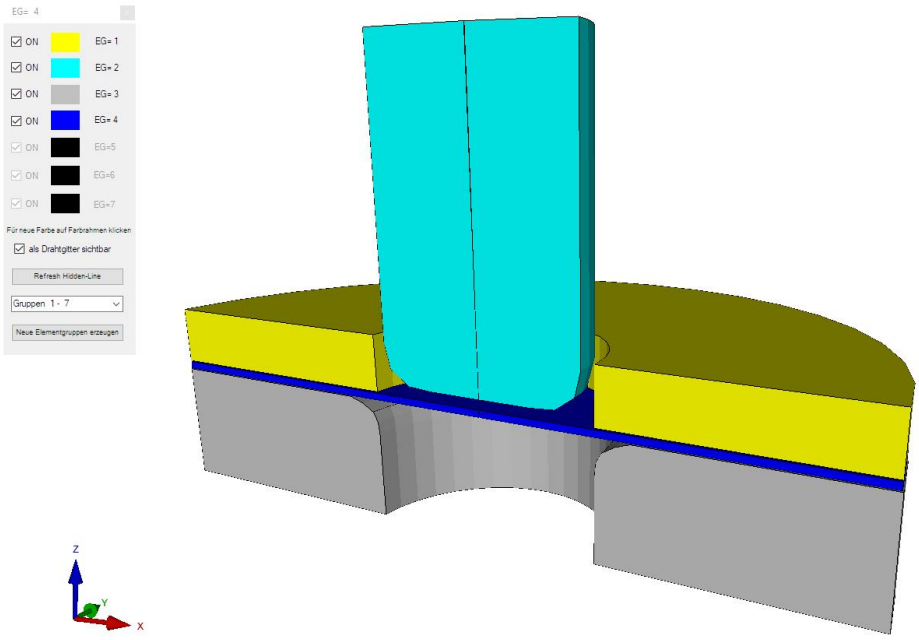
4. Hexahedral model with 180-degree rotation

- Select the "Mesh Generation" tab, the "Quad Meshes, Refine, Delete" menus, and the "Rotation" tab within the dialog box; then, create a revolved model with a mesh density of "18" and an end angle of "180" degrees (for a half-model).

- Enter an X-value of "-0.1" for the axis of rotation to prevent the generation of unstable pentahedral elements during the rotation process..



- Following a model review and a Y-Z axis swap with a coordinate factor, a hexahedral model is obtained, consisting of 11,736 elements, 15,314 nodes, and 4 element groups.



5. Defining Boundary Conditions

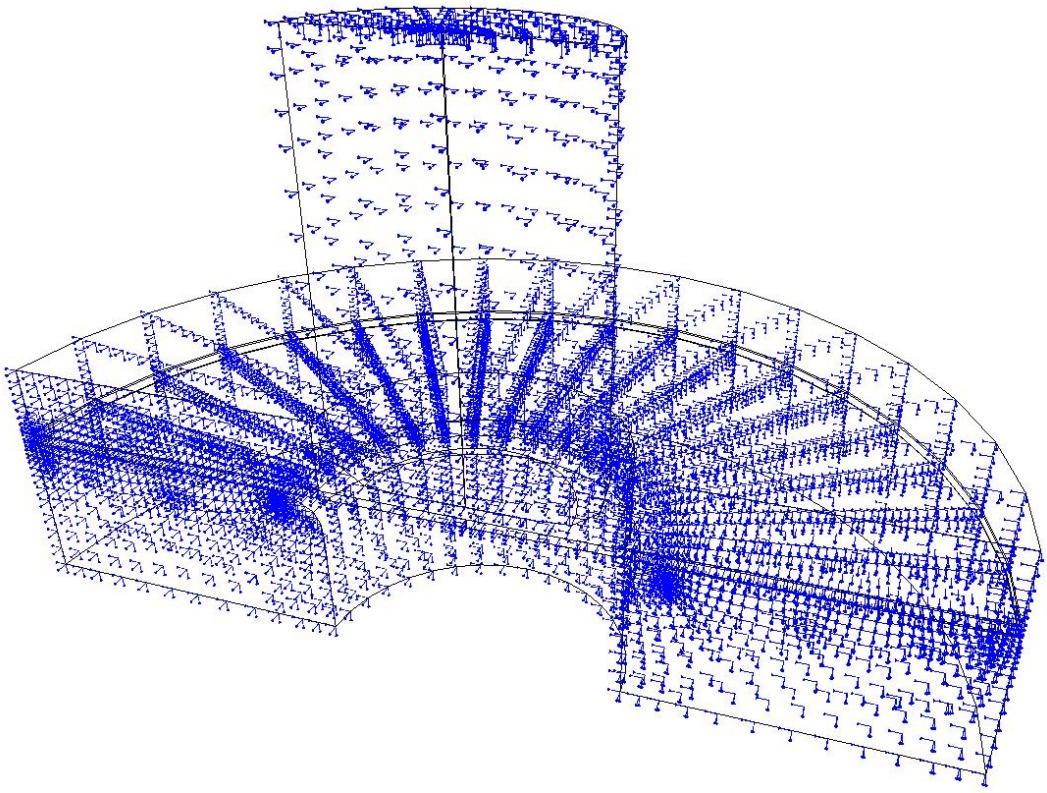
- Create element groups using the surface model, ensuring that the sheet metal blank (forming plate) is assigned to the final element group, EG 4.

