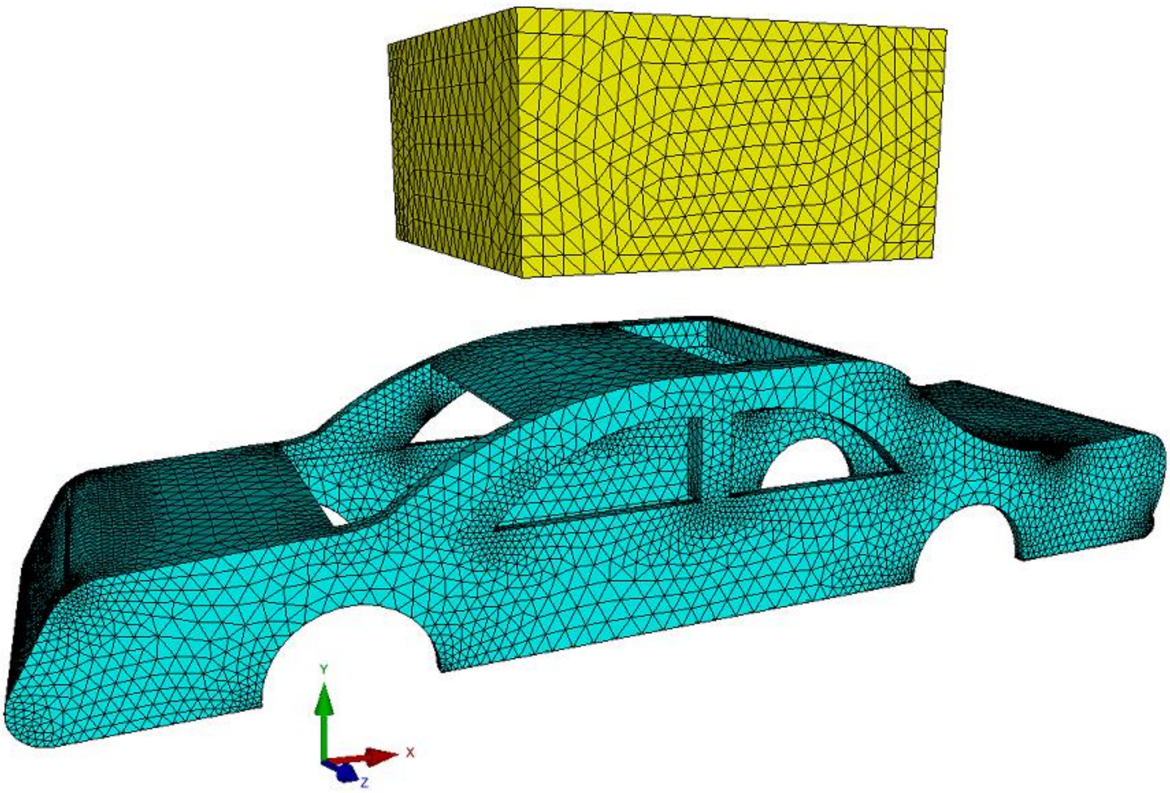
Part 37: Drop Test FEM Simulation of a 10-Ton Rock falls onto Car Roof

The deformations and stresses are to be simulated using the Finite Element System MEANS V13 (www.femcad.de) when a 10-ton rock falls onto a car roof.
The car body is made of lightweight aluminum, which is used in almost all electric cars today due to the additional battery weight of 100 kg.
Disadvantages, however, are the higher plastic deformations and the lower yield strength.

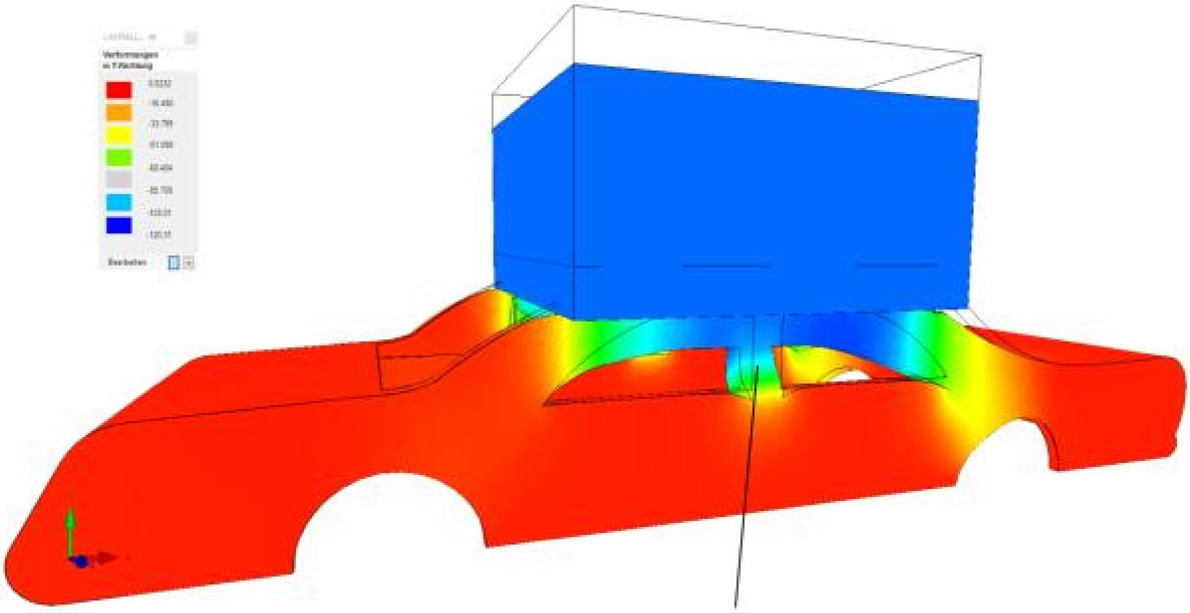
What are the:

- Impact force
- Impact speed
- Fall height of the rock
- Fall time of the rock

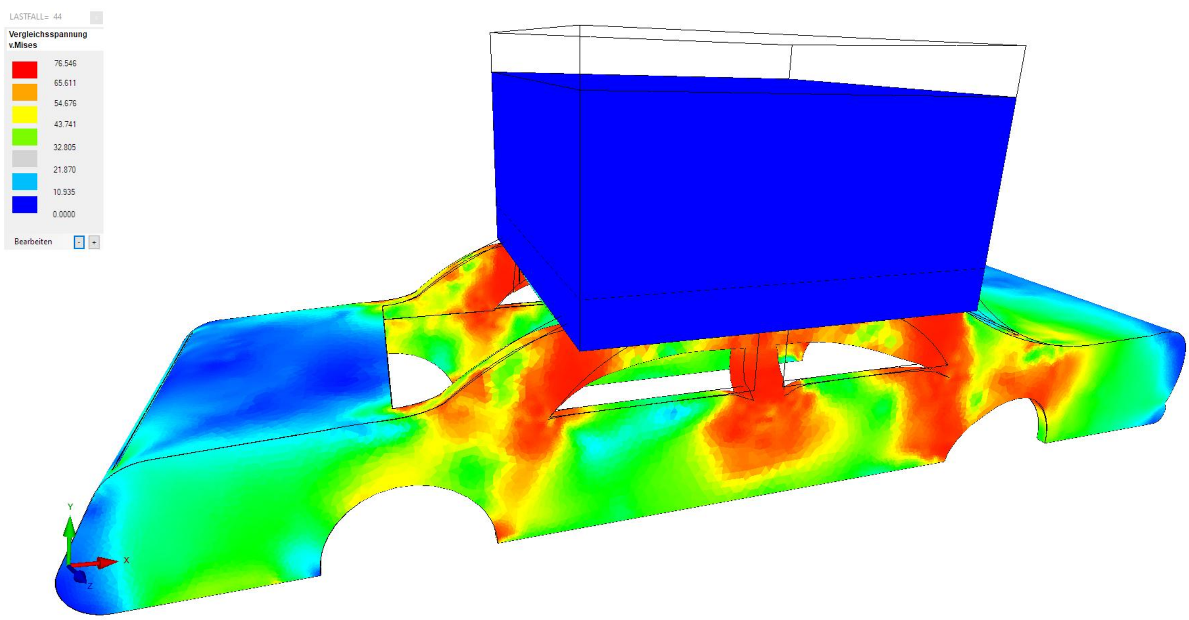
when the A-pillar has deformed by -90 mm?



