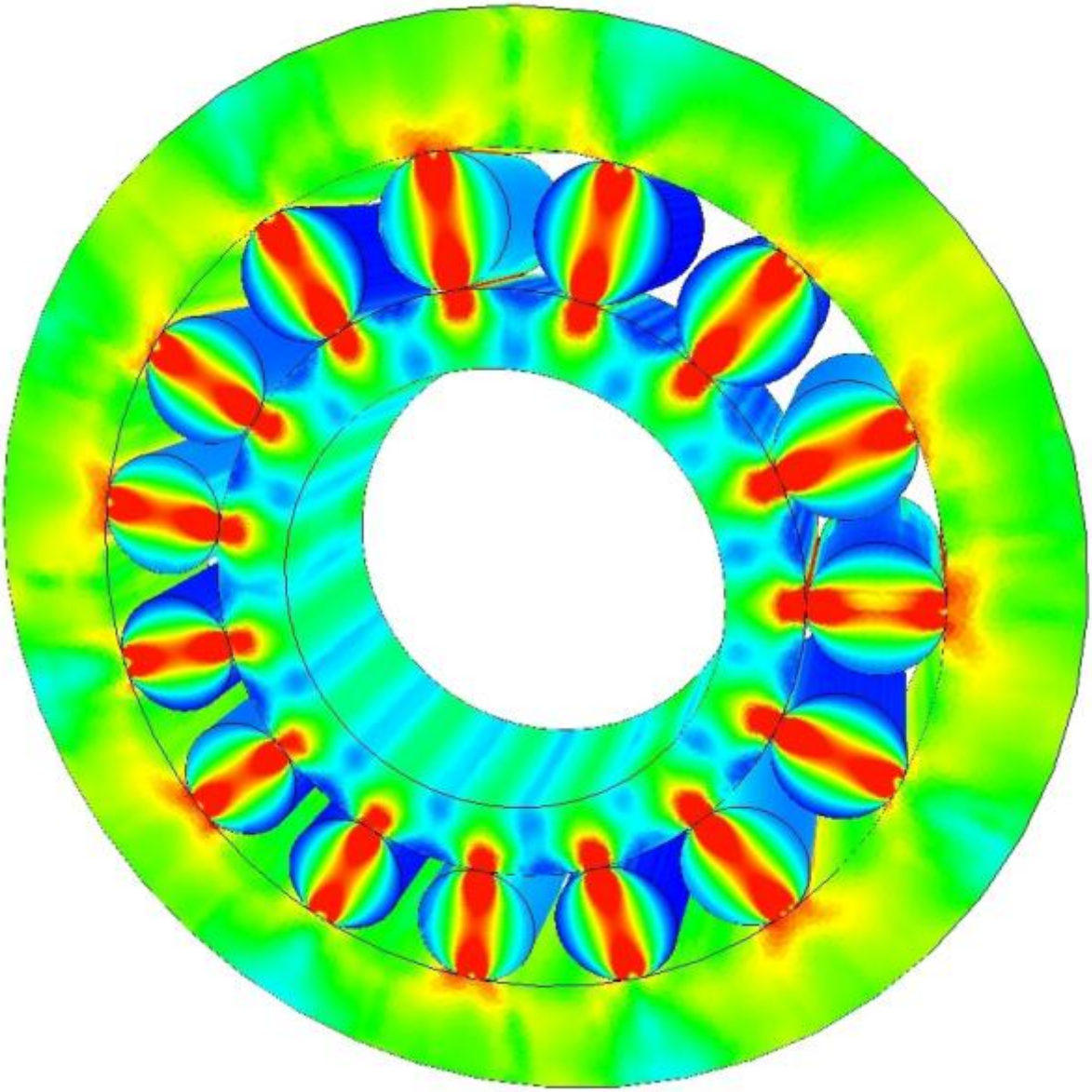
FEM-System MEANS V11

Rolling Contact of a Rail-Wheel-Model
Contact-Pressure of Bearings



(C) 2018 by HTA-Software

www.fem-infos.com

Tel. 0049-7844-98641 Germany

Part 7: Contact-Analysis mit MEANS V11

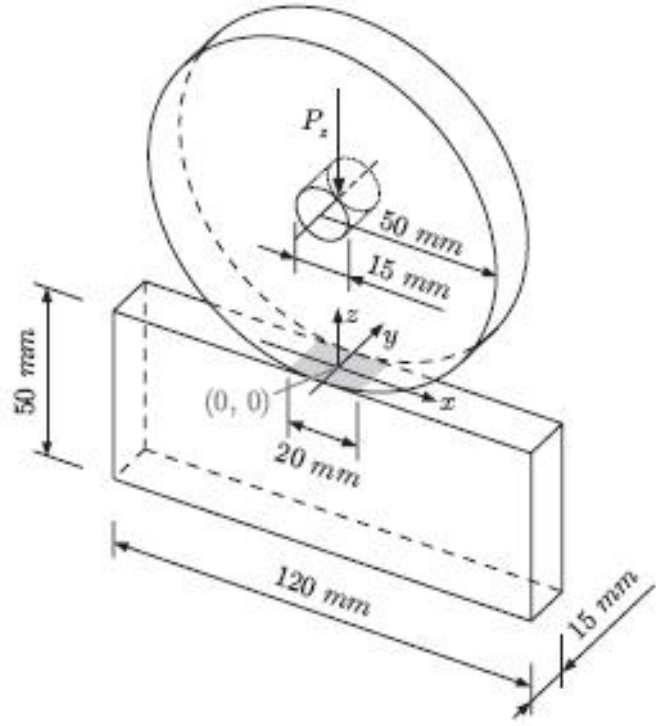
A wheel with an outer radius of 50 mm and an inner radius of 7.5 mm is pressed with a force of 214.2 KN perpendicular to a 120 mm railway track. Wheel and track are made of the steel S900A with a yield strength of 900 N/mm².

What is rolling contact

The recurrent exposure to high contact pressure tires the tracks by microstructural changes so far that fissures (Heat check cracks) occur. The phenomenon of rolling contact fatigue (Rolling Contact Fatigue RCF) is a significant cost factor for the manufacturer of train and rail systems, ball bearings or bolt clamped components.

Calculation of the Hertzian contact pressure

The contact surface between wheel and rail is 15 mm wide and 20 mm long.



Geometry			
Radius Cylinder	R_{Zyl}	= 50.00	[mm]
Width contact surface	b_{KF}	= 15	[mm]
Material			
Young's modulus E	E_{Mat}	= 210000.00	[N/mm²]
Load F prescribed	F_k	= 214200	[N]
Results			
semi contact widht	a	$= 1.520 \cdot (F_k \cdot r_{zyl} / (b_{kf} \cdot E_{Mat}))^{(1/2)}$ = 2.803	[mm]
max. contact pressure	σ_0	$= 0.418 \cdot (F_k \cdot E_{Mat} / (b_{kf} \cdot r_{Zyl}))^{(1/2)}$ = 3237.166	[N/mm²]

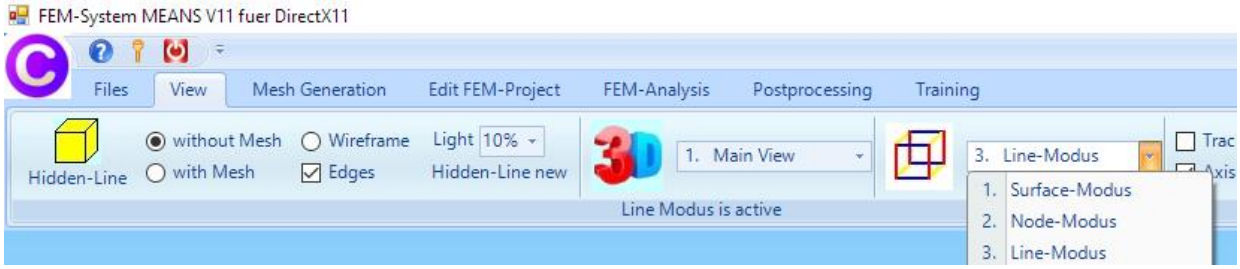
Using an online calculator it is calculated a contact pressure of 3237.166 N / mm ².

