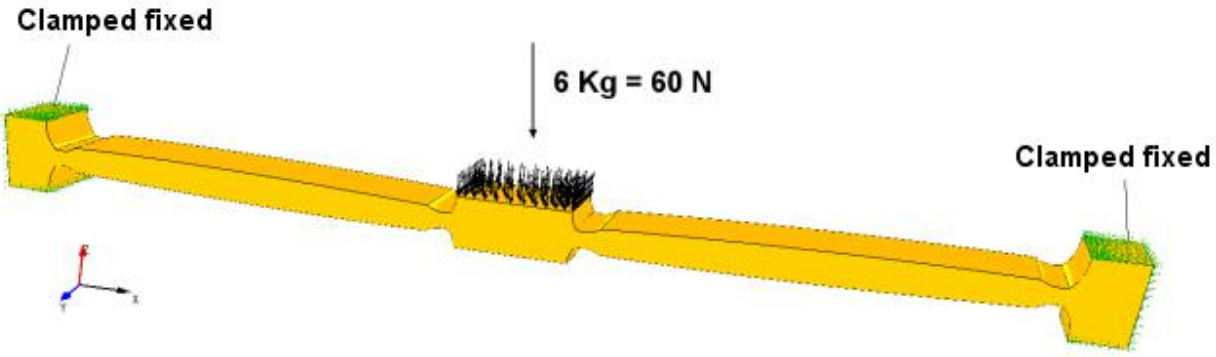
Part 13: Nonlinear Analysis with MEANS V11

A flat spring clamped on both sides is loaded with 6 kg or 60 N in the z direction. How different are the displacements and stresses v.Mises, S1 and S3 between a linear FEM-Analysis and a nonlinear FEM-Analysis with the add-on modul **MEANS-NONLIN**. Please visit the website www.fem-infos.com for a free demo from HTA-Software.

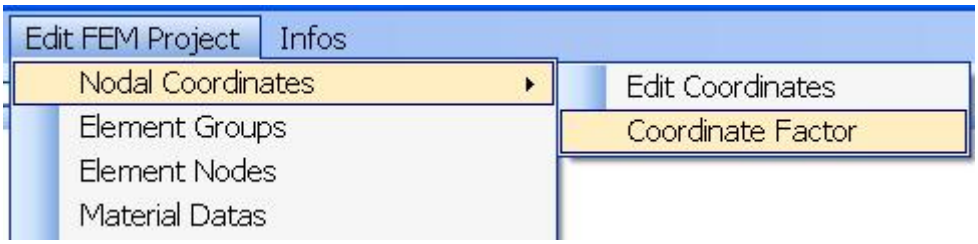


Currently, the following limitations for NONLIN must be observed:

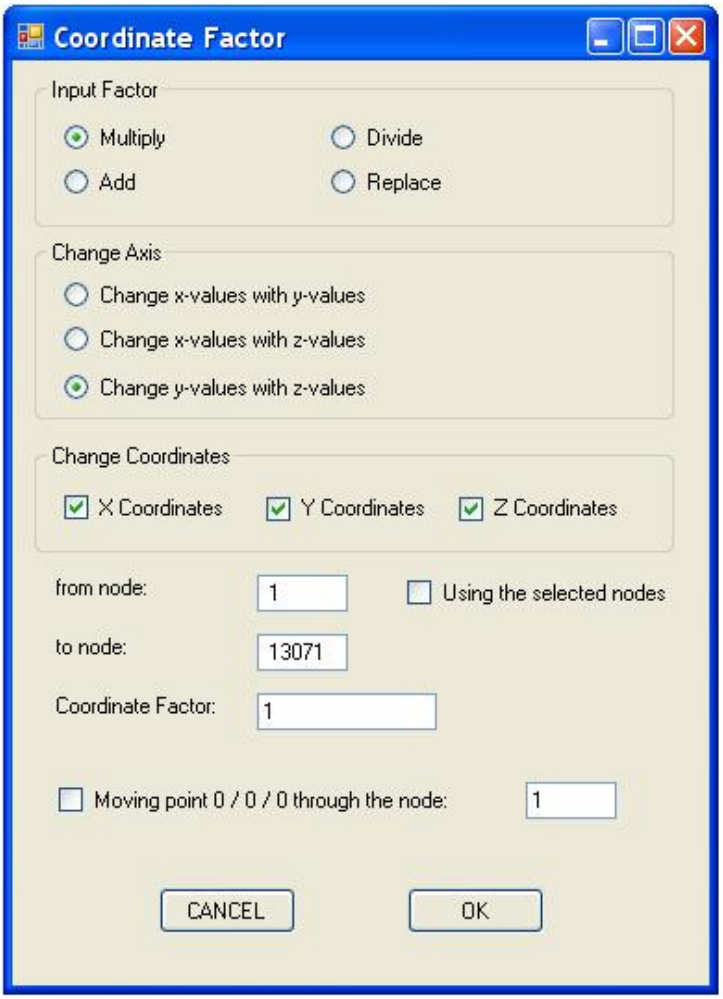
- Only a point load in z-direction with max. 5000 nodes
- Surface loads must be converted into a point load
- only Dr.Kühn-Solver can be used (Quick-Solver is not possible)
- max. 30 Load Increments are possible

Change Y- and Z-Coordinate Axes

For the point load in z direction we must change the Y- and Z-Axes with menu “Edit FEM Projekt”, “Nodal Coordinates” and “Coordinate Factor”.

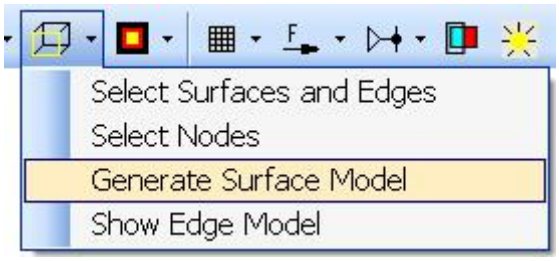


Choose “Change y-values with z-values” and “OK” to change the y- with z-axes.