FEM model of the car and FEM model of the rock for a nonlinear contact analysis



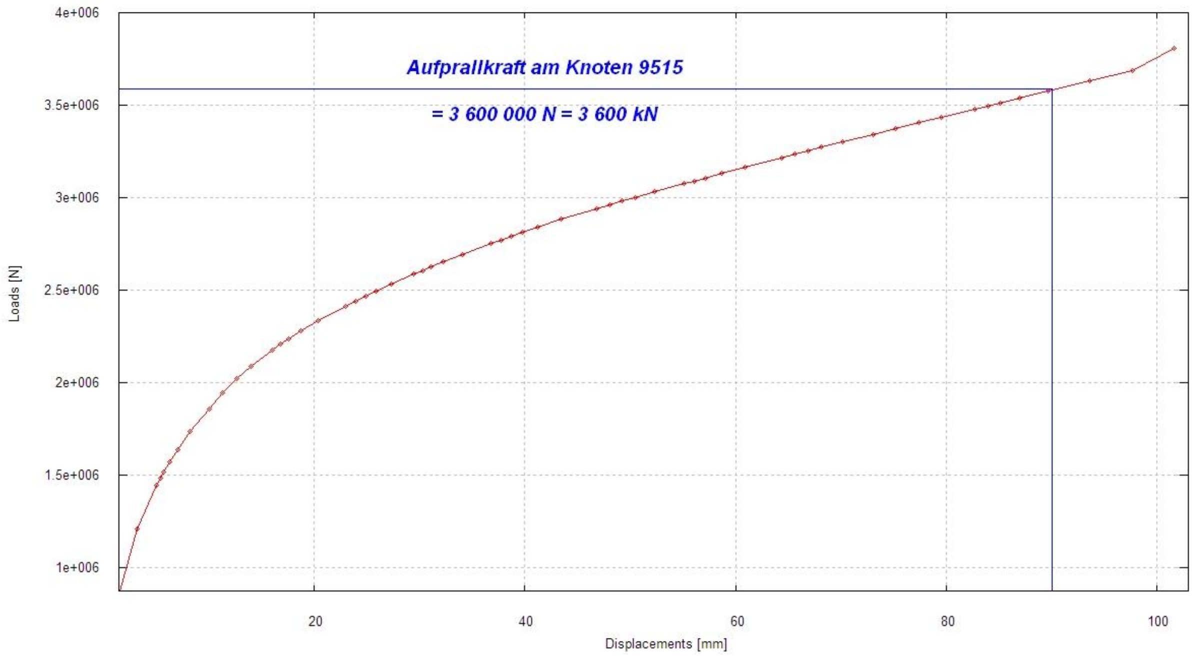
Y-deformation of -90 mm in the A-pillar area



v. Mises stress distribution of 76 N/mm² in the A-pillar area.

Calculating the impact force

The impact force can be read from the Load-Displacement Diagram for a deformation in the Y direction of -90 mm at node 9515.



The impact force of the rock is 3,600,000 N or 3,600 kN.

Calculating the impact velocity

First, the mass of the rock is calculated using a gravitational load, with a modulus of elasticity of 19,000 N/mm² and a density of 2,000 kg/m³. After the calculation, the sum of the support forces in the Y direction can be read as 12,188 N, resulting in a rock mass of 1,219 kg.

With the impact force, the mass, and the deformation of -90 mm, the impact speed can be calculated using the braking distance formula:

$$F = \frac{m\, v^2}{2 * S}$$

F = impact force

m = rock mass

S = distance traveled during an impact

v = average speed of the rock and car

This is solved for v:

$$v = \sqrt{\frac{F * 2 * S}{m}} = \sqrt{\frac{3600 \text{ kN} * 2 * 0.09 \text{ m}}{1219 \text{ kg}}}$$

$$= 23 \text{ m/s} = 82 \text{ km/h}$$

The impact speed of the rock is twice as high due to the stationary car:

$$v = (v_{Auto} + v_{Fels}) / 2$$

$$v_{Fels} = v * 2 = 23 \text{ m/s} * 2 = 46 \text{ m/s}$$

The impact speed of the rock is therefore 46 m/s or 166 km/h.

Calculating the Fall Height and Fall Time

Using the speed of the rock, the fall height and fall time can be calculated using the free fall formula:

$$V = \sqrt{2 * g * H}$$

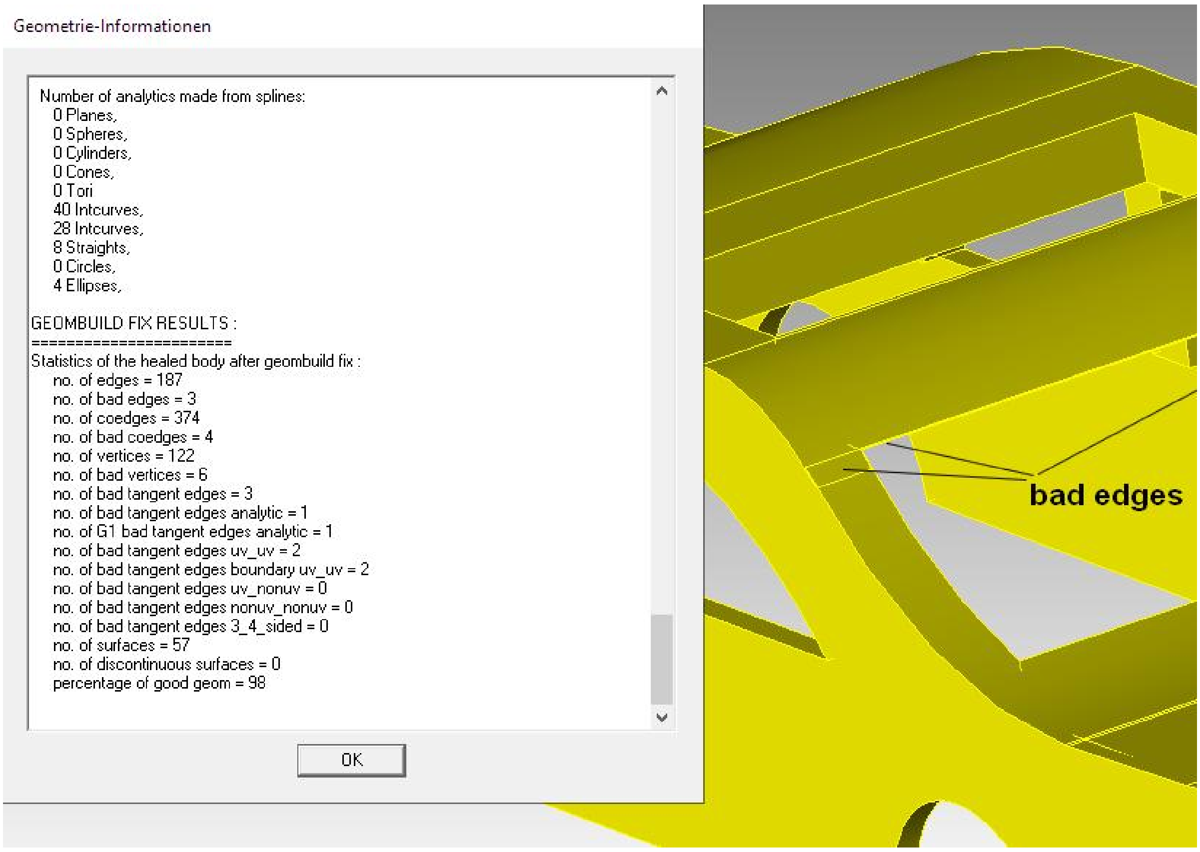
$$H = \frac{V^2}{2 * g} = \frac{46 \text{ m/s} * 46 \text{ m/s}}{2 * 9.81 \text{ m/s}^2} = 105 \text{ m}$$

$$t = \sqrt{\frac{2 * H}{g}} = \sqrt{\frac{2 * 105 \text{ m}}{9.81 \text{ m/s}^2}} = 4.6 \text{ s}$$

The fall height of the rock is 105 m with a fall time of 4.6 s.

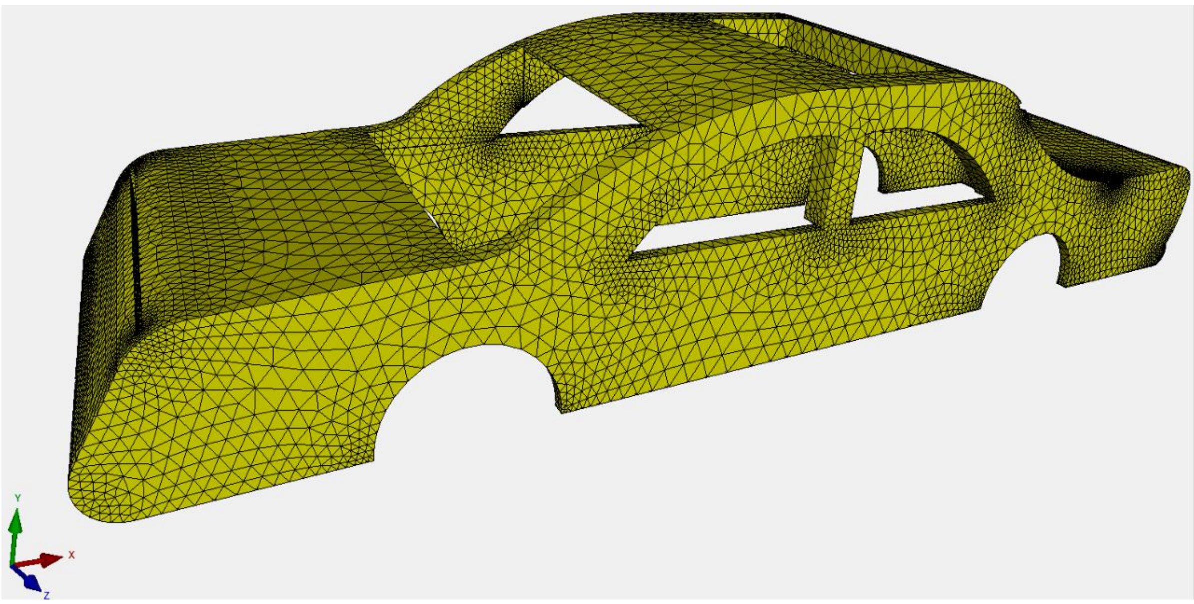
FEM Model Car

The FEM model Car is generated from a STEP file from www.grabcad.com using the 3D mesh generator NETGEN. However, six "bad edges" or six unconnected edges must first be optimized using the repair tool Transmagic.