How big are the contact pressure with the FEM-System MEANS V11 with the add-on module CONTACT from www.fem-infos.com

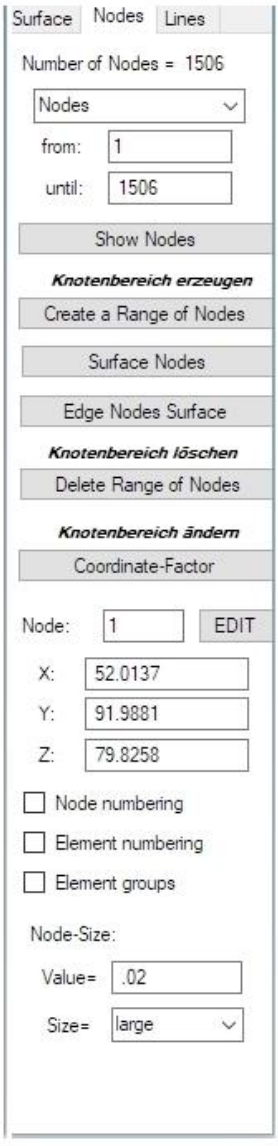
Mesh Generation with MEANS V11

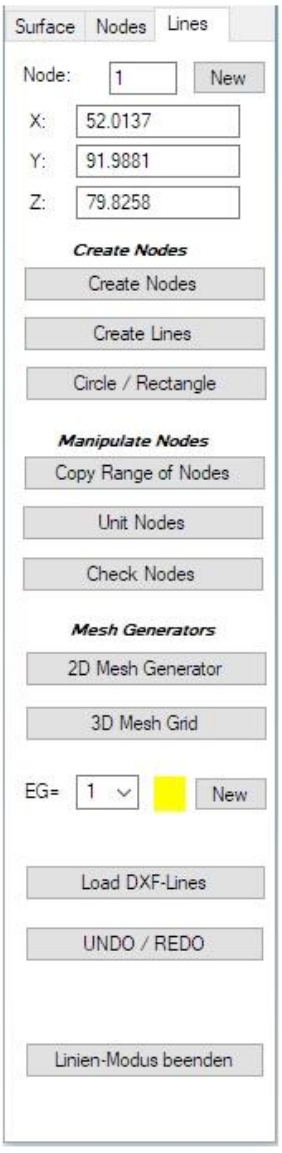
In MEANS V10, the model had to be created with AutoCAD and via the DXF interface. Now with MEANS V11 it is possible to create the complete model in just a few steps in the Lines- and Nodes-Mode.

With MEANS V11, the "View" and "3. Line-Modus" tabs are first selected to enter a circle and a rectangle with 2 element groups and 2 mesh refinements.

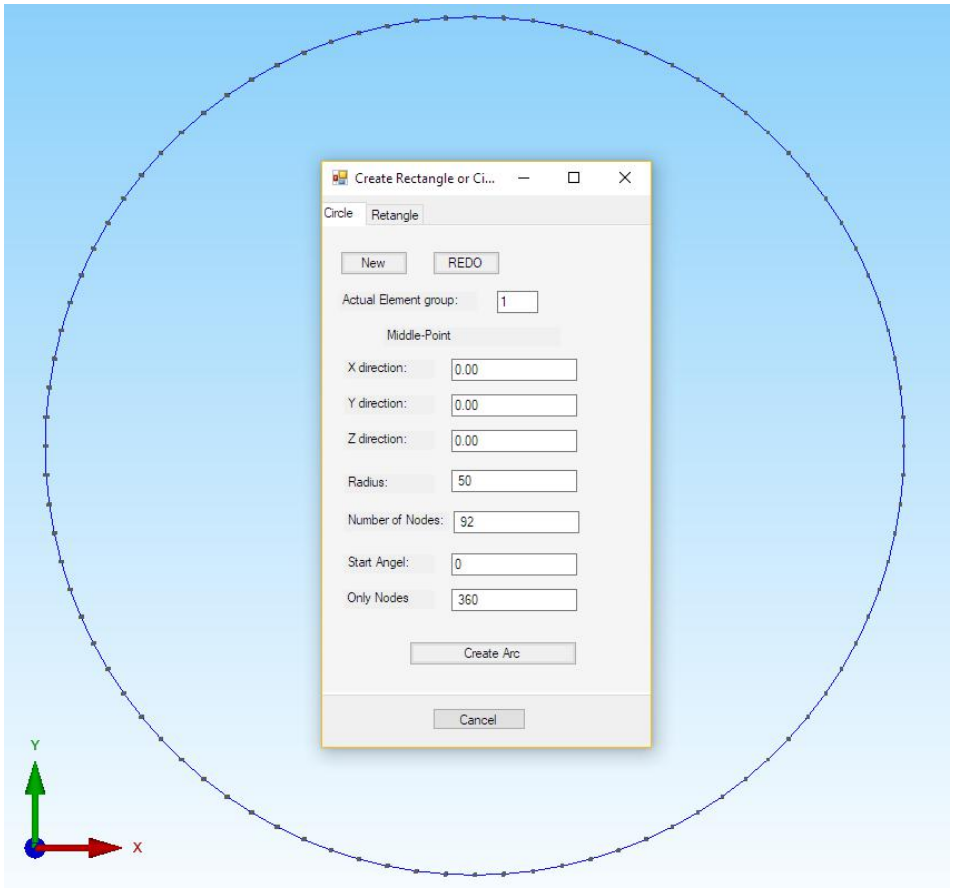


During the input you can switch between Nodes- and Lines-Modus.