Create the Surface Model

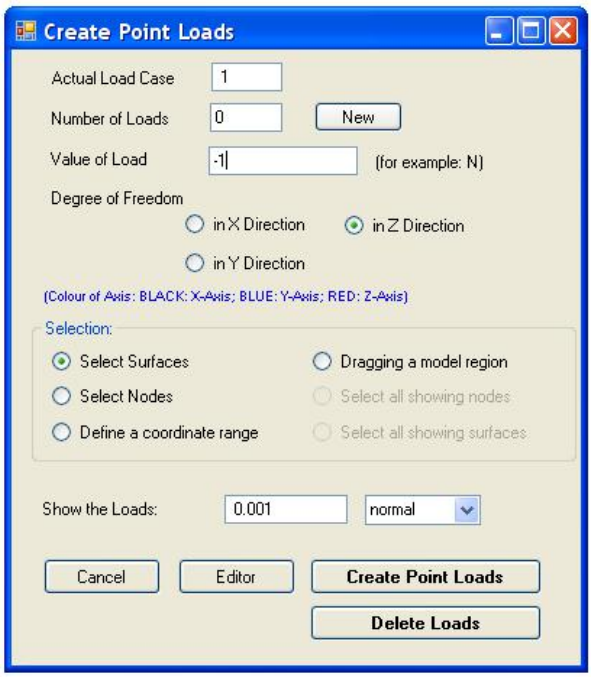
Create a moderate surface model with icon  and menu “Generate Surface Model” with 30 surfaces.



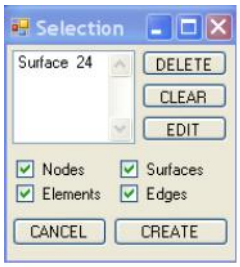
Create a Point Load

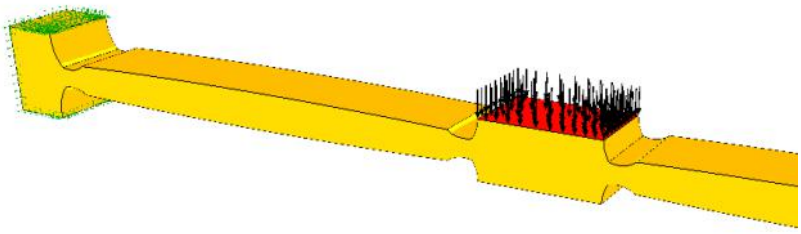
Select menu “Point Loads” and “Step 2: Create Point Load” and input the value of “-1”

in “z Direction”. Choose “Select Surfaces” and “Create Point Loads” and click on the surface 24 with a Double Click.



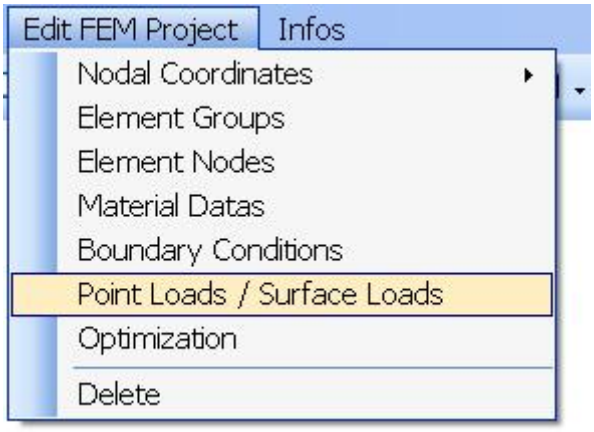
Select in the selectbox “Create” to create the point load with 113 nodes.





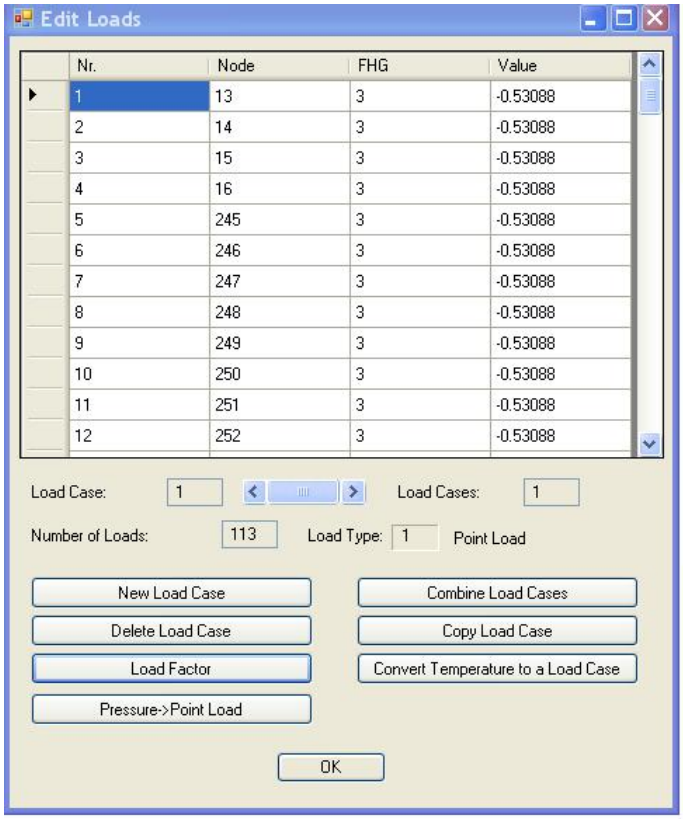
Edit the load values

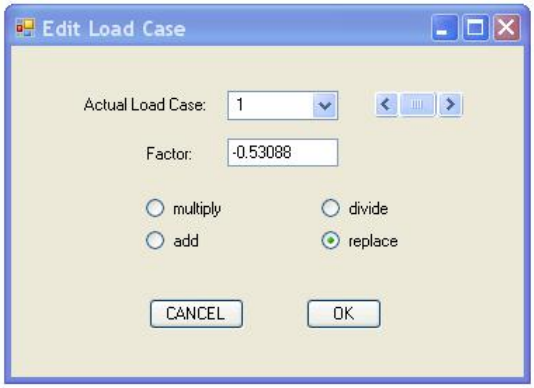
Select menu “Edit FEM Project” and”Point Loads / Surface Loads” to edit the load values.