Element Group	Designation	Boundary Conditions and Restraints
EG 1	Blank holder	Fixed in X, Y, and Z directions
EG 2	Punch	Fixed in X and Y directions; prescribed displacement of -80 mm in the Z direction (at the top)
EG 3	Die	Fixed in X, Y, and Z directions
EG 4	Sheet metal blank	Fixed in X and Y directions at the small central surface; front surface fixed in Y direction (due to symmetry)

- Display EG 1 and EG 3; apply fixed restraints in the X, Y, and Z directions to all surfaces of the blank holder and the die.

- Display EG 2; apply fixed restraints to the punch in the X and Y directions, and specify a displacement of -80 mm in the Z direction (at the top).

- Display EG 4; apply fixed restraints at the center in the X and Y directions, and fix the front surface in the Y direction (due to symmetry).



6. Creating Contact Surfaces

Point loads are treated as contact nodes, and surface loads as contact surfaces, provided that "Contact Analysis" is enabled in the material non-linearity dialog box. Under these conditions, the following three load cases are defined:

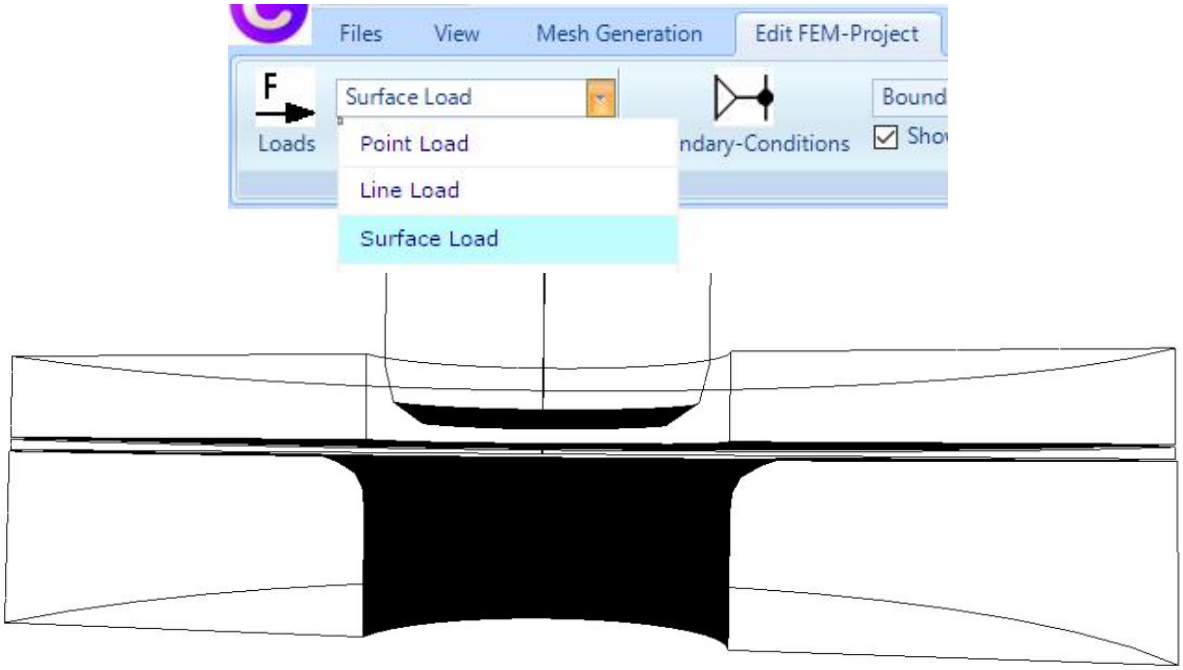
Load Case 1
Point or surface load representing the external load.

Load Case 2
Point loads (for 2D elements) involving MASTER contact nodes;
Surface loads (for 3D elements) involving MASTER contact surfaces.

Load Case 3
Point loads (for 2D elements) involving SLAVE contact nodes;
Surface loads (for 3D elements) involving SLAVE contact surfaces.

Follow these steps to define the contact surfaces:

- In the "Edit FEM Project" tab, define Load Case 1 by applying a small point load; note that the external load is already defined via the specified punch displacement of -80 mm.
- Define Load Case 2, which includes the MASTER contact surfaces, by applying surface loads to the lower blank holder plate, the upper die plate, and the underside of the punch.