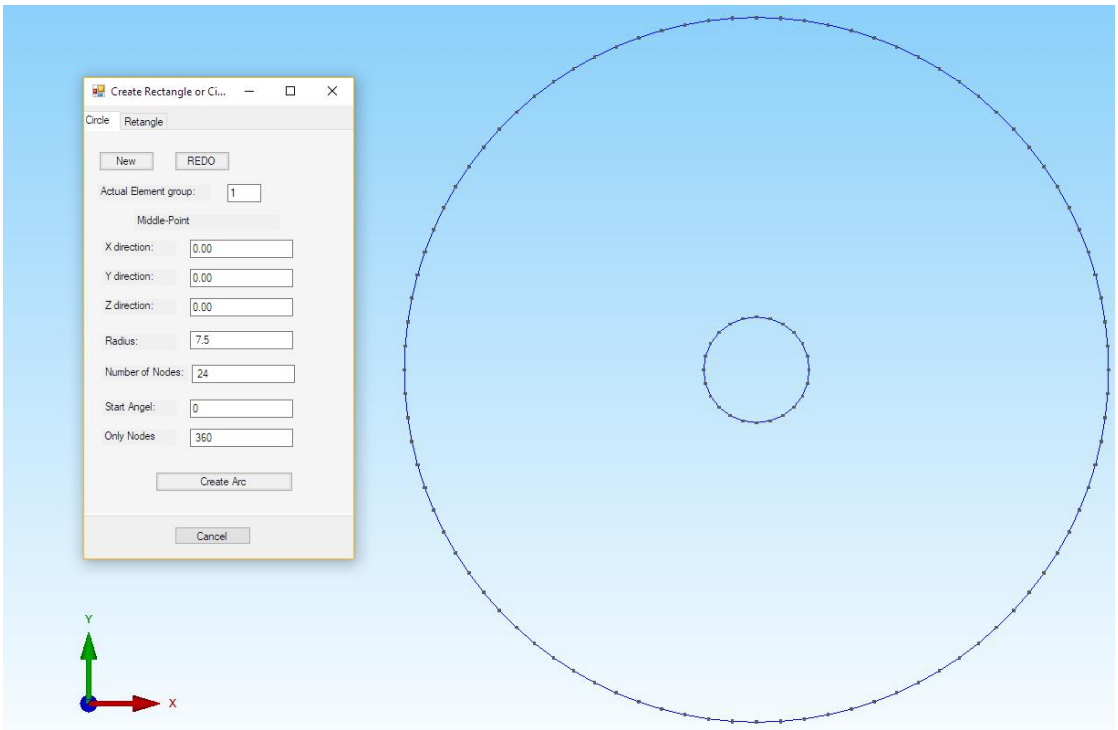




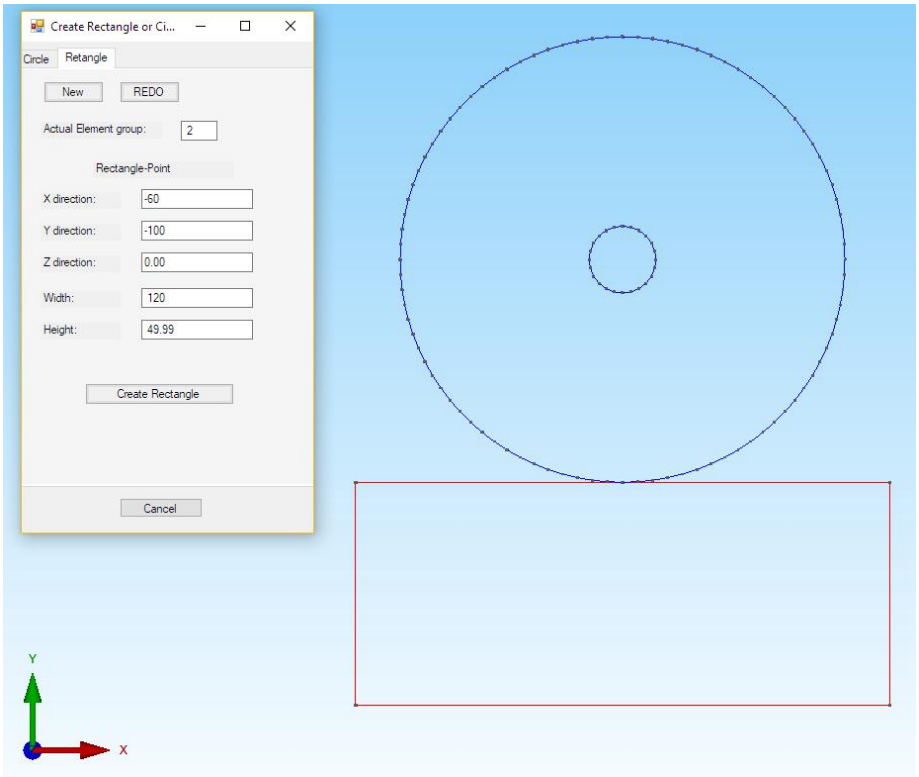
Enter a full circle with a radius of 50 mm at the zero point. Also, a fine raster of 92 nodes is chosen so that the circle is as round as possible and not too square. Otherwise you get too high a contact pressure at the corners and edges instead of a desired contact pressure is evenly distributed over the surface.



Repeat to create a circle with radius 7.5 mm and 24 nodes.

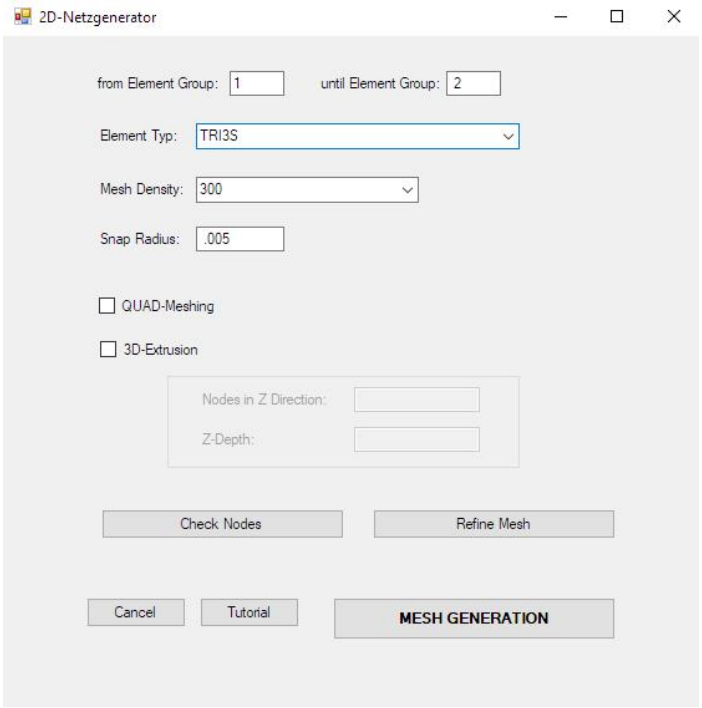


Now create the 2 element group with a rectangle of width 120 mm and height 50 mm. However, it is given a height of 49.99 mm so that a distance of 0.01 mm is between circle and rectangle, otherwise the solver would find no contact nodes and cancel the analysis.

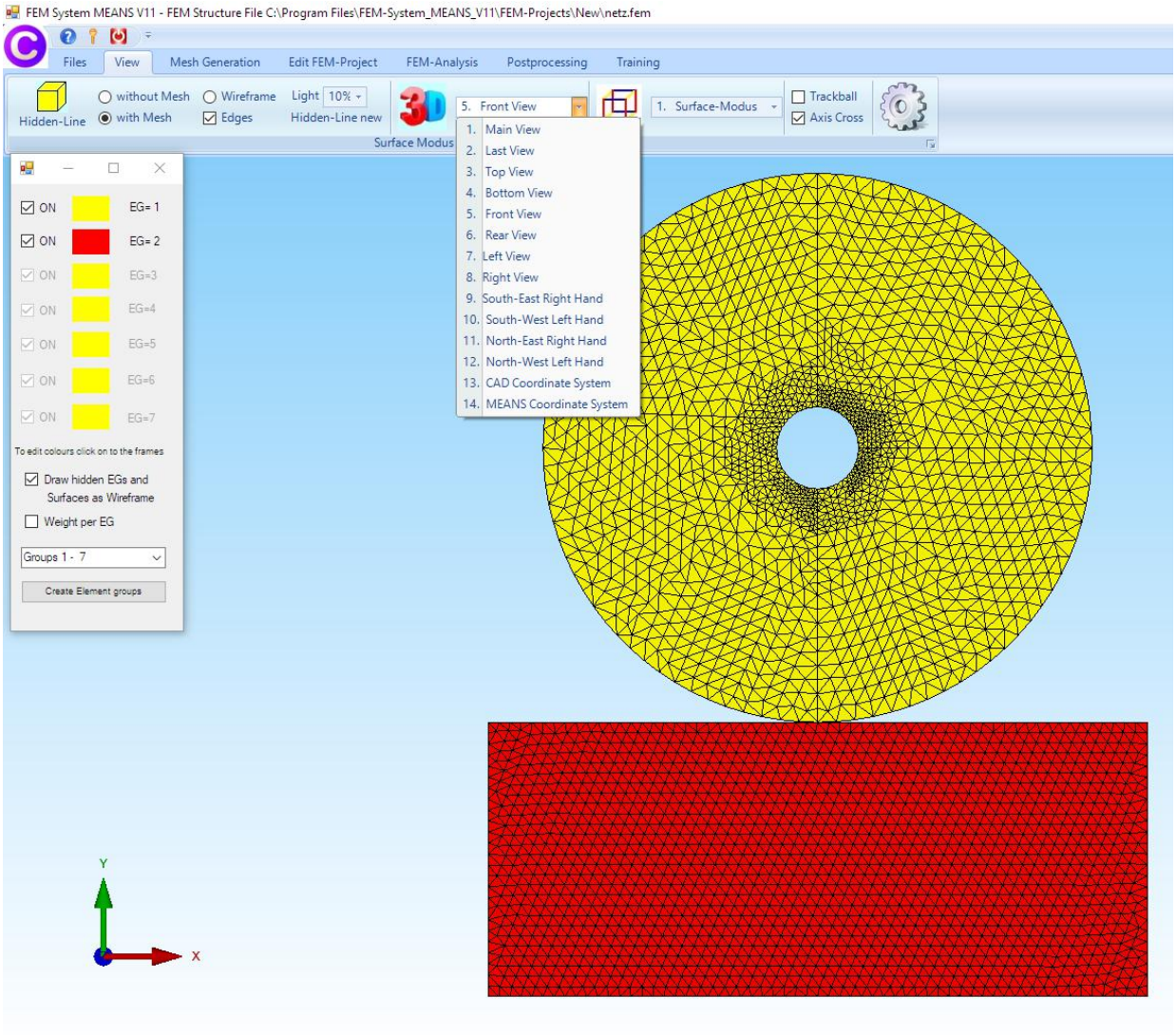


Mesh generation without mesh refinement

Now save the line model under any name and select menu "2D mesh generator" to generate a mesh without mesh refining and extruding:



Select the "Front View" view so that the 2D mesh can be seen in the XY plane, then choose the "Edit FEM Project" and "Element Groups" tabs to give each element group a color.