Calculate the load value by dividing the load of 60 N by the number of loads of 113 to the load value of -0.53088 N.

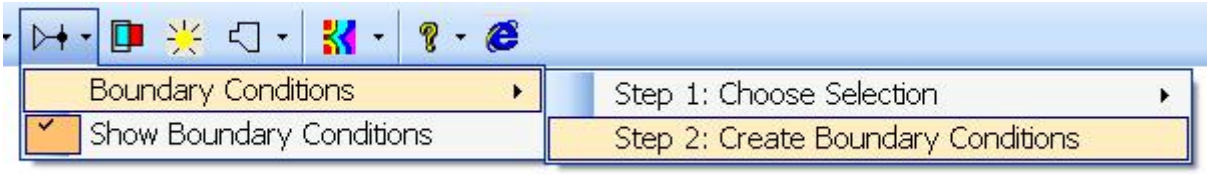
Select “Load Factor” and input the Factor “-0.53008” and choose “replace” in order to replace the load values.



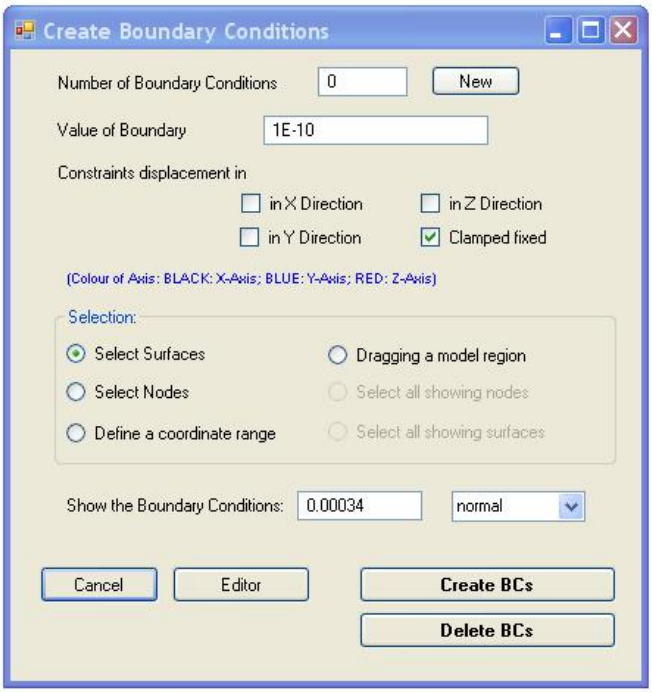


Create the Boundary Conditions

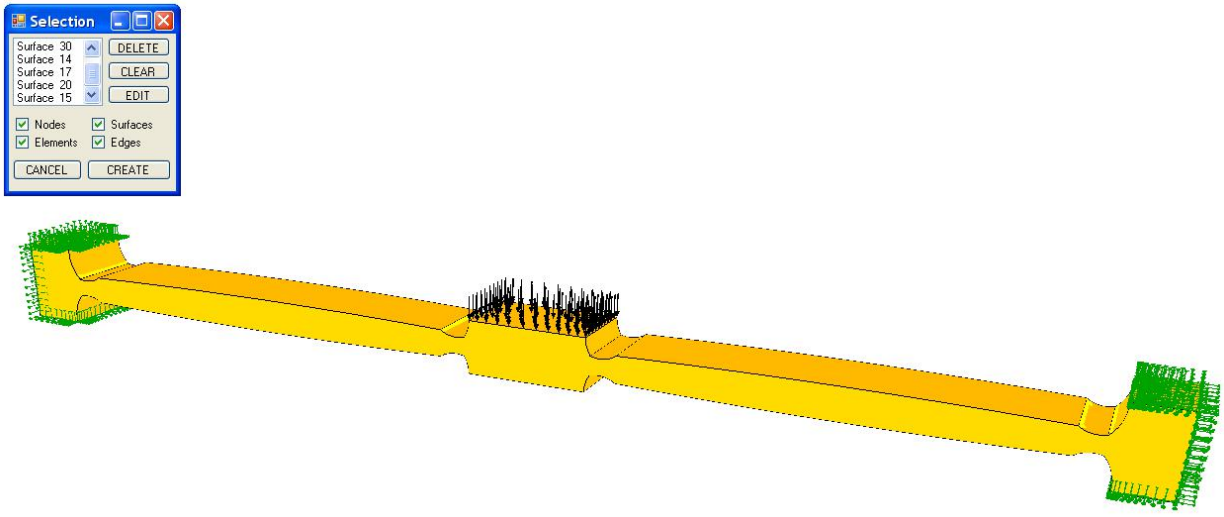
The flat spring is clamped fixed on both sides now select menu “Boundary Conditions” and “Step 2: Create Boundary Conditions”.



In the next dialogbox select “Clamped fixed” and “Select Surfaces” and create with “Create BCs” the BCs with a Double Click on the surface 14, 15, 17, 20, 27 and 30.

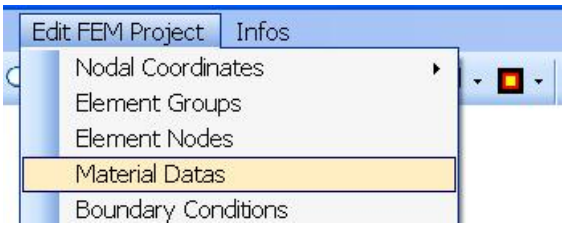


Select in the selectbox “Create” to create the Boundary Conditions.