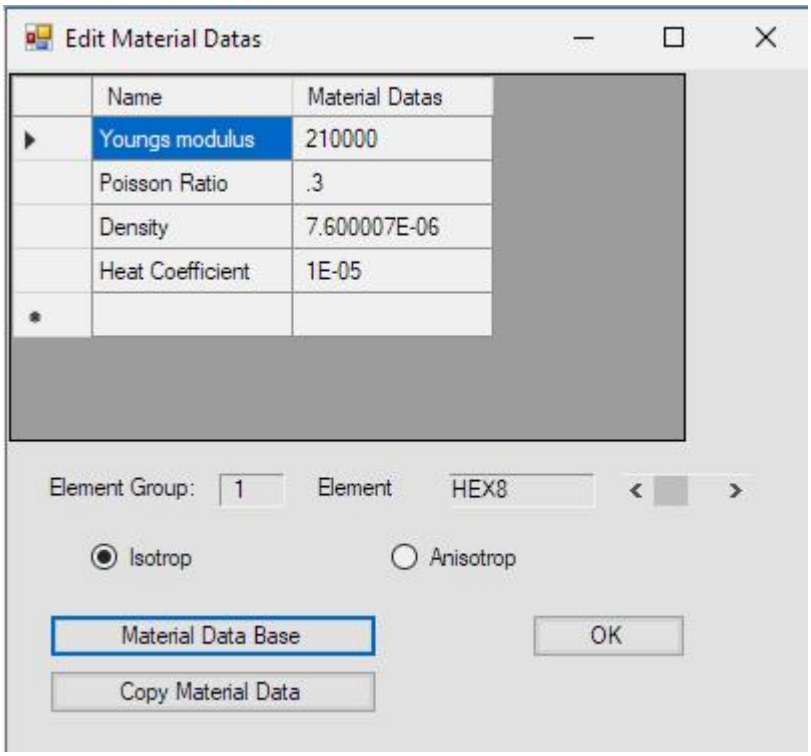


- Load case 3, involving the SLAVE contact surfaces, is generated by applying a surface load to the top and bottom sides of the sheet metal blank.