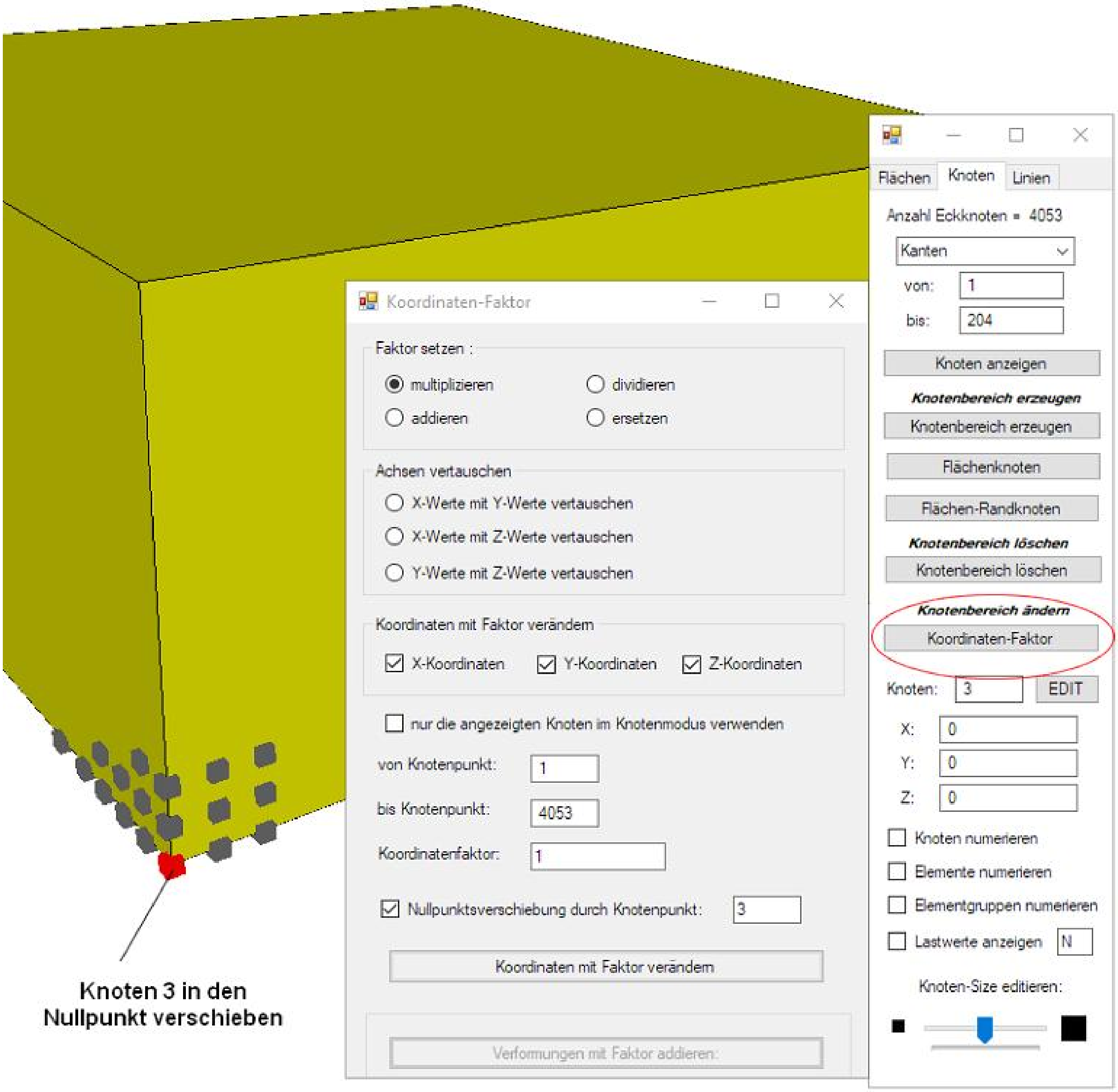
After mesh generation, you get an FEM model consisting of 197,248 tetrahedra and 41,070 nodes. Save the FEM model as Car.FEM.

.



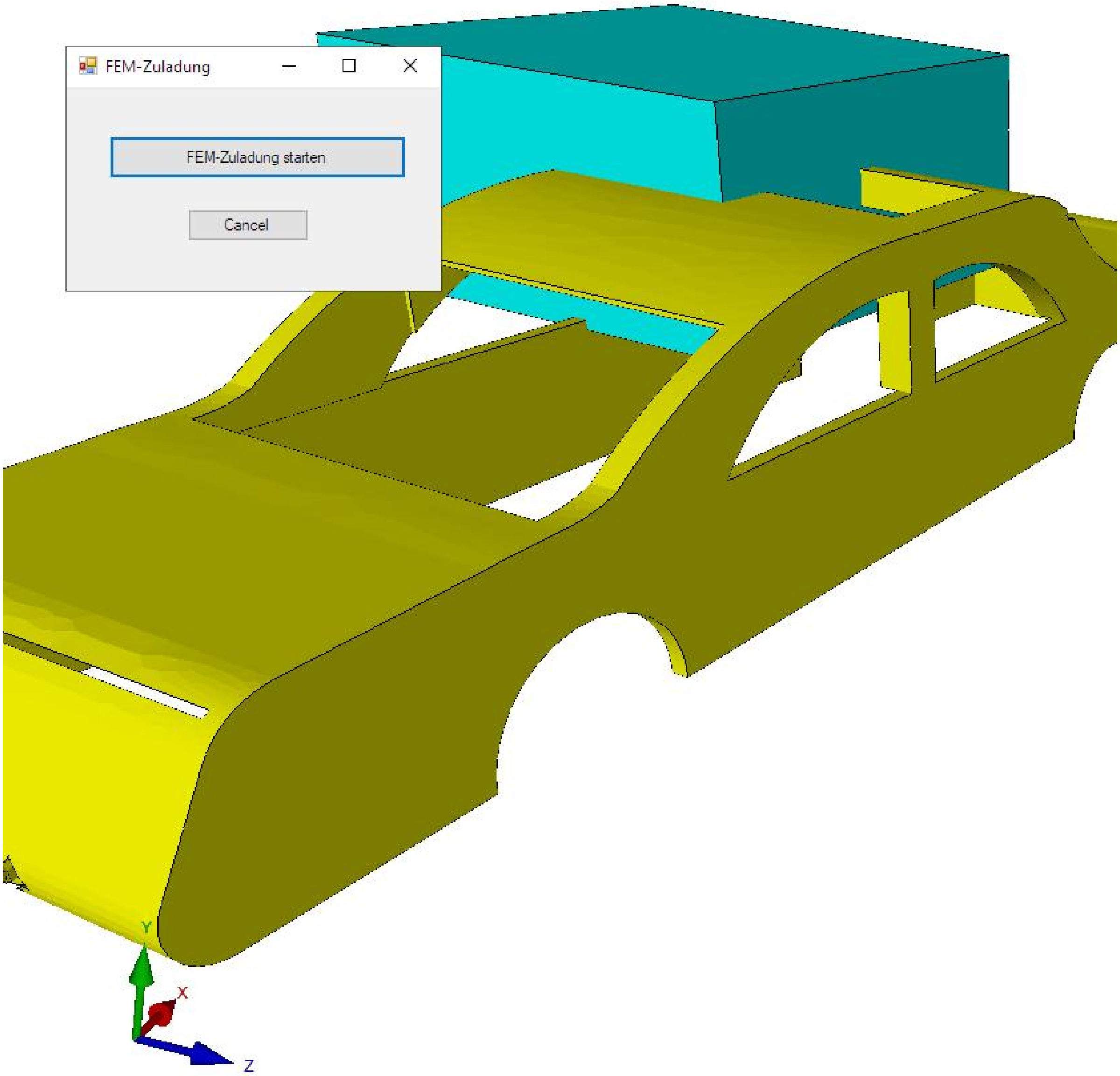
FEM Model Rock

Create a CAD cube measuring 1000 mm x 500 mm x 1000 mm and create an FEM model consisting of 19,136 tetrahedra and 4,053 nodes. Perform a zero point shift through node 3 using the "Coordinate Factor" menu in Node Mode and save the FEM mesh as "Rock.FEM."