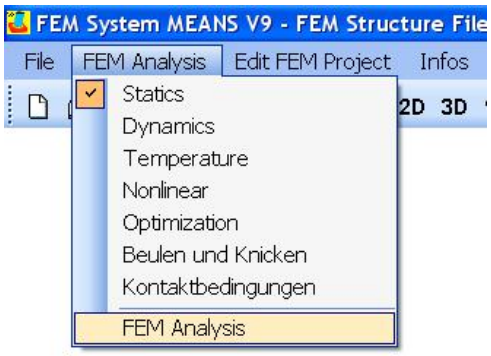
Input Material Datas

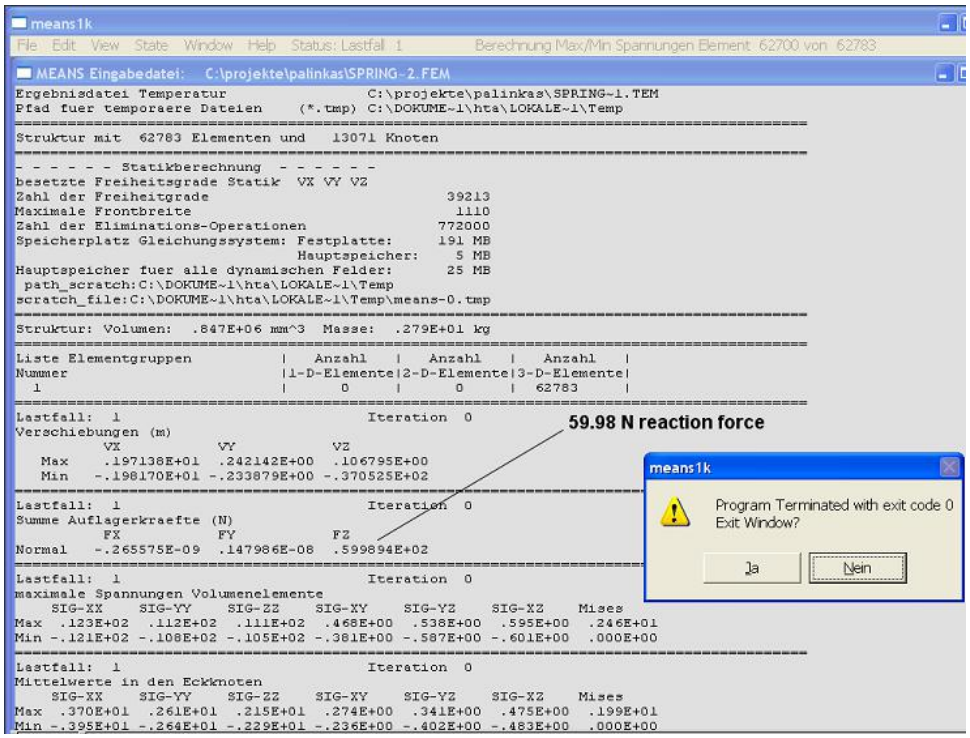
Select the menu „Edit FEM Project“ and „Material Datas“ and input the material datas for steel or other material.



Linear FEM-Analysis

First save the FEM file und the name “spring_linear.fem” and then choose FEM Analysis and begin with menu “Step 1: Starting FEM Analysis“ to calculate the displacements and stresses.



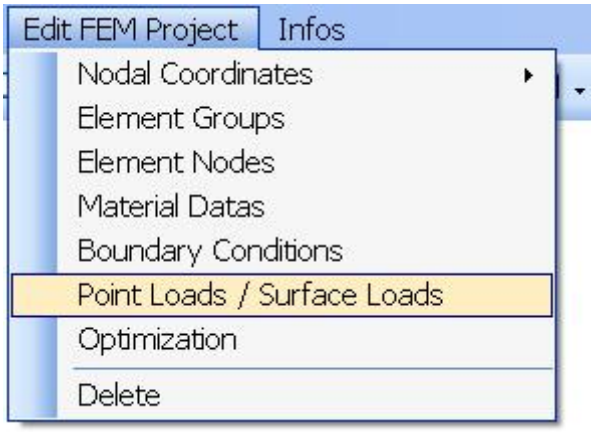


Nonlinear FEM-Analysis

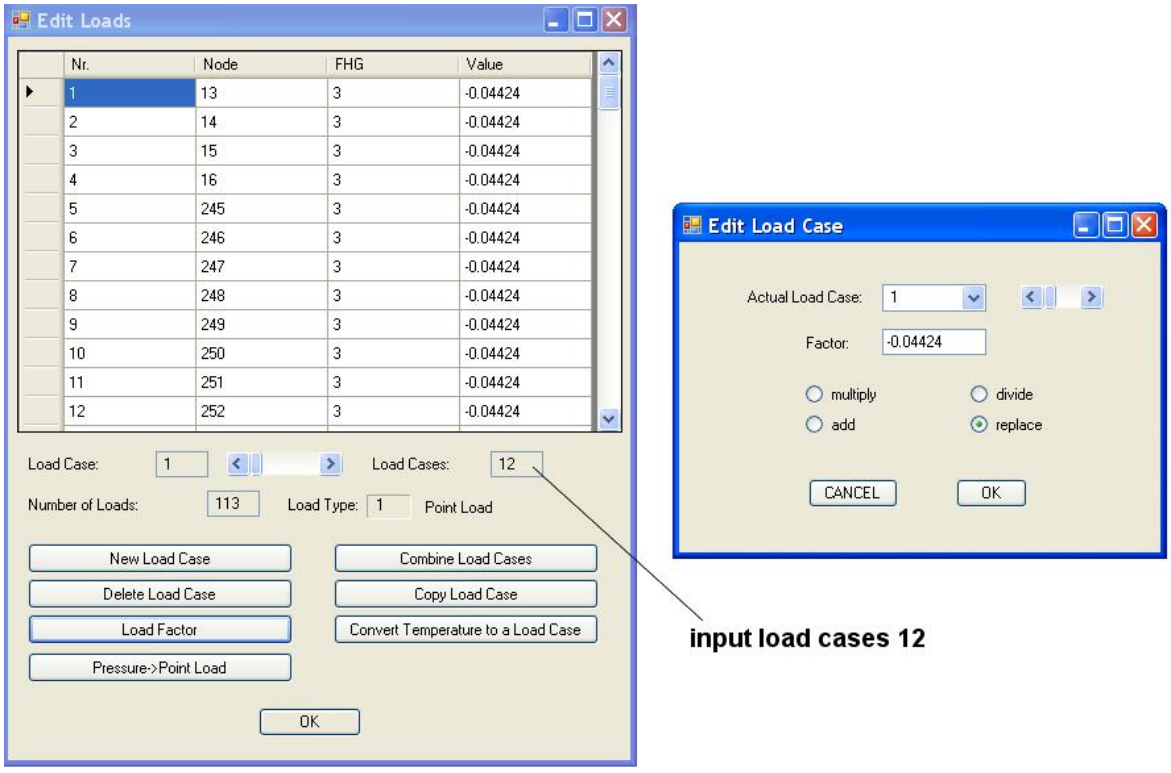
After the linear FEM-Analysis save the FEM file with name “spring_nonlinear.fem” and edit the loads with new load increments as following.