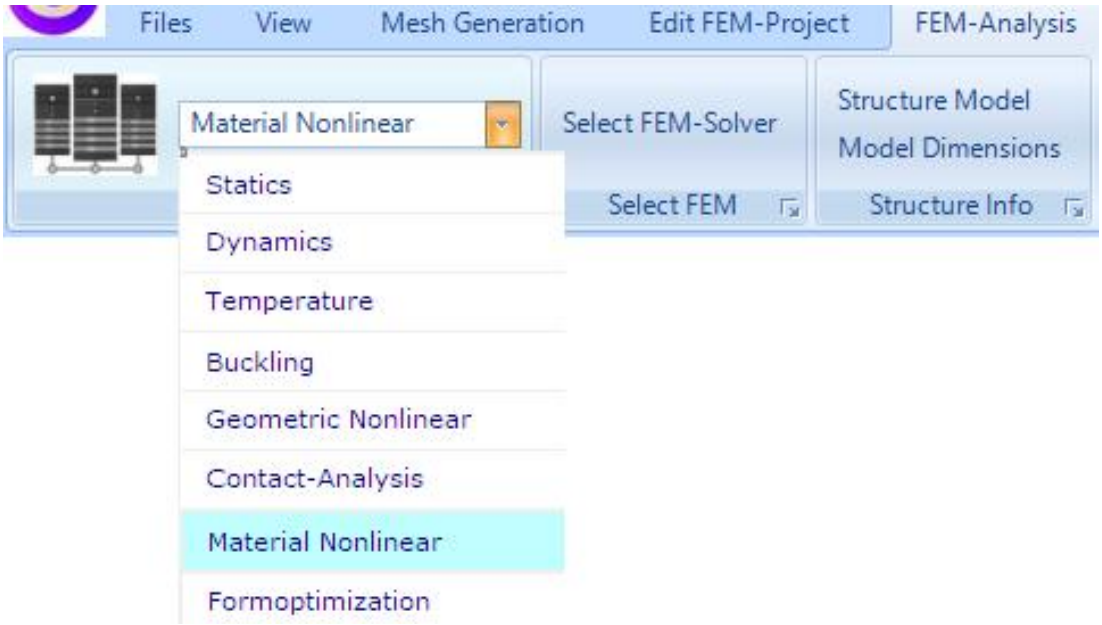


7. Material Nonlinearity with Friction

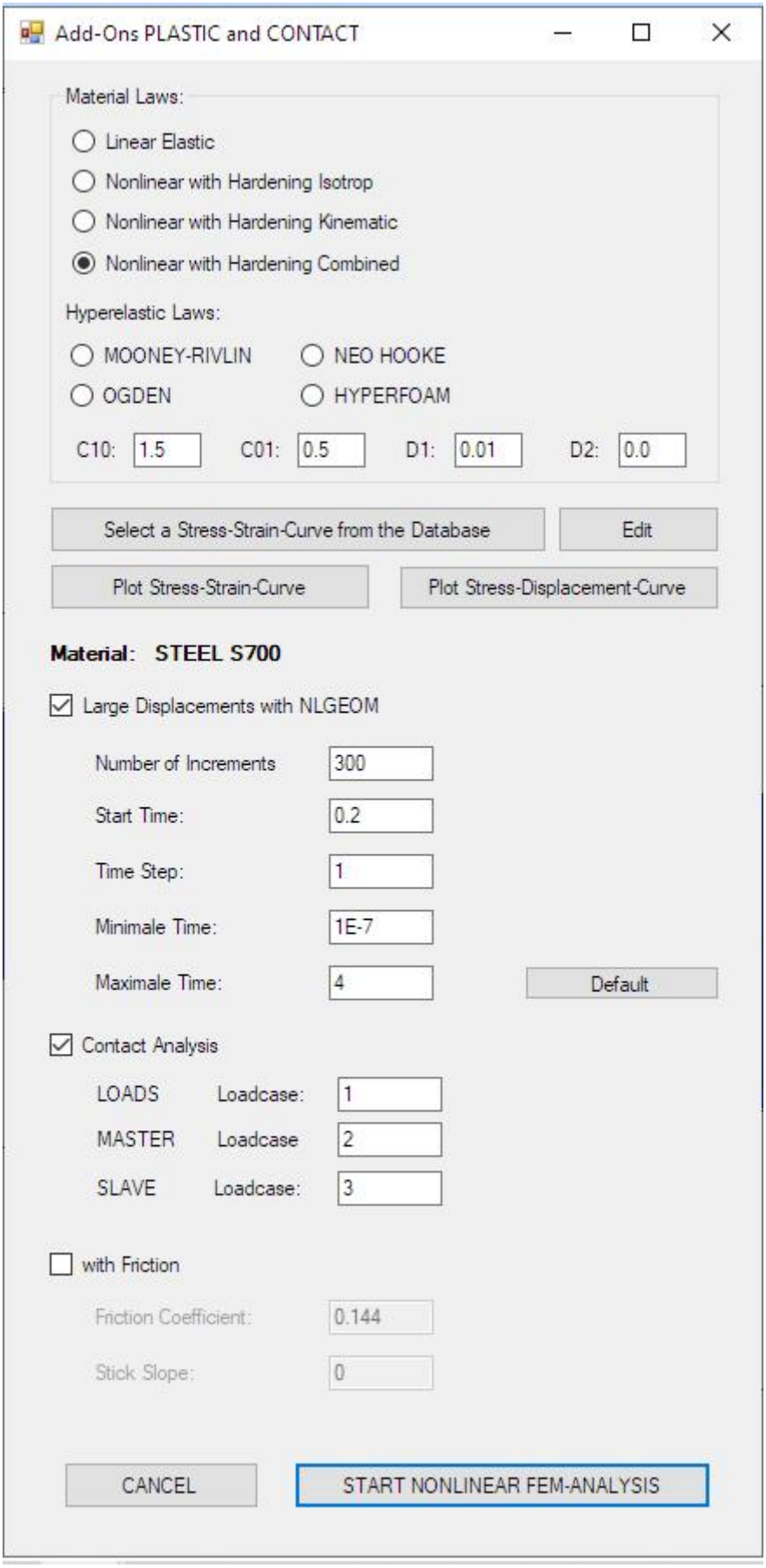
- First, select "Edit FEM Project" and "Material Data," then enter a Young's modulus of 210,000 MPa and a Poisson's ratio of 0.3 for all element groups.



- Next, select the "FEM Analysis" tab and choose "Material Nonlinearity."