Merge FEM Models

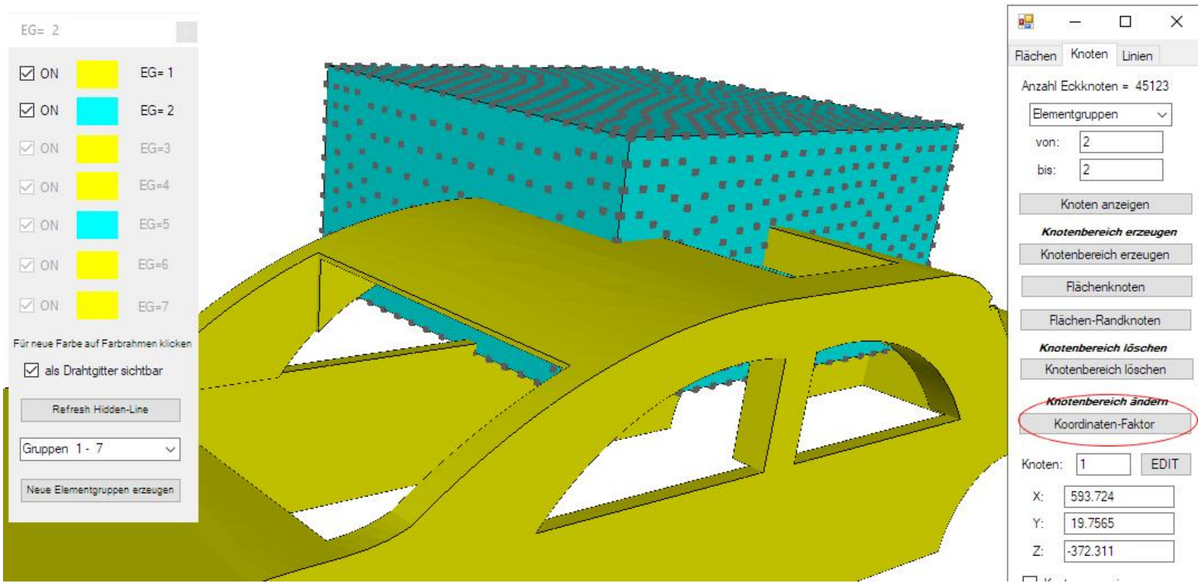
First, load the FEM model Auto.FEM normally. Then, select the "File" tab and the "FEM Merge" menu to add the second FEM model Rock.FEM.



You will receive an FEM model consisting of 216,384 TET4 elements, 45,123 nodes, and two element groups.

Coordinate Move of the Rock

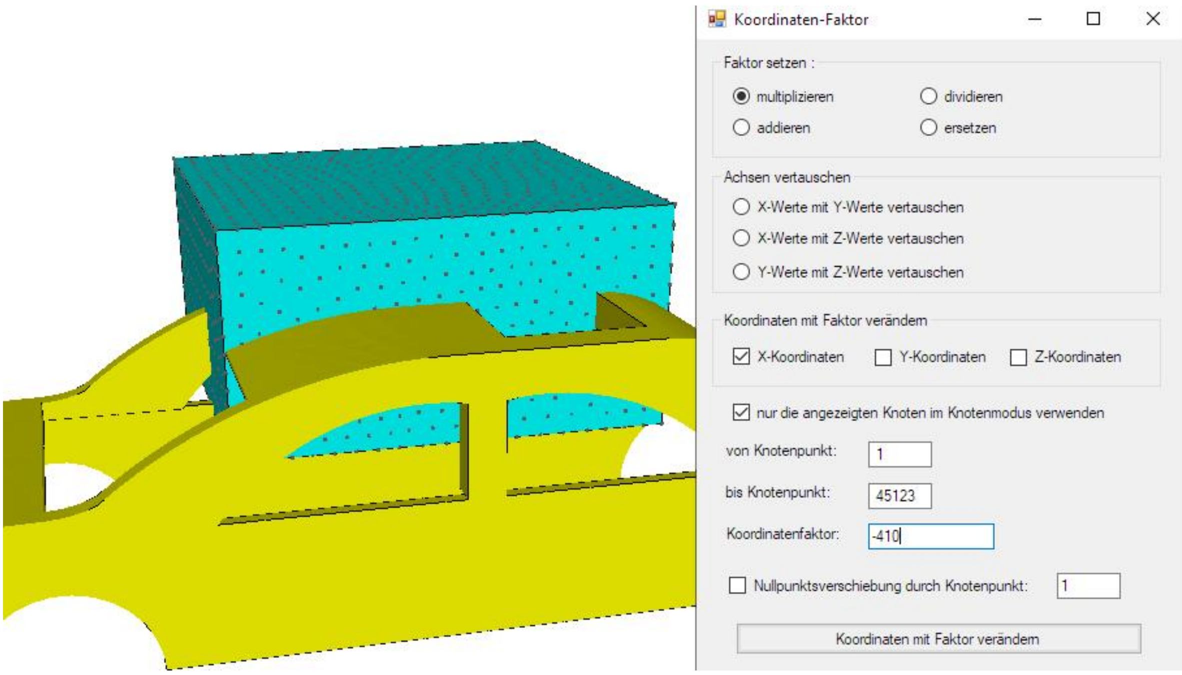
Display all nodes of element group 2 in Node Mode



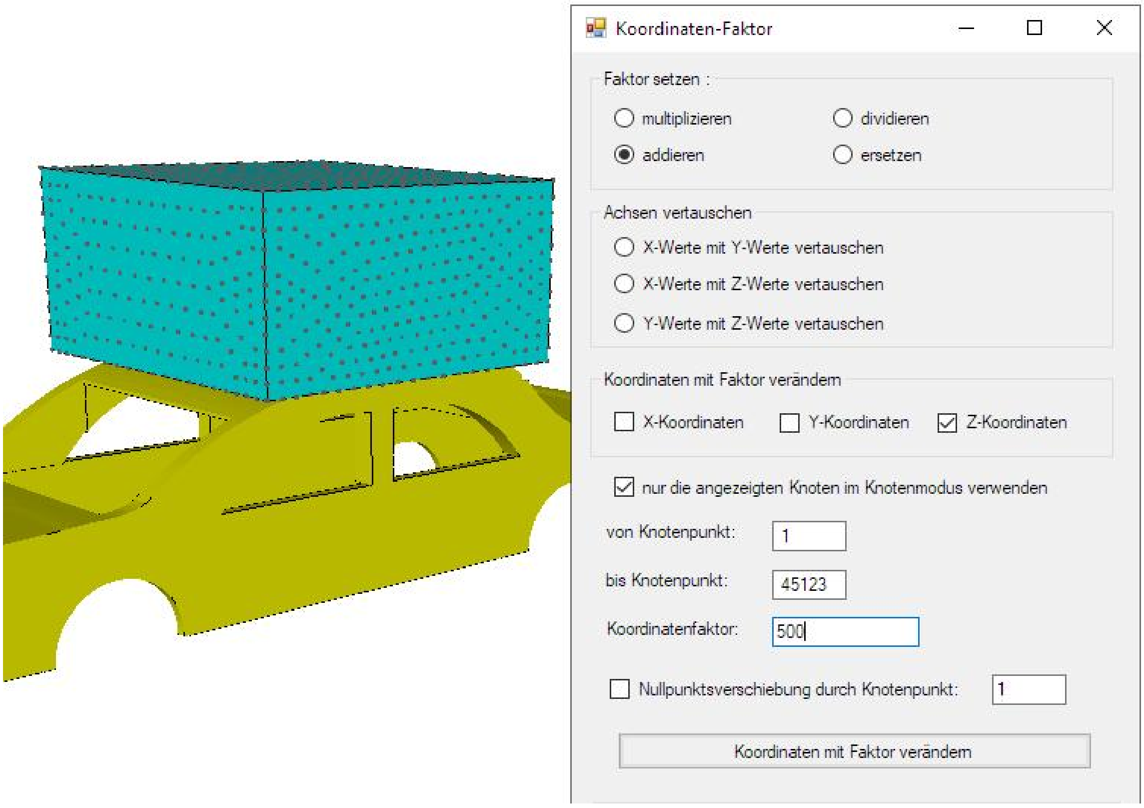
and use "Coordinate Factor" to move the rock by three coordinate moves at the following distances:

- 410 mm in the X direction
315 mm in the Y direction
500 mm in the Z direction

First coordinate move of element group 2 by - 410 mm in the X direction:



Third coordinate move of element group 2 by 500 mm in the Z direction:



Material Data

Using the "Edit FEM Project" tab and the "Material Data" menu, enter the following material data:

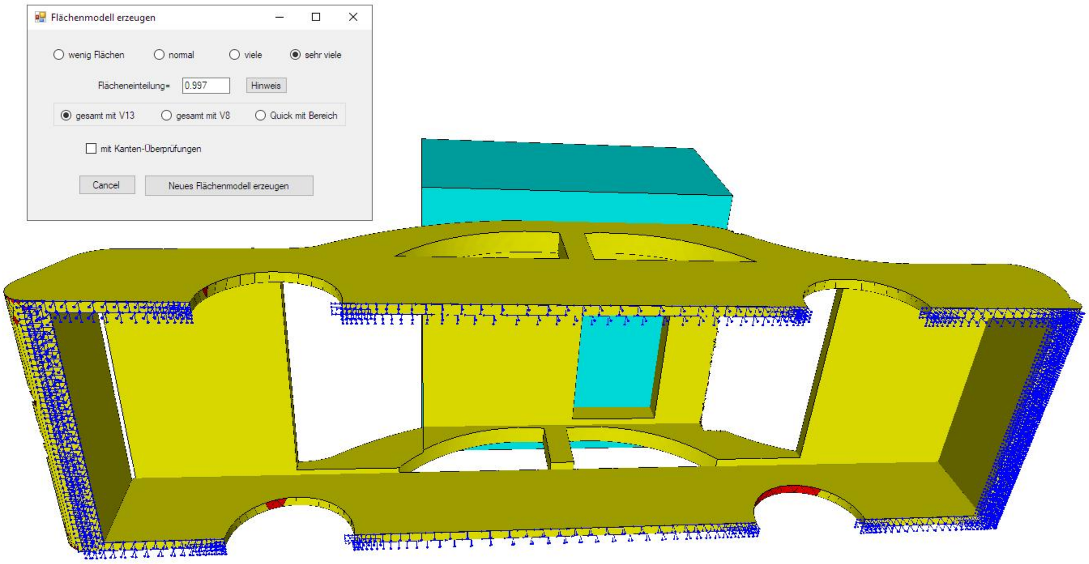
Element group 1: Car body
Material: Aluminum
Young's modulus: 71,000 MPa
Poisson's ratio = 0.34
Density = 2,700 kg/m³