Create 12 load increments

Select menu “Edit FEM Project” and”Point Loads / Surface Loads” to edit the load values.

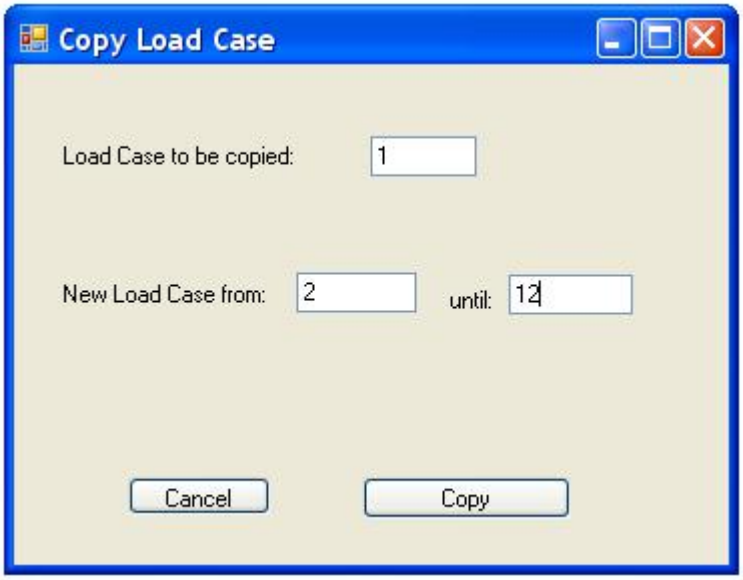


Calculate the new load value by dividing “-0.53008” by the number of load increments of “12” to the load value “-0.04424 N”.
Select “Load Factor” and input the Factor “-0.04424” and choose “replace” in order to replace the load values.



Copy Load Case

Select “Copy Load Case” and input the new Load Cases from “2” to “12” and select “Copy”. Also you must change the load cases to “12” and quit the editor with “OK”.

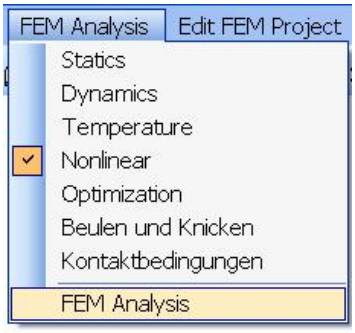


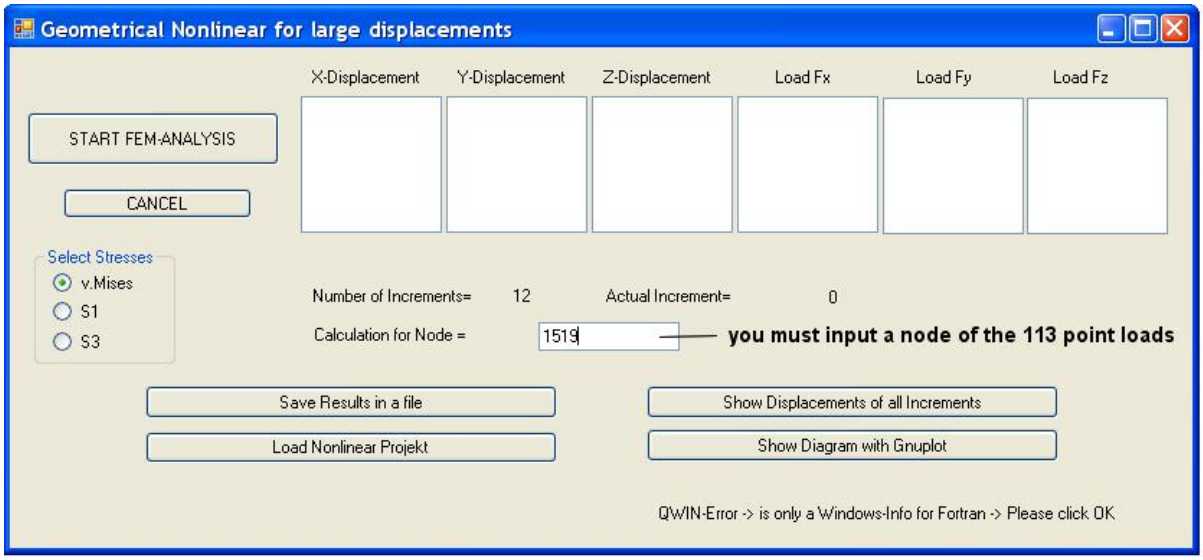
Check the loads again

It is better to queck the loads again with menu “Point Loads / Surface Loads” and now you must have 12 load cases with the load value “-0.04424”.