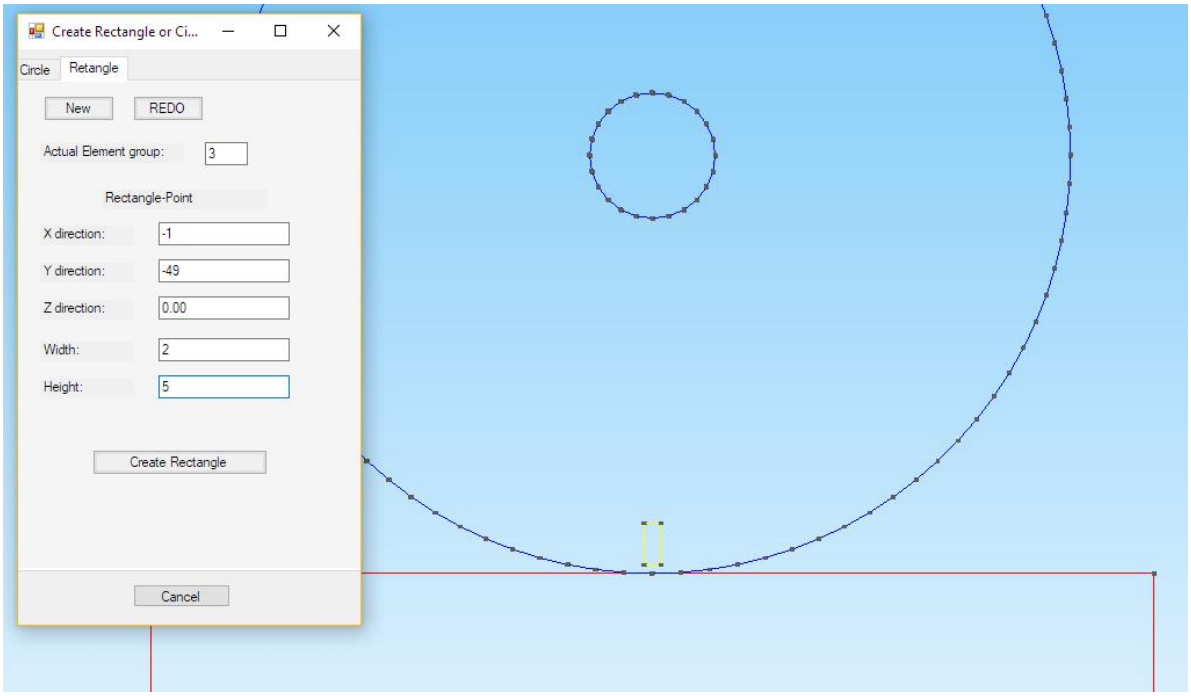


Mesh generation with mesh refinement

The mesh for a contact analysis is not fine enough. Therefore, the line model must be extended with two rectangles for mesh refinement.

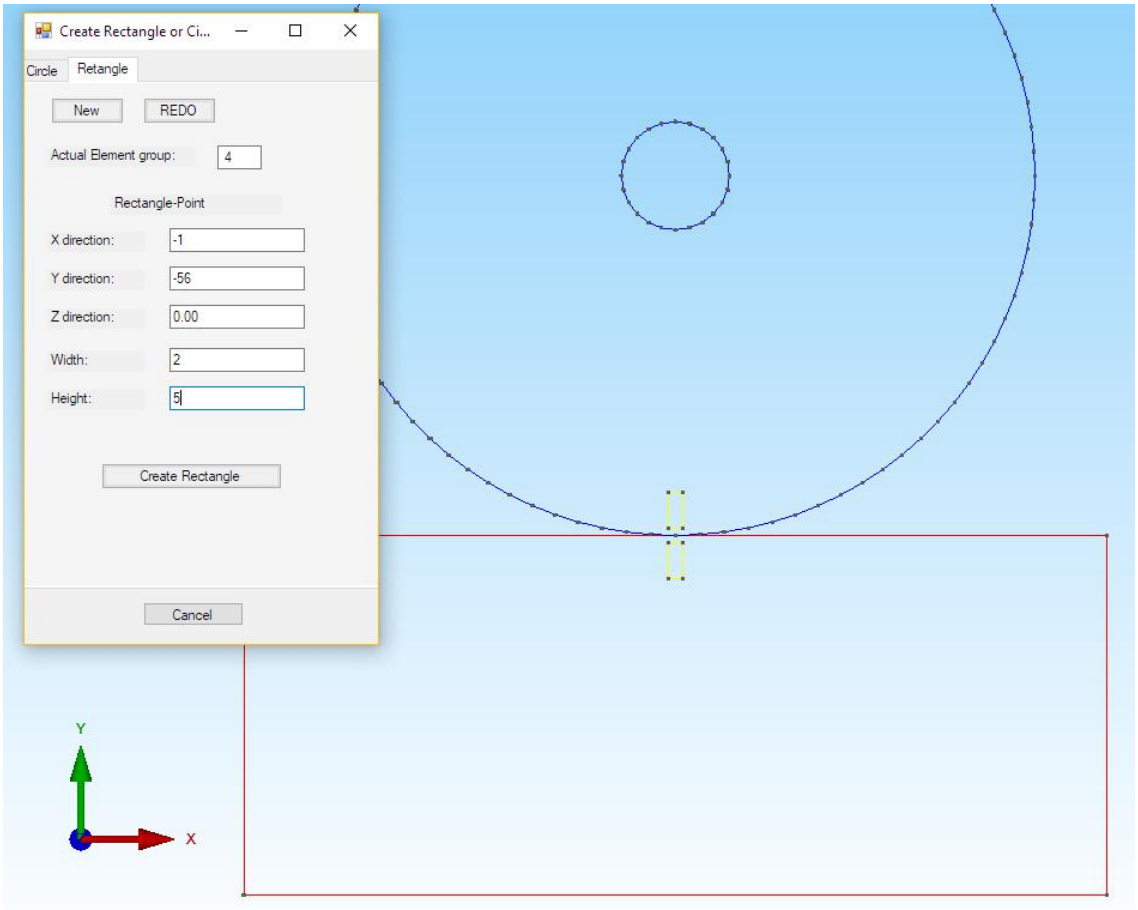
Load the line model again and select "Rectangle / Circle" and create the following rectangle

Element group = 3
Starting point in X direction = - 1 mm
Starting point in Y direction = - 45 mm
Width = 2 mm and Height = 5 mm