Yield strength = 70 MPa -> the nonlinear stress-strain curve for Alu 999 must be selected in the FEM analysis "Material-Nonlinear"

Element Group 2: Rock
Material: Natural hard rock granite
Young's modulus: 100,000 MPa
Poisson's ratio = 0.25
Density = 3100 kg/m³

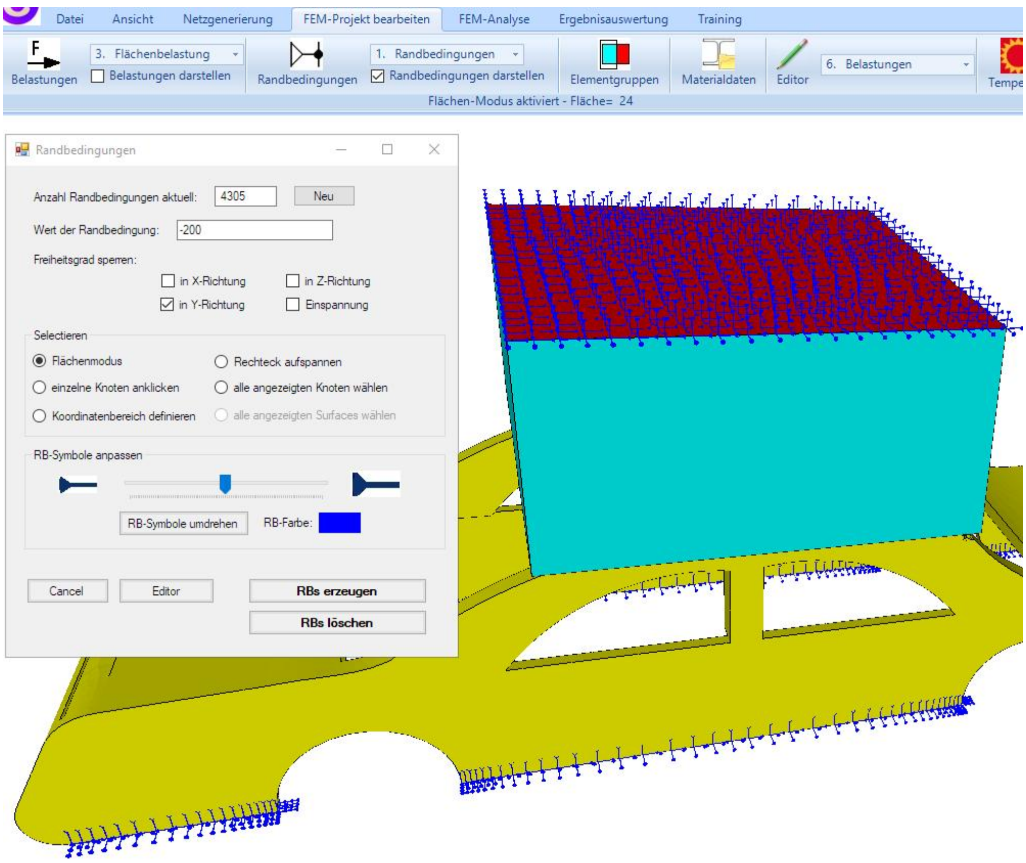
Create boundary conditions

Create a fine surface model in Surface mode with the "Very Many" setting and use "Edit FEA Project" and "Boundary Conditions" to firmly constrain the soil in the X, Y, and Z directions.



Create prescribed boundary conditions

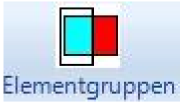
The upper and lower surface of the rock are fixed in the X and Z directions, and a prescribed displacement of "-200" mm is specified in the Y direction.



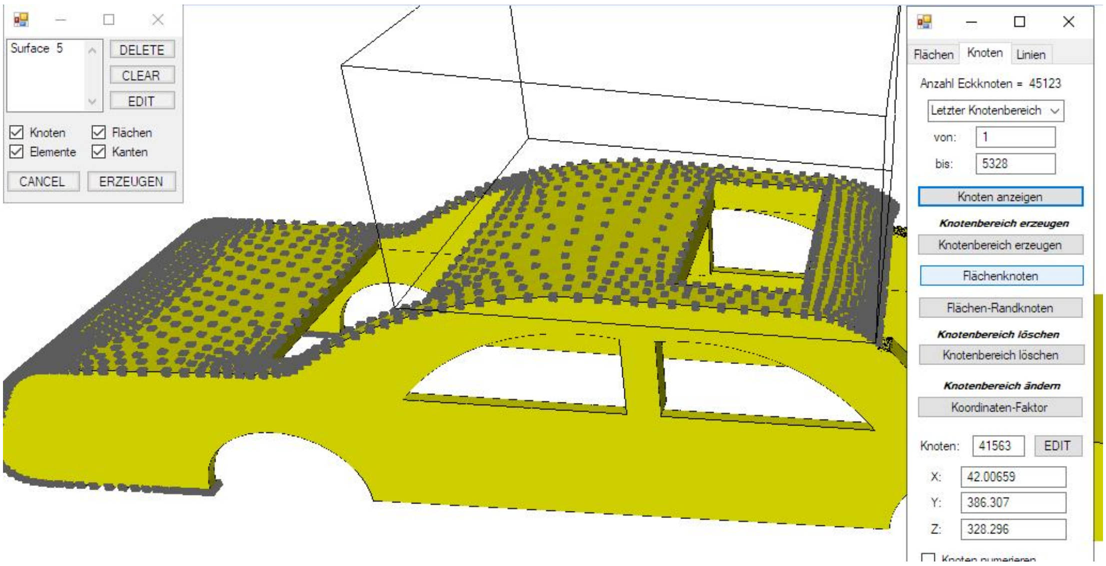
Create Load Case 1

Create any desired, but very small, nodal load, as the external load is already specified by the boundary conditions above.

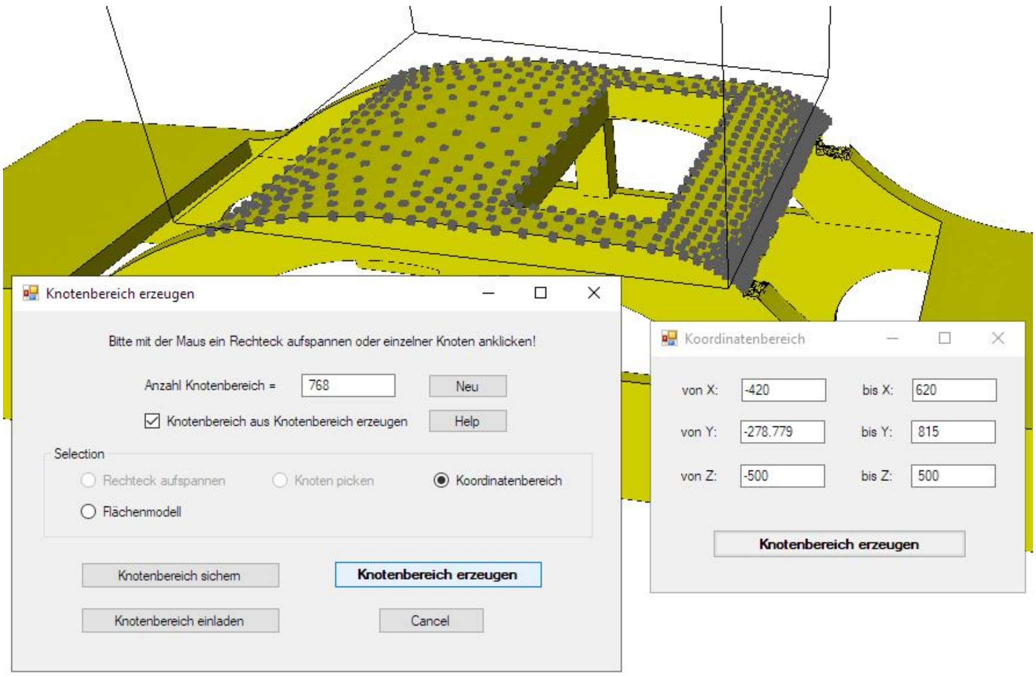
Create Load Case 2 with Master Contact Surface