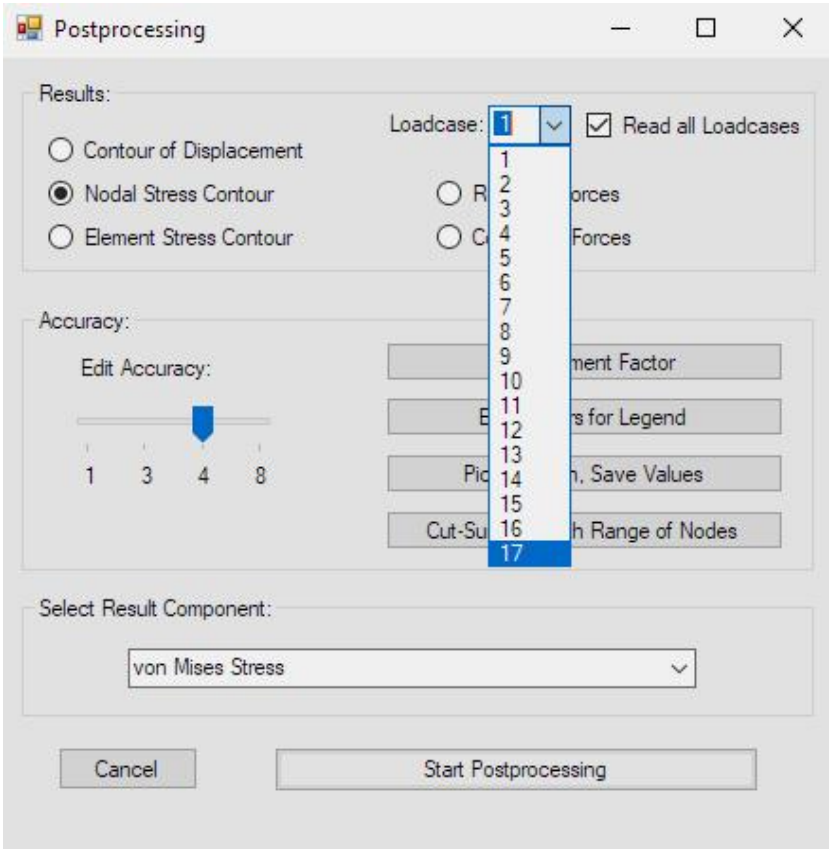


- Select “Nonlinear with Hardening Isotropic” or “Nonlinear with Hardening Combined,” choose the “S700” stress-strain curve, accept the default settings, and enter a friction coefficient of 0.1 for sliding friction between the contact