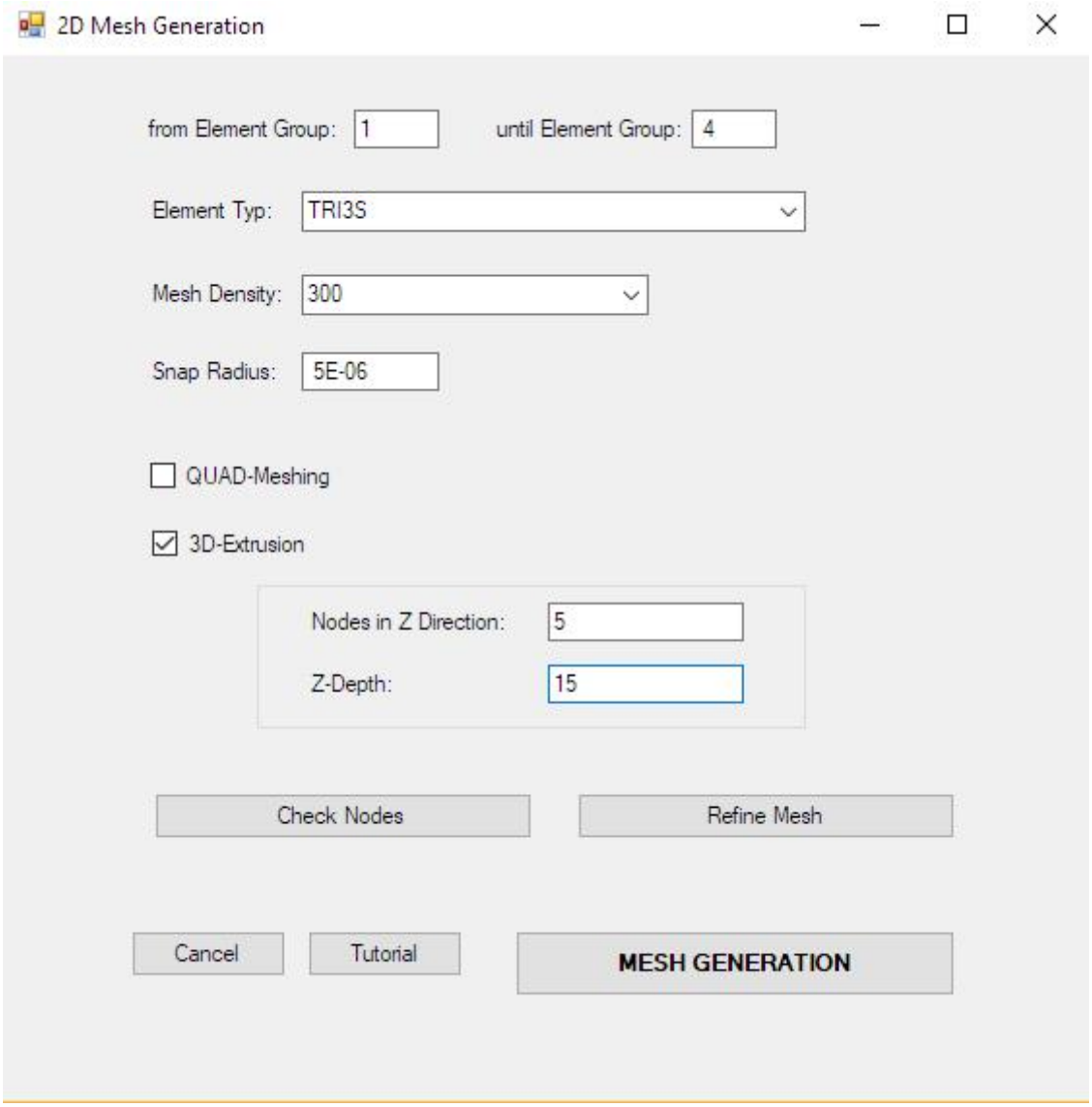


Then select "Circle / Rectangle" again and create the following rectangle with the width 2 mm and height 5 mm with the element group 4:

Element group = 4
Starting point in X direction = - 1 mm
Starting point in Y direction = - 56 mm
Width = 2 mm and Height = 5 mm



In Line-Mode, select the "2D Mesh Generator" and "Refine Mesh" menu



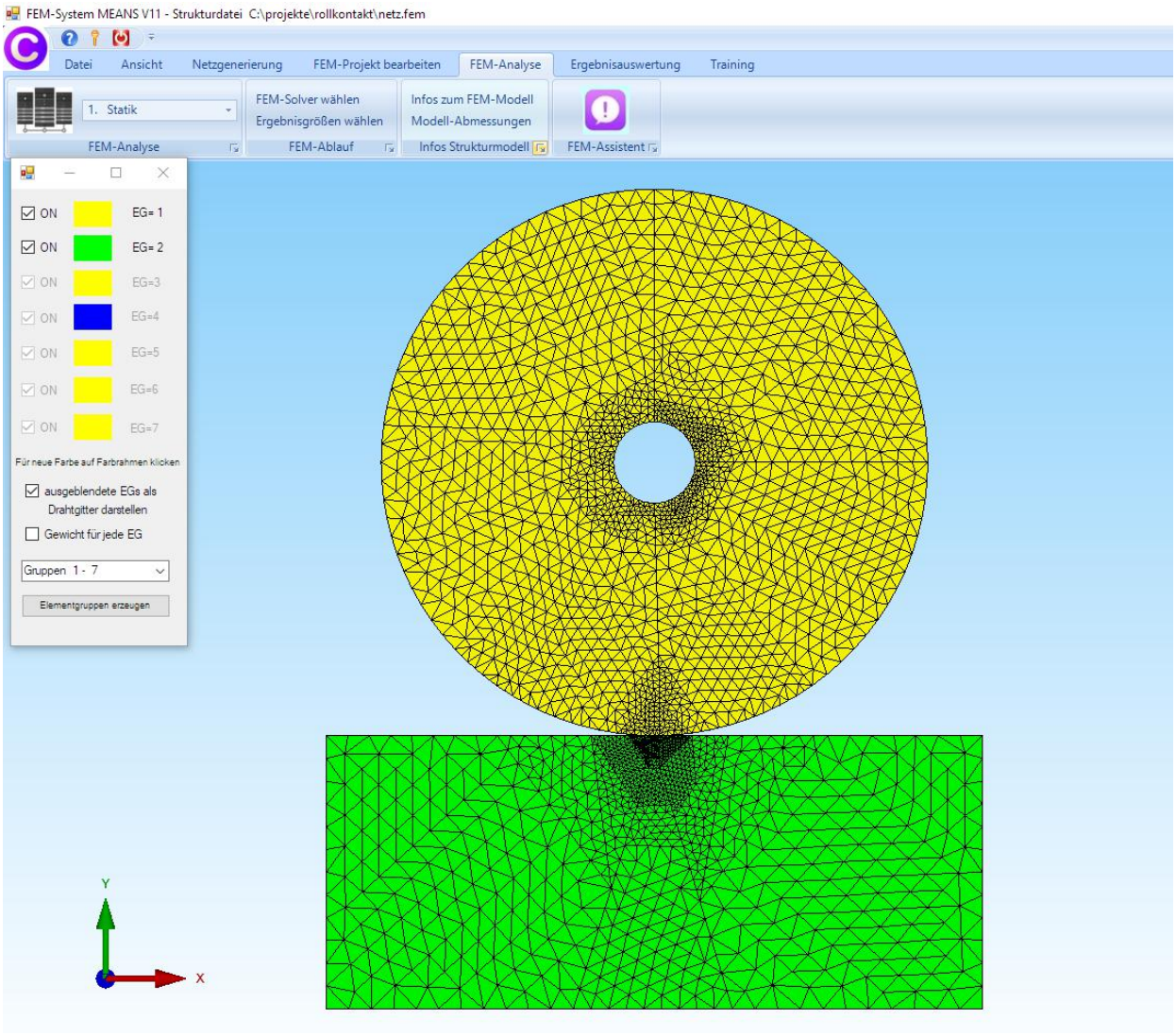
and enter the following 4 lines for controlling the mesh generation:

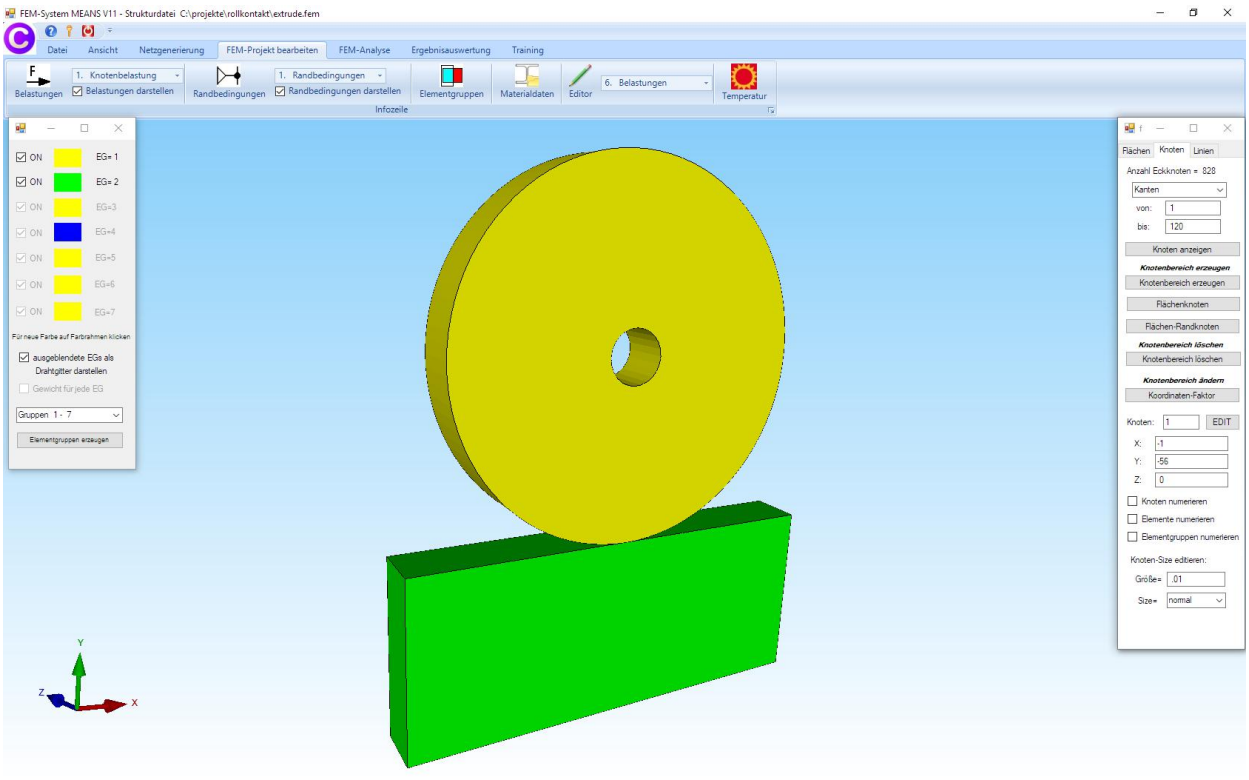
Element Group	Main Group	Subgroup	Refine	Holes	Mesh Density
1	1	0	0	0	300
2	2	0	0	0	200
3	0	1	1	0	0
4	0	2	1	0	0

also activate “3D-Extrusion” and enter:

Number of nodes in Z direction = 5
Z - Depth = 15 mm

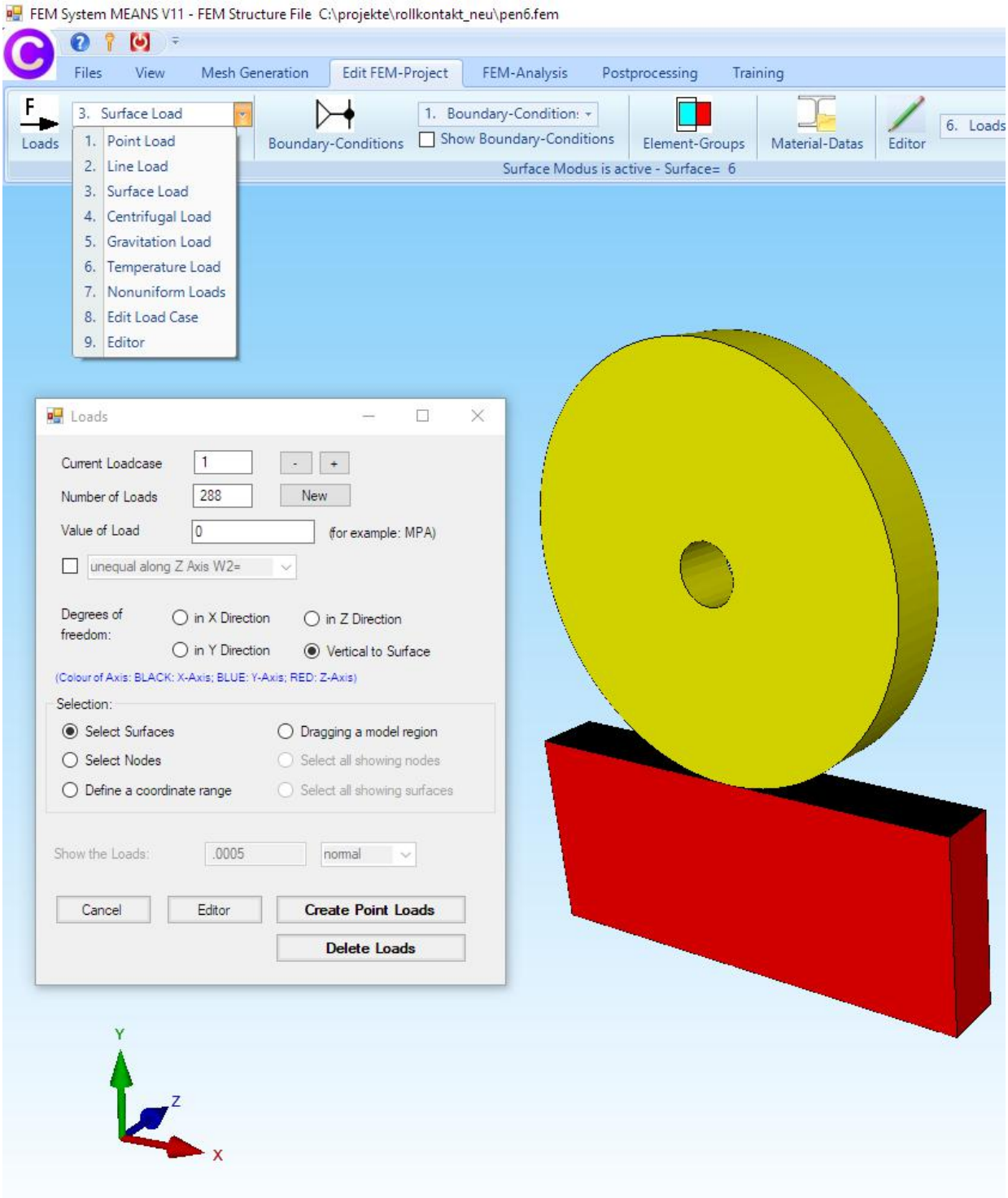
Then select the menu "MESH GENERATION" to generate a FEM structure with 20 476 PEN6 pentahedron elements and 13 565 nodes in one step with a very fine mesh density at the contact surfaces.


FEM-System MEANS V11 - Strukturdatei C:\projekte\rollkontakt\netz.fem
Datei
Ansicht
Netzgenerierung
FEM-Projekt bearbeiten
FEM-Analyse
Ergebnisauswertung
Training
1. Statik
FEM-Solver wählen
Ergebnisgrößen wählen
Infos zum FEM-Modell
Modell-Abmessungen
FEM-Analyse
FEM-Ablauf
Infos Strukturmodell
FEM-Assistent
ON EG= 1
ON EG= 2
ON EG=3
ON EG=4
ON EG=5
ON EG=6
ON EG=7
Für neue Farbe auf Farbrahmen klicken
ausgeblendete EGs als Drahtgitter darstellen
Gewicht für jede EG
Gruppen 1 - 7
Elementgruppen erzeugen
Y
X


FEM-System MEANS V11 - Strukturdatei C:\projekte\rollkontakt\extrude.fem
Datei
Ansicht
Netzgenerierung
FEM-Projekt bearbeiten
FEM-Analyse
Ergebnisauswertung
Training
Belastungen
1. Knotenbelastung
Belastungen darstellen
Randbedingungen
1. Randbedingungen
Randbedingungen darstellen
Elementgruppen
Materialdaten
Editor
6. Belastungen
Temperatur
Infozeile
Flächen
Knoten
Linien
Anzahl Eckknoten = 828
Kanten
von: 1
bis: 120
Knoten anzeigen
Knotenbereich erzeugen
Knotenbereich erzeugen
Flächenknoten
Flächen-Randknoten
Knotenbereich löschen
Knotenbereich löschen
Knotenbereich ändern
Koordinaten-Faktor
Knoten: 1
EDIT
X: -1
Y: -56
Z: 0
Knoten numerieren
Elemente numerieren
Elementgruppen numerieren
Knoten-Size editieren:
Größe= .01
Size= normal
Y
Z
X