Create the master contact surface by hiding element group 2 using 

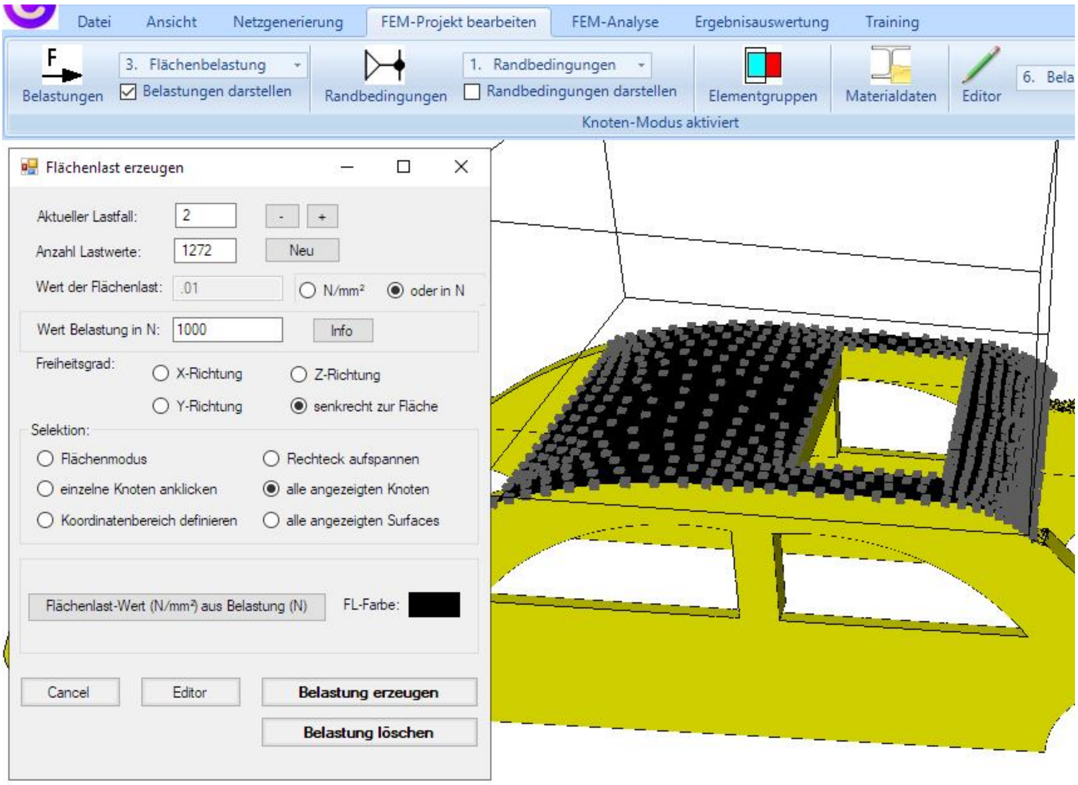
and then displaying the surface nodes of surface 5 in Node Mode.



Reduce this node of range again with “Create a Node Range” with a second node range from -420 mm to 620 mm in the X direction.

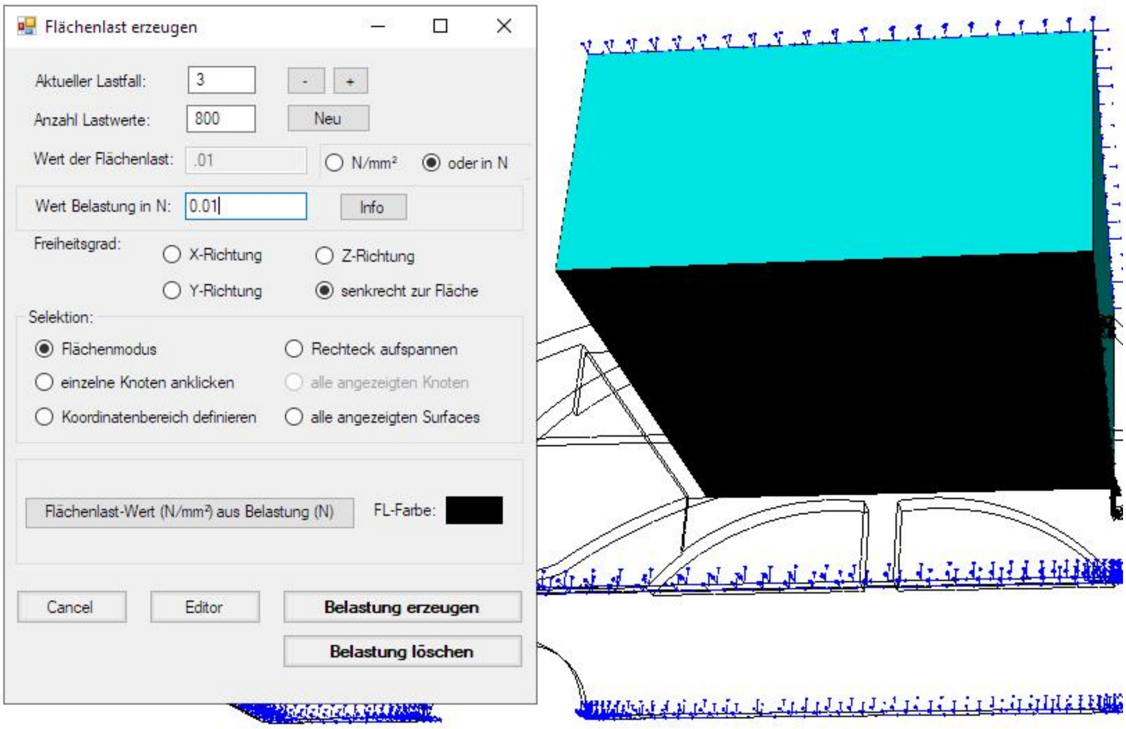


Finally, a surface load is created with load case 2 and the "all displayed nodes" selection without a load value.



Create load case 3 with a Slave Contact Surface

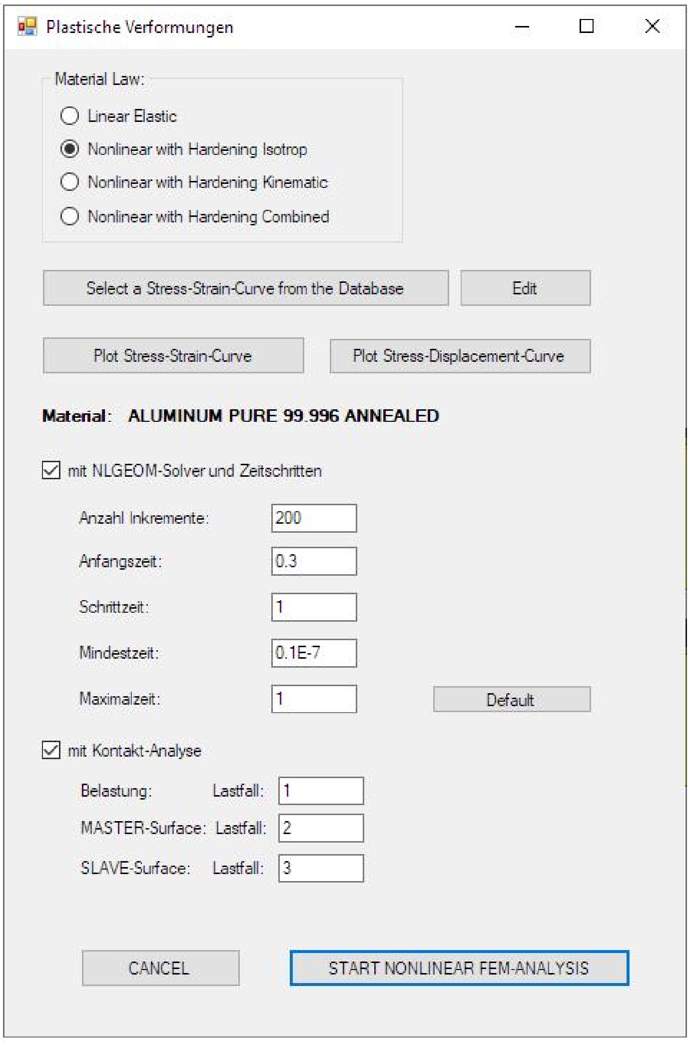
Create the slave contact surface by redisplaying element group 2 and hiding element group 1, and create a surface load with load case 3 and the "surface mode" selection on the lower surface of the rock.



Nonlinear Contact Analysis

The FEA assembly consisting of the car body and the rock can now be calculated using a nonlinear contact analysis. Select the "FEA Analysis" tab and the "Material - Nonlinear" menu and define:

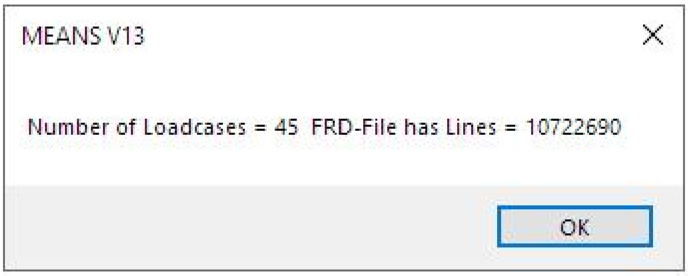
Use the "Select a Stress Curve from the Database" menu to select the stress-strain curve "ALUMINIUM PURE 99 996 ANNEALED."
Use the "With NLGEOM Solver and Time Steps" option to configure the solver for large deformations and adopt the number of increments, initial time, step time, minimum time, and maximum time from the default settings.
Use the "With Contact Analysis" option to enable contact analysis.