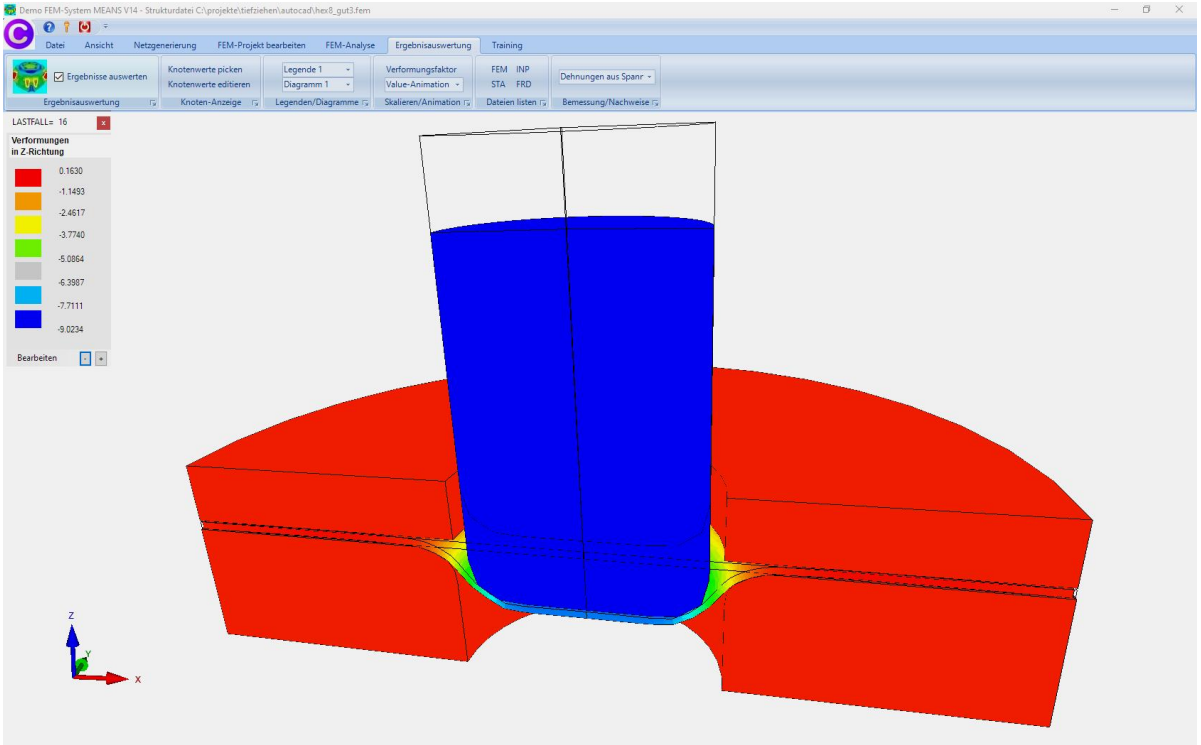

Add-Ons PLASTIC and CONTACT
Material Laws:
Linear Elastic
Nonlinear with Hardening Isotrop
Nonlinear with Hardening Kinematic
Nonlinear with Hardening Combined
Hyperelastic Laws:
MOONEY-RIVLIN
NEO HOOKE
OGDEN
HYPERFOAM
C10: 1.5
C01: 0.5
D1: 0.01
D2: 0.0
Select a Stress-Strain-Curve from the Database
Edit
Plot Stress-Strain-Curve
Plot Stress-Displacement-Curve
Material: STEEL S700
Large Displacements with NLGEOM
Number of Increments 300
Start Time: 0.2
Time Step: 1
Minimale Time: 1E-7
Maximale Time: 4
Default
Contact Analysis
LOADS Loadcase: 1
MASTER Loadcase 2
SLAVE Loadcase: 3
with Friction
Friction Coefficient: 0.144
Stick Slope: 0
CANCEL
START NONLINEAR FEM-ANALYSIS

8. Postprocessing

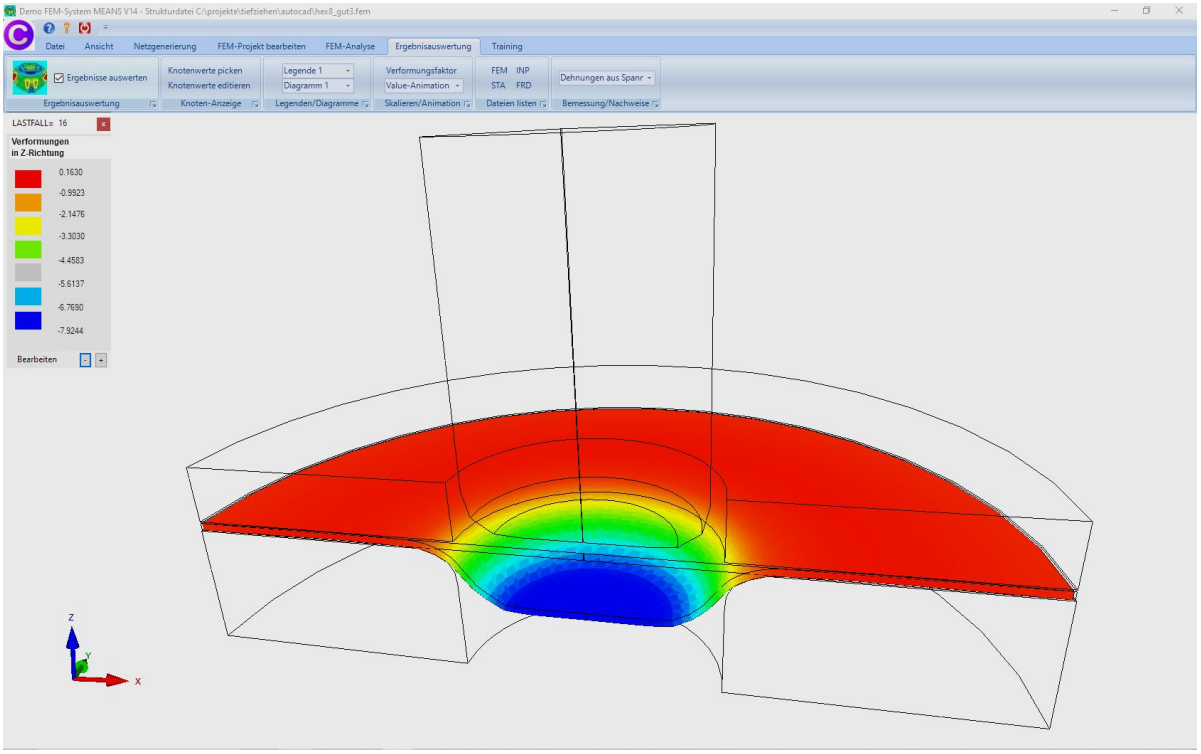
Select the "Postprocessing" tab and choose the final load case, 17, to display the deformation and stress distribution.