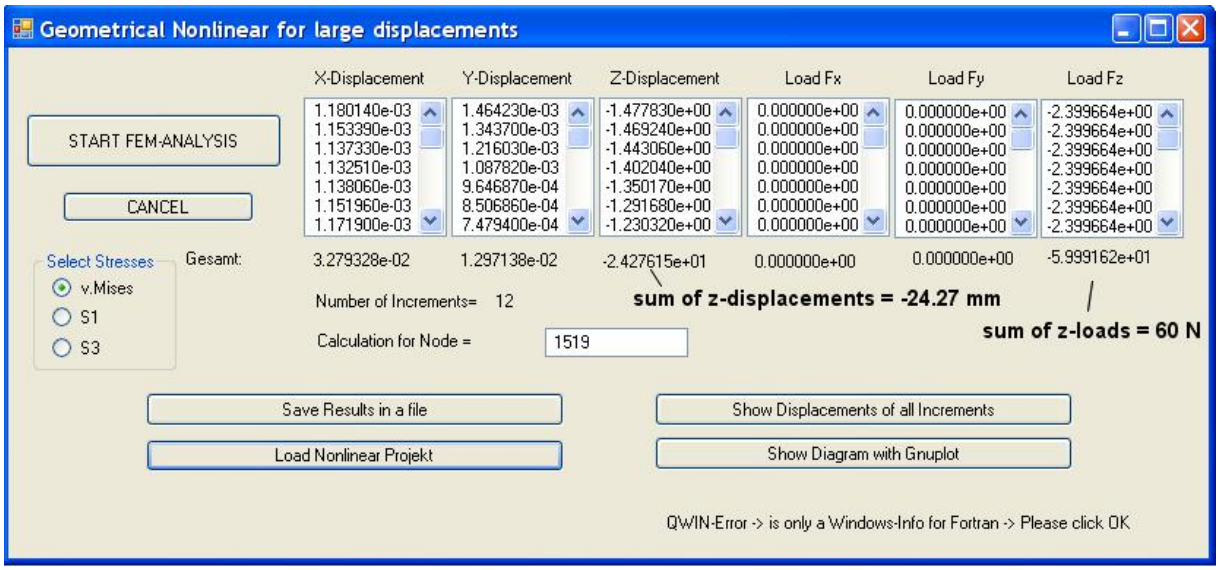
Nonlinear FEM-Analysis

Save again the FEM file under the name “spring_nonlinear.fem” and then select “Nonlinear” and choose FEM Analysis to calculate the displacements and stresses with 12 load increments. Input the node 1519 from the 113 load points in order to evaluate the results for each increment. Select “Start FEM-Analysis” to start NONLIN.



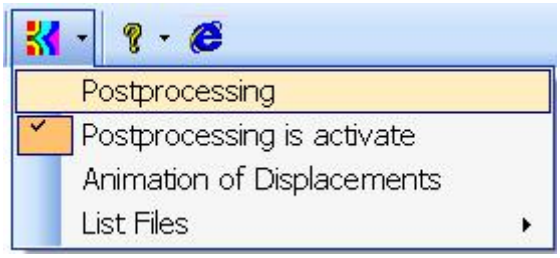


After the nonlinear Calculation select “Show Displacements of all Increments”



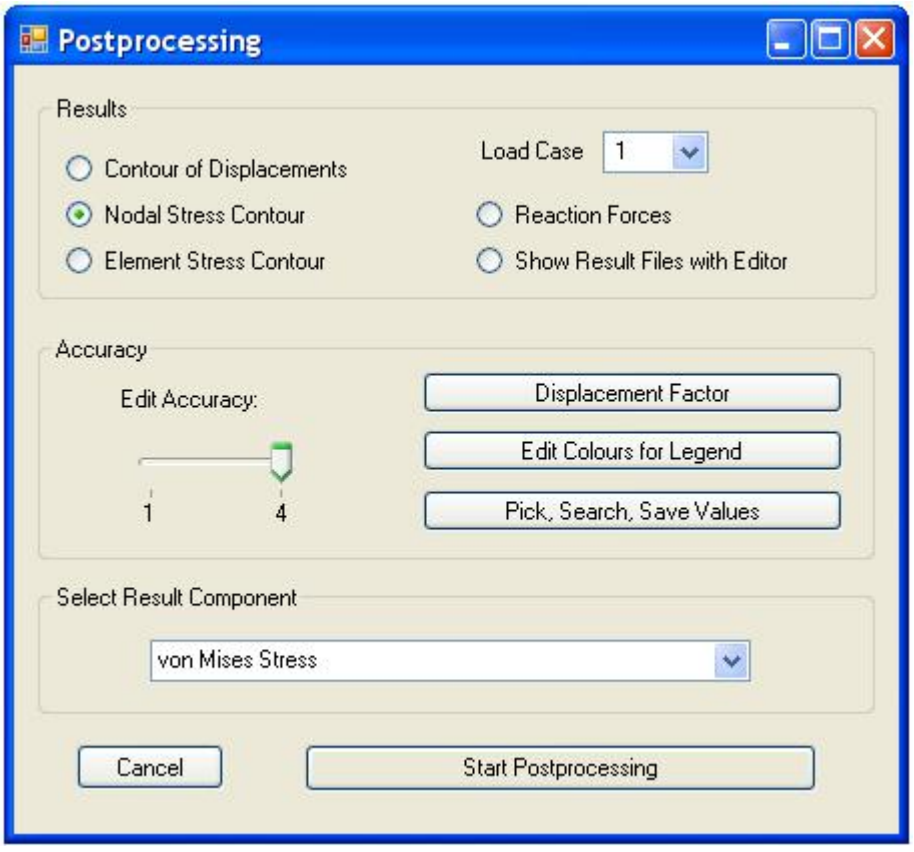
Postprocessing

Choose the menu „Postprocessing“ to evaluate the displacements and stresses.