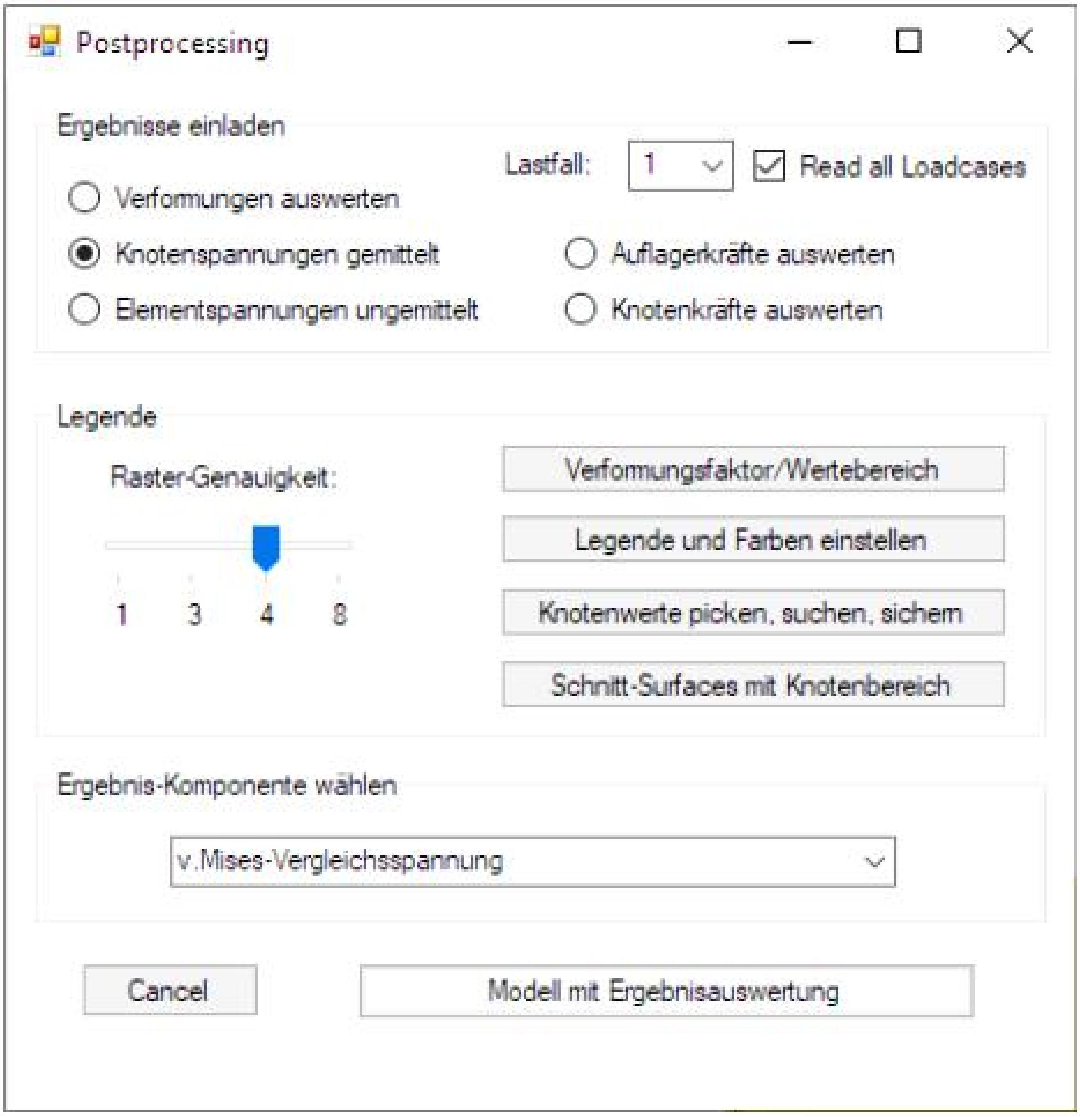
Postprocessing

Select the "Postprocessing" tab and click the icon  to evaluate the results.

Activate "Read all load cases" to first determine the number of load cases from the FRD file, as the FEM solver often converges slowly towards the end with the error message "too many cutbacks" and aborts the calculation.



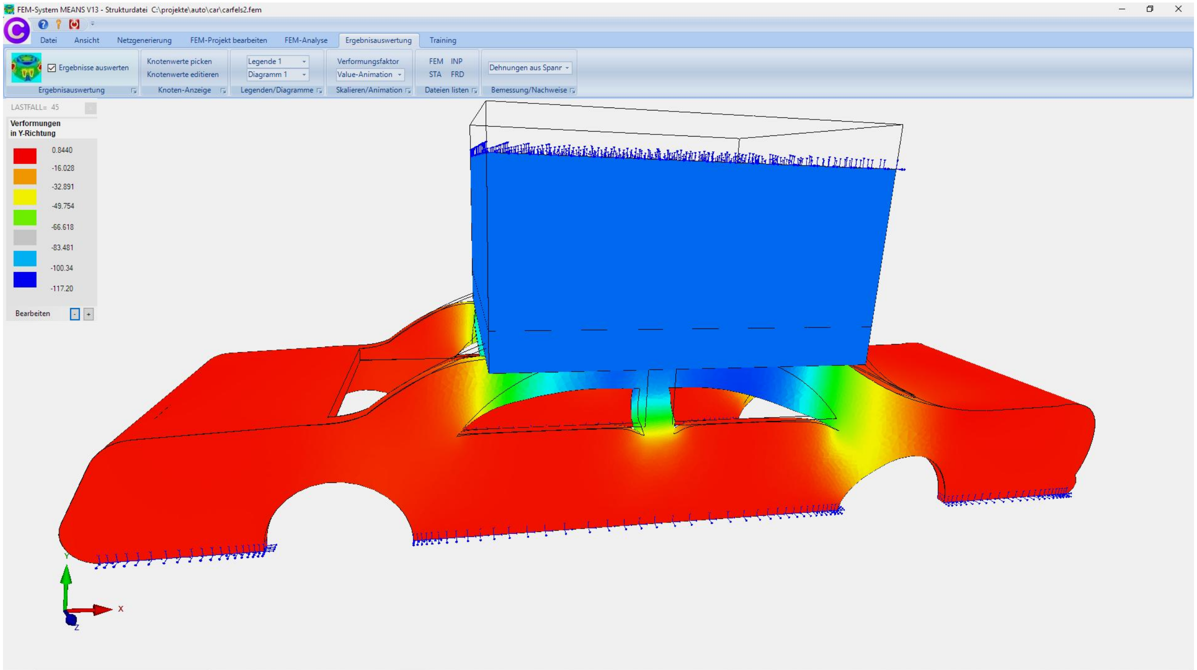
Select "Model with Results Evaluation" from the menu with the desired load case to display the stress or deformation distribution.



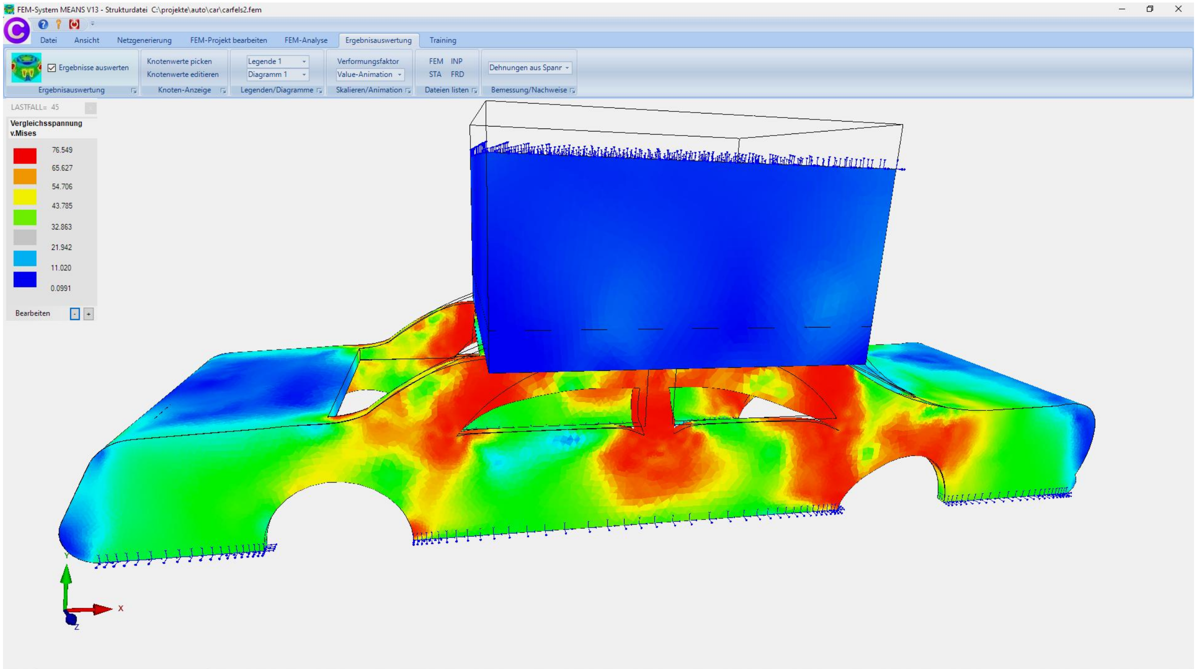
Stress and deformation distribution

For the final load case 45, the following plastic deformations and nodal stresses with a stress-strain curve for aluminum result:

Maximum plastic deformation in the Y direction = -117.20 mm



v.Mises nodal stresses = 76 MPa

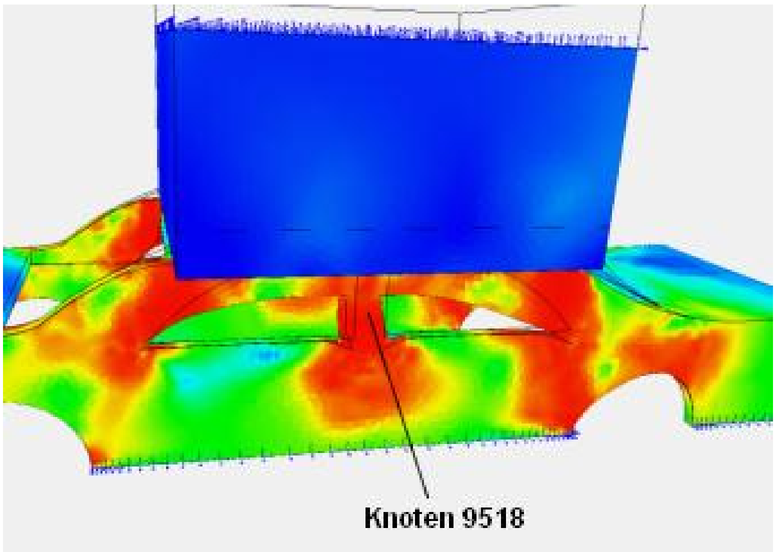


Sum of impact force = 3,800,795 N = 3,800 kN = 380 tons

Select the "Postprocessing" tab, "Pick Node Values," and "Sum of Reaction Forces" to display the sum of reaction forces.