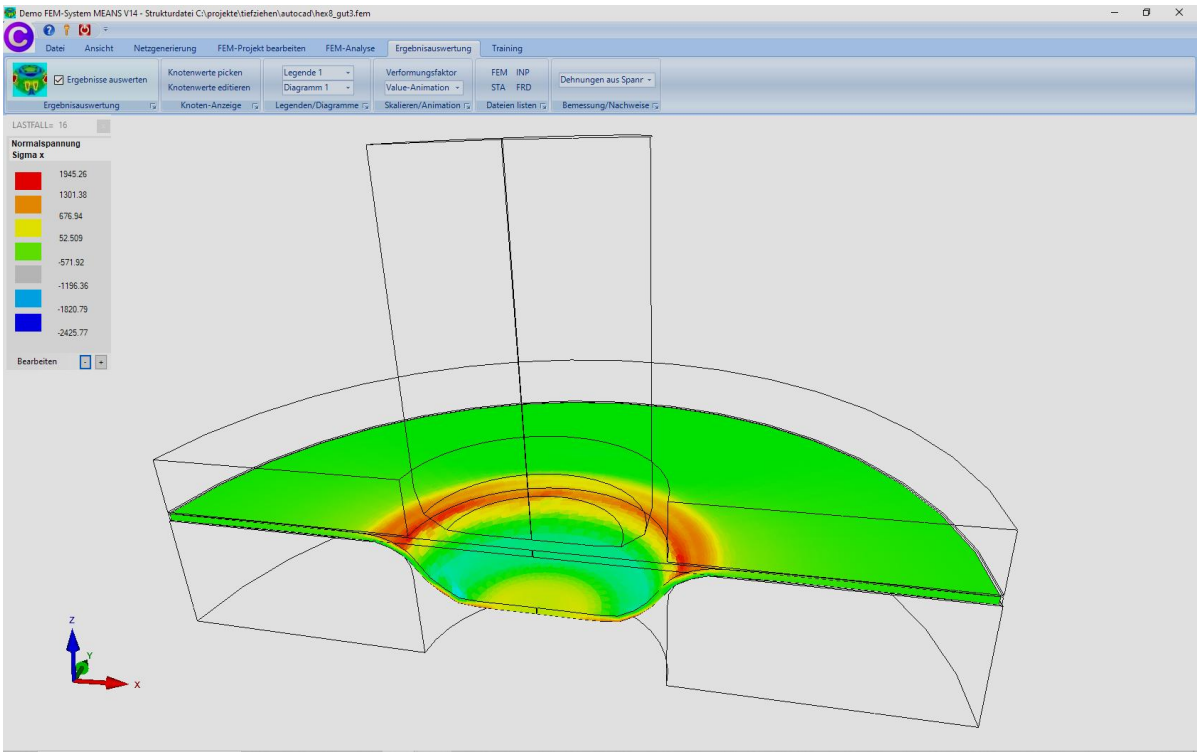
Max. Displacements in Z Direction on the Punch = - 9.02 mm