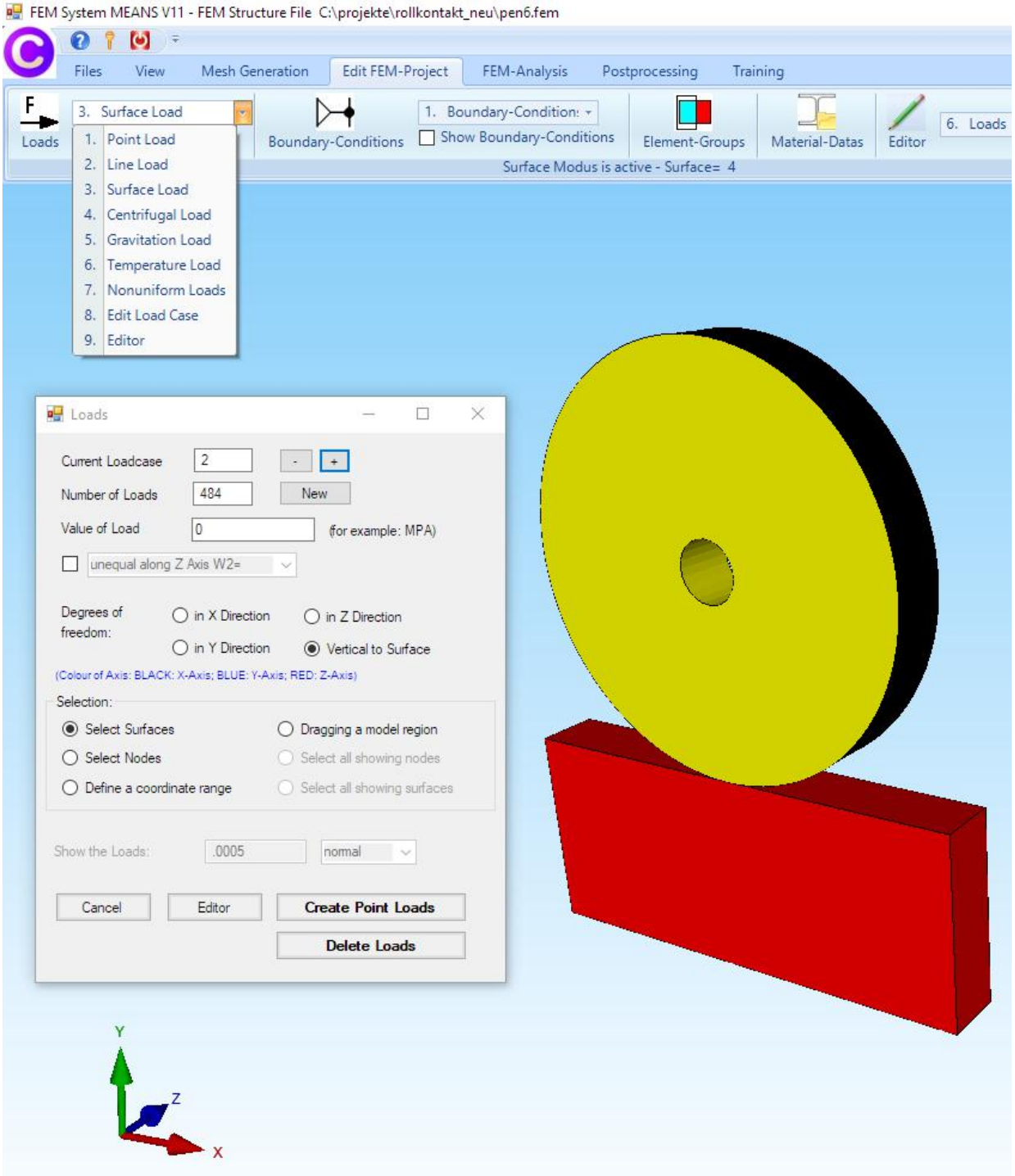
Load Case 1 with a Master contact surface

In the contact analysis, the Master contact surface is always load case 1 with a surface load and with a Value of Load “0”. Select "Edit FEM-Project" tab and "3. Surface Load" and click on surface 6 for the Master contact surface.



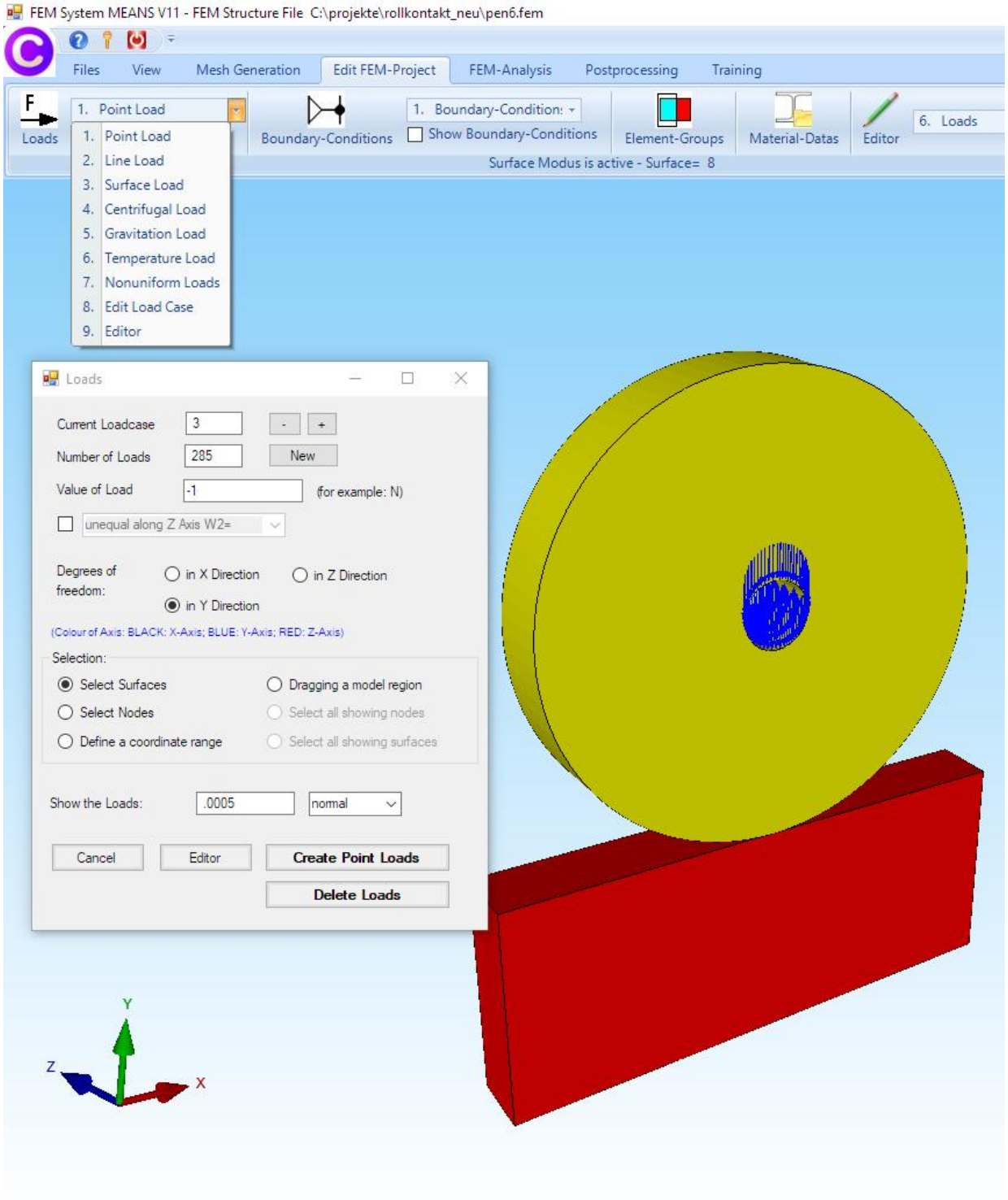
Load Case 2 with a Slave contact surface

In the contact analysis, the Master contact surface is always load case 2 with a surface load and with a Value of Load “0”. Select "Edit FEM-Project" tab and "3. Surface Load" and click on surface 4 for the Slave contact surface.