Load-Displacement-Diagram

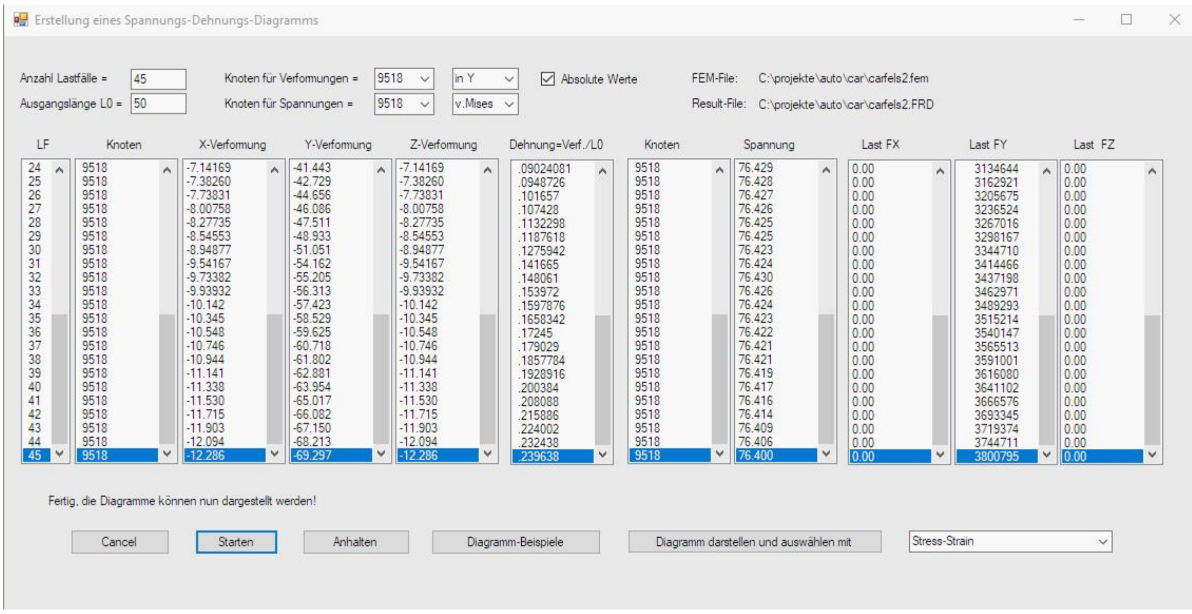
In addition, the impact force on the A-pillar at node 9518 can be determined from the load-deformation diagram.



Select the "Postprocessing" tab, then the "Stress-Strain Diagram" menu to open an 11-column dialog box.

Enter node 9518 and select "Start" to list the deformations, stresses, and forces at that node for each load case.

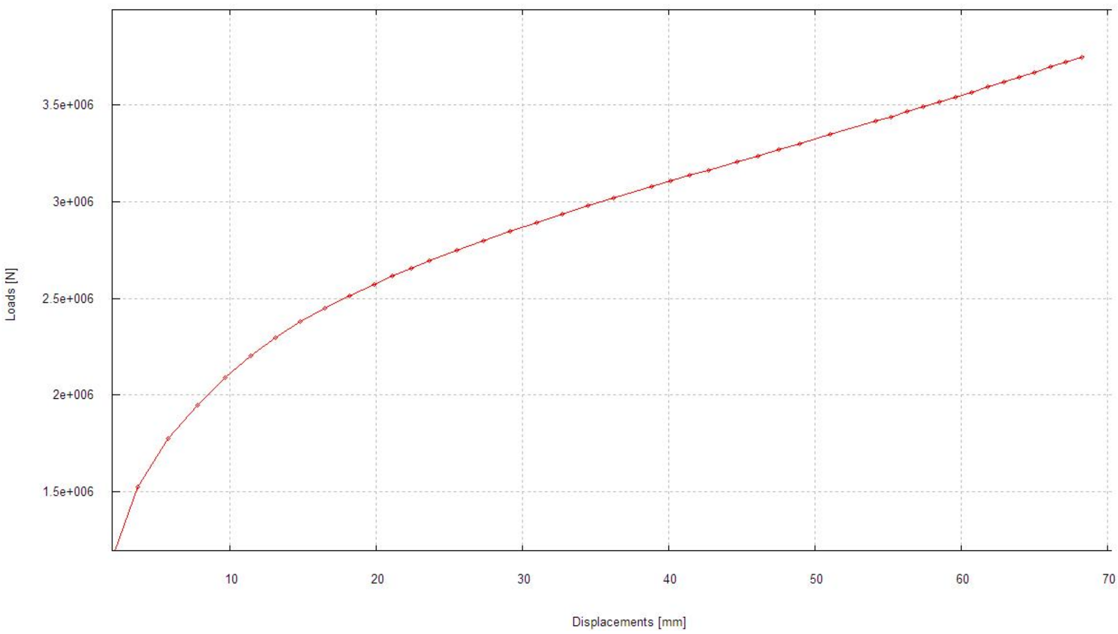
After a few minutes, select "Display diagram and select with" and "Load-Displacement" to display the load-deformation diagram..



LF	Knoten	X-Verformung	Y-Verformung	Z-Verformung	Dehnung=Verf./L0	Knoten	Spannung	Last FX	Last FY	Last FZ
24	9518	-7.14169	-41.443	-7.14169	.09024081	9518	76.429	0.00	3134644	0.00
25	9518	-7.38260	-42.729	-7.38260	.0948726	9518	76.428	0.00	3162921	0.00
26	9518	-7.73831	-44.656	-7.73831	.101657	9518	76.427	0.00	3205675	0.00
27	9518	-8.00758	-46.086	-8.00758	.107428	9518	76.426	0.00	3236524	0.00
28	9518	-8.27735	-47.511	-8.27735	.1132298	9518	76.425	0.00	3267016	0.00
29	9518	-8.54553	-48.933	-8.54553	.1187618	9518	76.425	0.00	3298167	0.00
30	9518	-8.94877	-51.051	-8.94877	.1275942	9518	76.423	0.00	3344710	0.00
31	9518	-9.54167	-54.162	-9.54167	.141665	9518	76.424	0.00	3414466	0.00
32	9518	-9.73382	-55.205	-9.73382	.148061	9518	76.430	0.00	3437198	0.00
33	9518	-9.93932	-56.313	-9.93932	.153972	9518	76.426	0.00	3462971	0.00
34	9518	-10.142	-57.423	-10.142	.1597876	9518	76.424	0.00	3489293	0.00
35	9518	-10.345	-58.529	-10.345	.1658342	9518	76.423	0.00	3515214	0.00
36	9518	-10.548	-59.625	-10.548	.17245	9518	76.422	0.00	3540147	0.00
37	9518	-10.746	-60.718	-10.746	.179029	9518	76.421	0.00	3565513	0.00
38	9518	-10.944	-61.802	-10.944	.1857784	9518	76.421	0.00	3591001	0.00
39	9518	-11.141	-62.881	-11.141	.1928916	9518	76.419	0.00	3616080	0.00
40	9518	-11.338	-63.954	-11.338	.200384	9518	76.417	0.00	3641102	0.00
41	9518	-11.530	-65.017	-11.530	.208088	9518	76.416	0.00	3666576	0.00
42	9518	-11.715	-66.082	-11.715	.215886	9518	76.414	0.00	3693345	0.00
43	9518	-11.903	-67.150	-11.903	.224002	9518	76.409	0.00	3719374	0.00
44	9518	-12.094	-68.213	-12.094	.232438	9518	76.406	0.00	3744711	0.00
45	9518	-12.286	-69.297	-12.286	.239638	9518	76.400	0.00	3800795	0.00

Load-Displacement Diagram

A plastic deformation of -70 mm results in an impact force of approximately 350 tons at node 9815. This is a plausible force when compared to the pressing force of an automobile press, which is over 500 tons.



Stress-Strain Diagram

Up to the aluminum yield strength of 75 MPa, stresses and strains are elastic. After that, plastic deformation begins, meaning the strains increase but the stresses change only slightly. The material begins to flow, and irreversible permanent deformation occurs.