Max. Displacement in Z Direction on the blank = - 7.92 mm

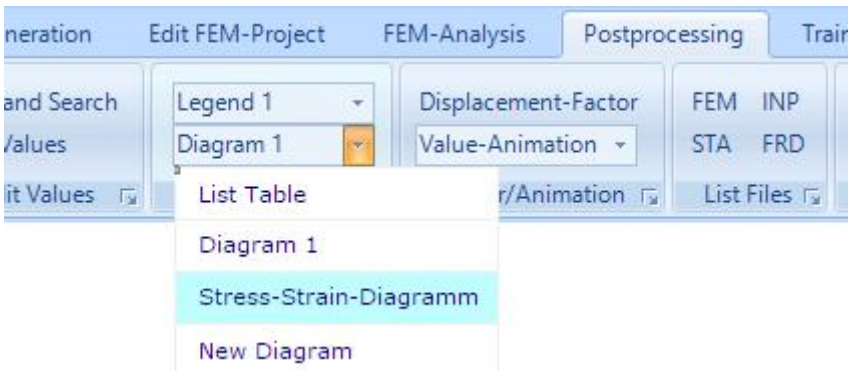


Max. tensile stress = 1,945 MPa and max. compressive stress = -2,425 MPa



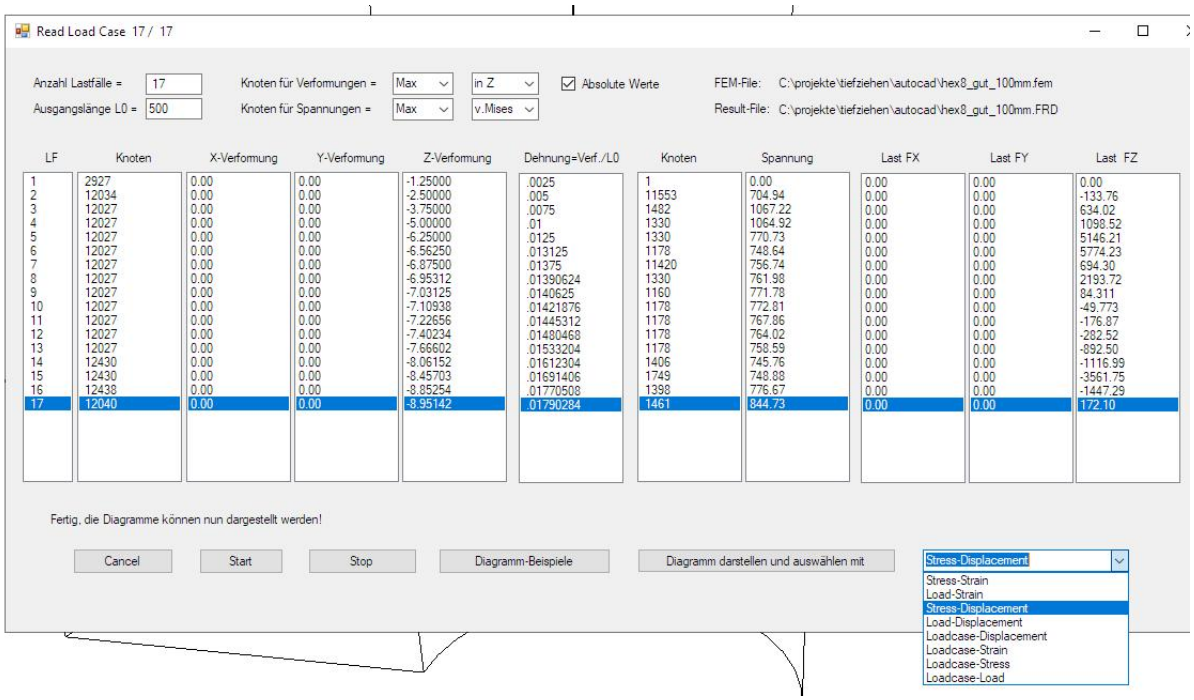
9. Diagram Evaluations

Select the "Result Evaluation" tab and the "Stress-Strain Diagram" option to clearly evaluate the result progressions for load cases 1–17 in the form of diagrams.



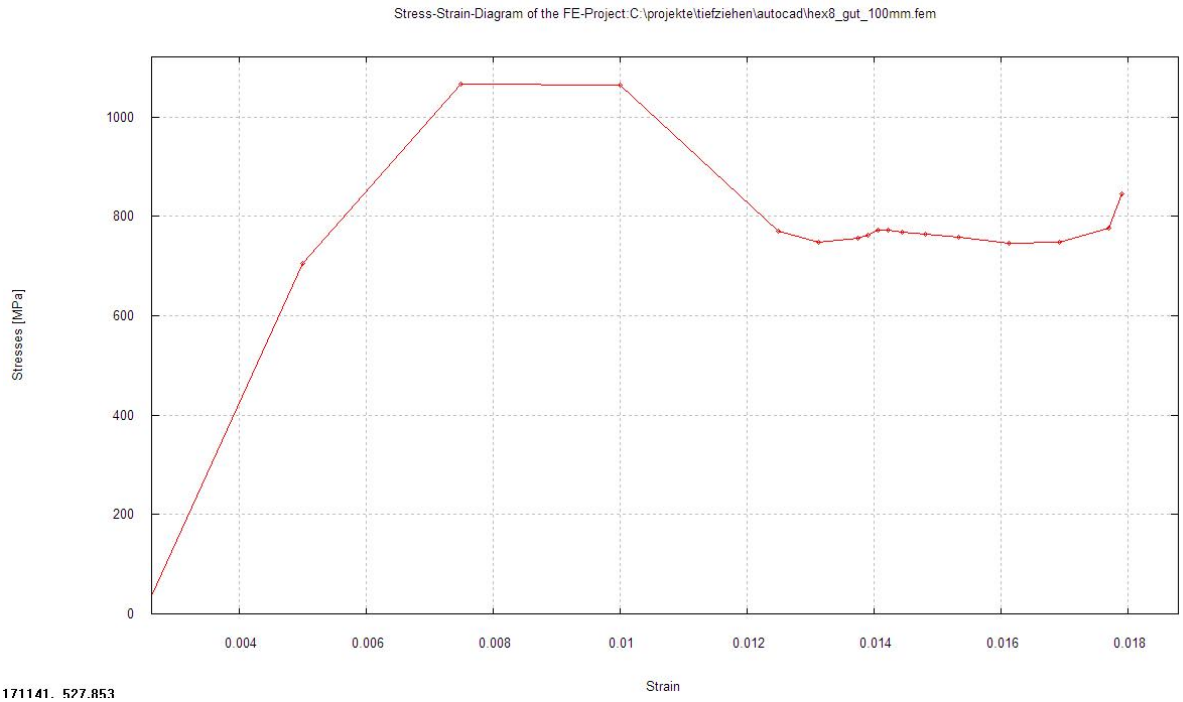
The following diagrams can be displayed:

- Stress-Strain
- Stress-Displacement
- Load-Displacement
- Loadcase Displacement
- Loadcase Strain
- Loadcase Stress
- Loadcase Load



- The material nonlinearity is clearly evident; the von Mises stresses increase linearly up to the yield strength, then flatten out and cease to rise - the deformations, on the other hand, become increasingly larger and more plastic until failure.

Stress-Strain-Diagram



Stress-Displacement-Diagram