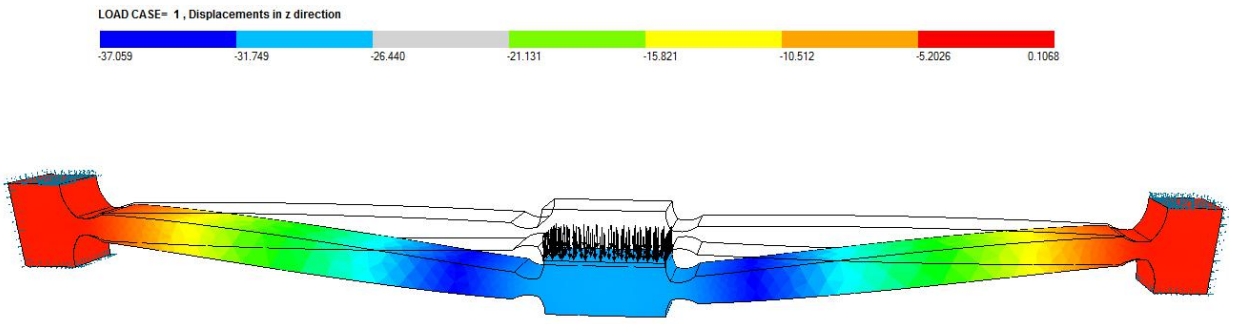
Please select following Postprocessing:

- Results: **Nodal Stress Contour**
- Load Case: **1**
- Edit Accuracy: **4**
- Result Component: **von Mises**

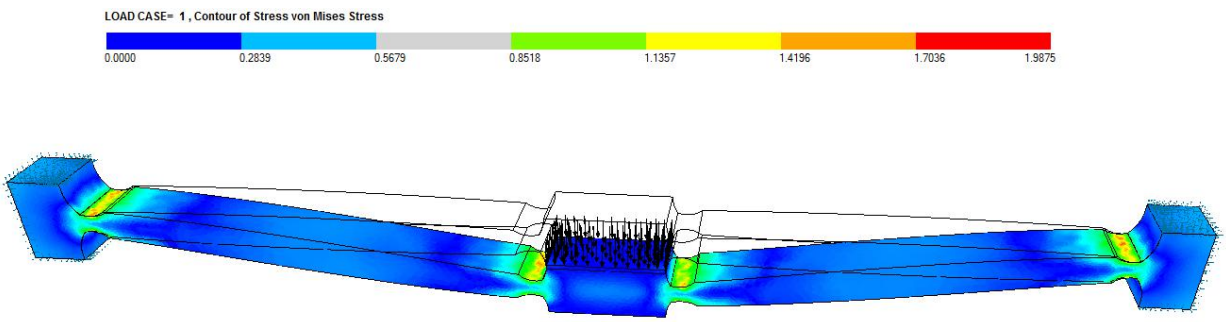


Linear Results:

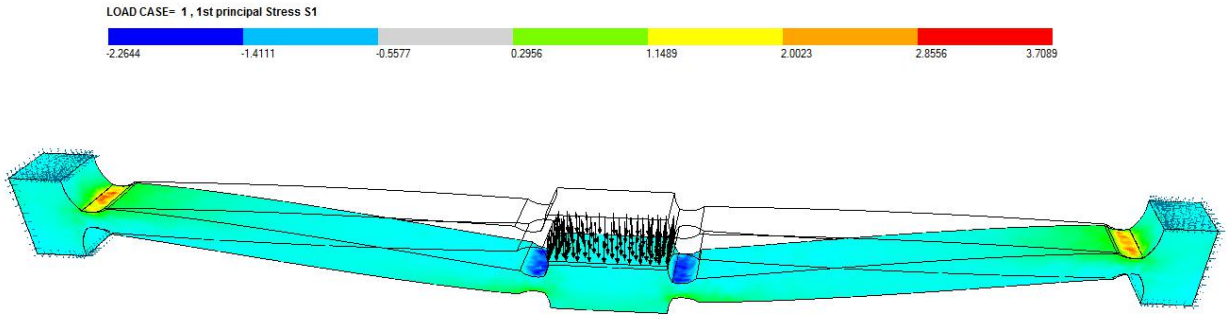
max. Displacement = - 37.05 mm



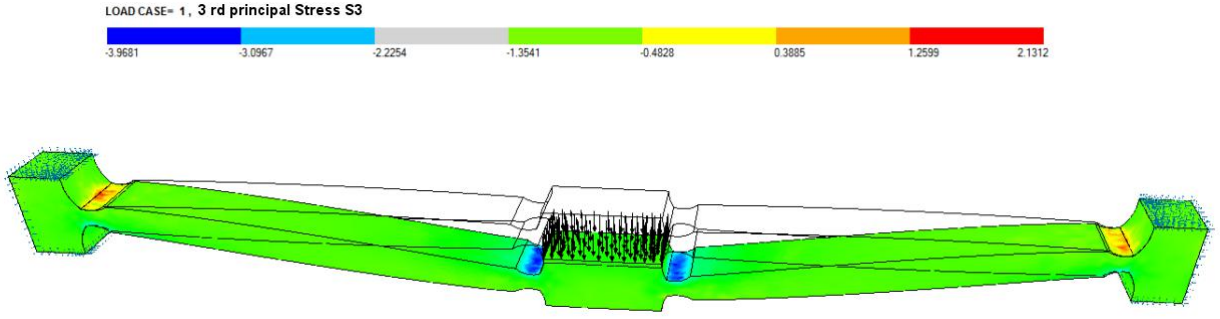
max. v.Mises-Stress = 1.98 MPa



max. S1 = 3.7 MPa

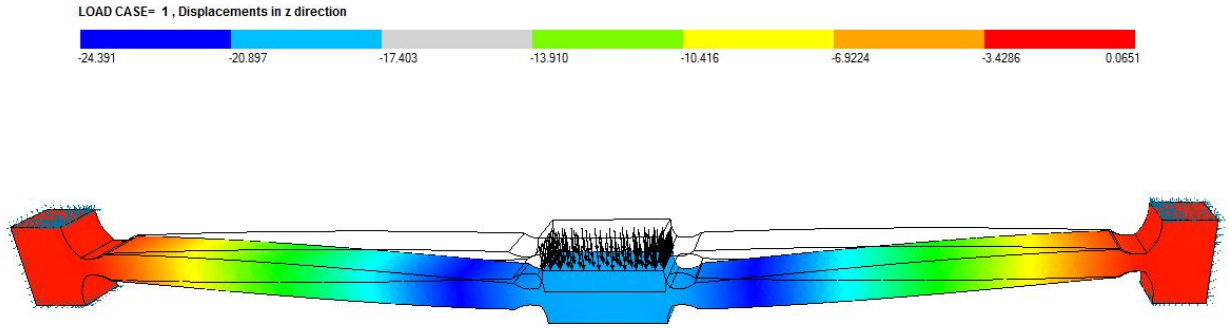


max. S3 = -3.9 Mpa

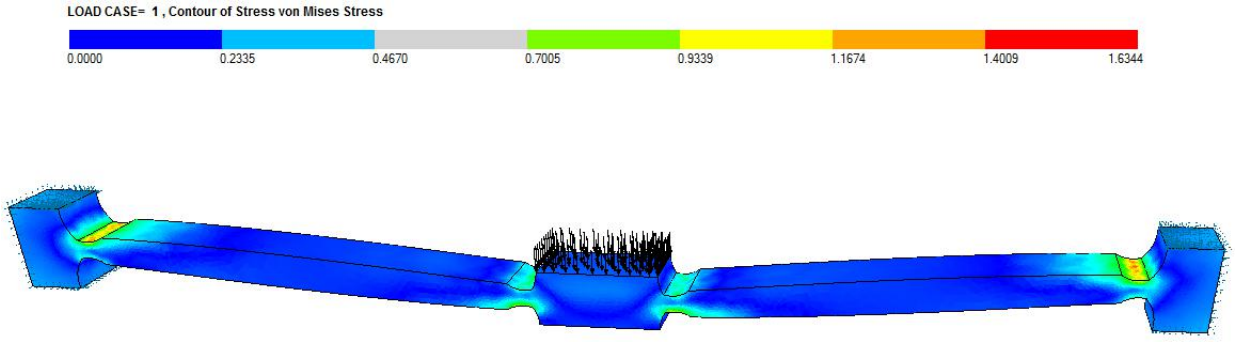


Nonlinear Results:

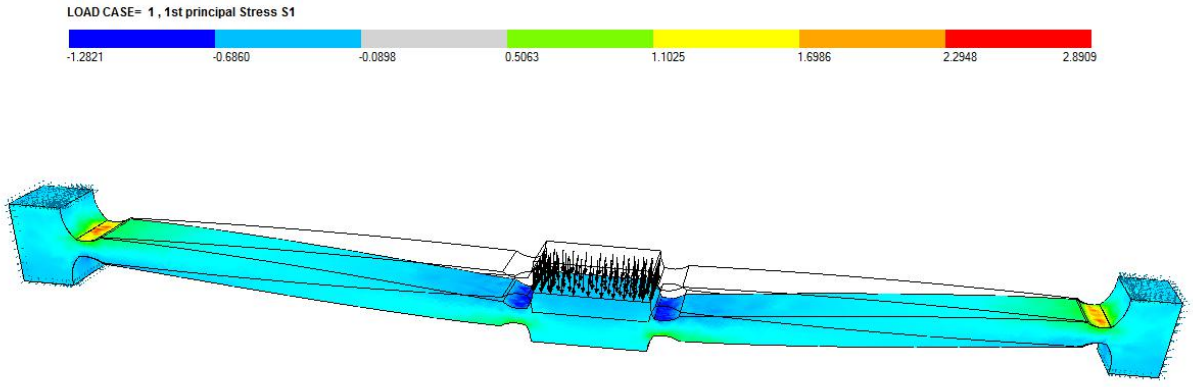
max. Displacement = - 24.39 mm



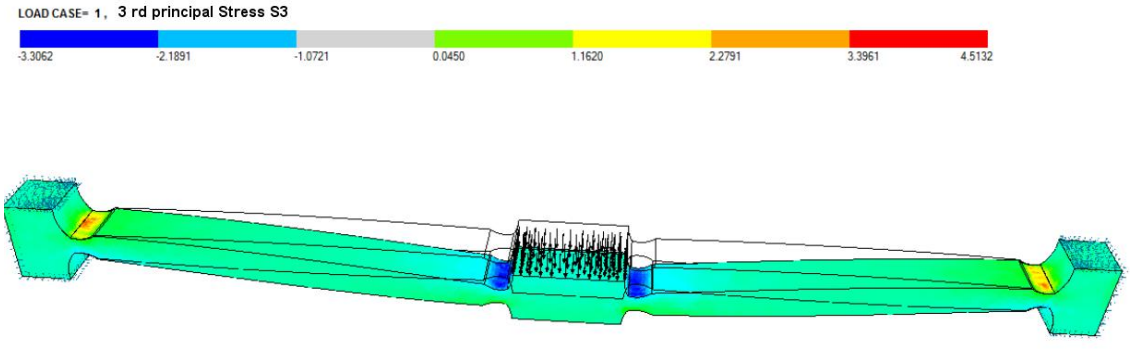
max. v.Mises-Stress = 1.63 MPa



max. S1 = 2.89 MPa

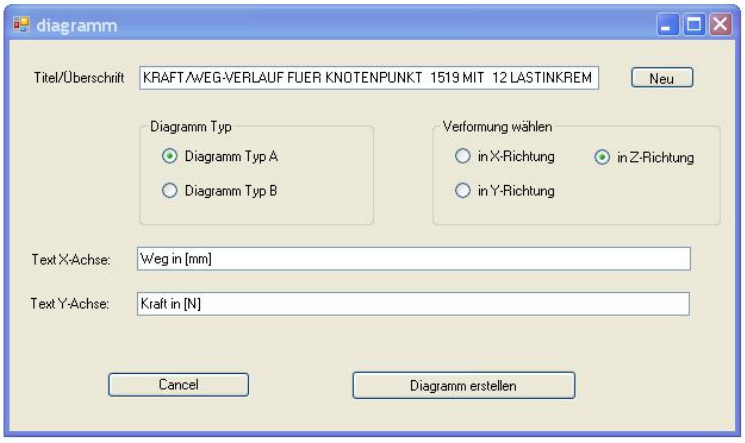


max. S3 = - 3.3 Mpa



Show Diagramm with Gnuplot

Choose menu „Show Diagramm with GNUPLOT“ and edit the titel or the name for the x- and y-axis. Choose “Start Diagramm” to start gnuplot.exe.



Choose „Open“ and select the File „Diagram.plt“ to show the displacments and the loads for node 1519 in a diagram.