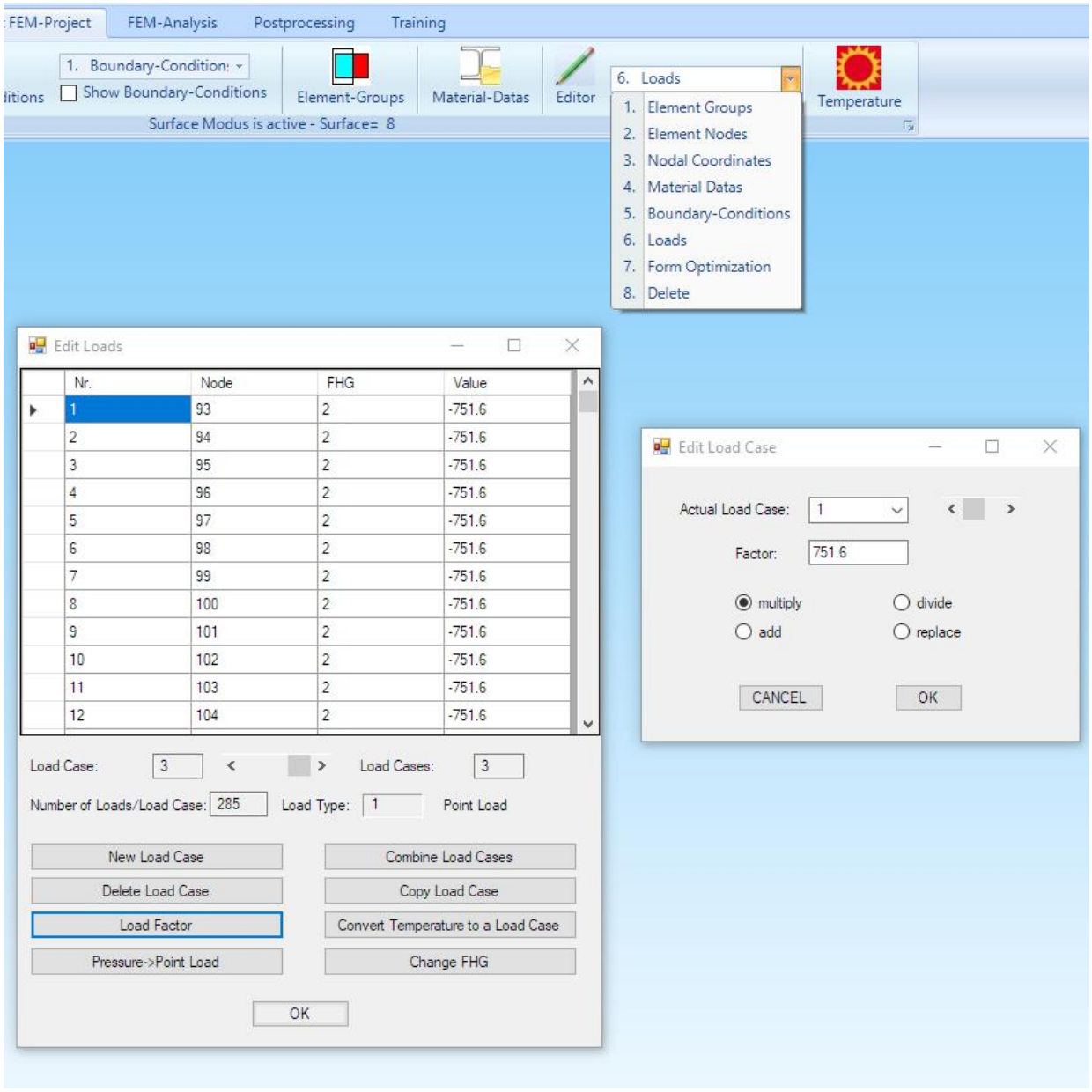


Load Case 3 with a Point Load

In contact analysis, Load Case 3 is always the actual load. A central weight load of -214 200 N in the Y direction must be entered.
Select "Edit FEM-Project" and "1. Point Load" and create a Point Load in Y-direction with a Value of Load "-1" by clicking on surface 8 of the inner circle.



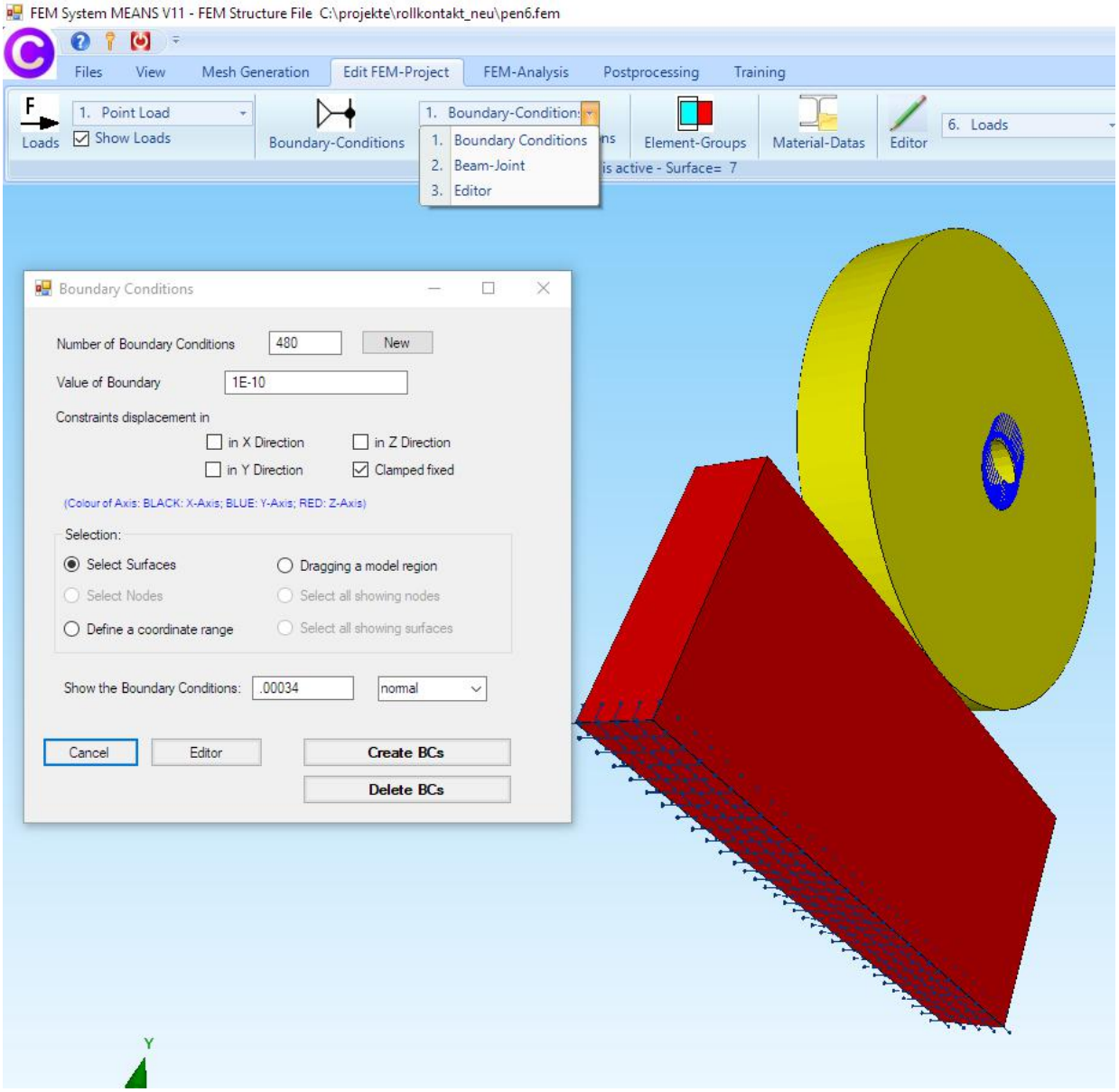
A Point Load with 285 node values and the load value -1 was generated. Now select "Editor" and "6. Loads " and multiply the loads from load case 3 by the load case factor 214 200 / 285 = 751.6.



Create Boundary Conditions

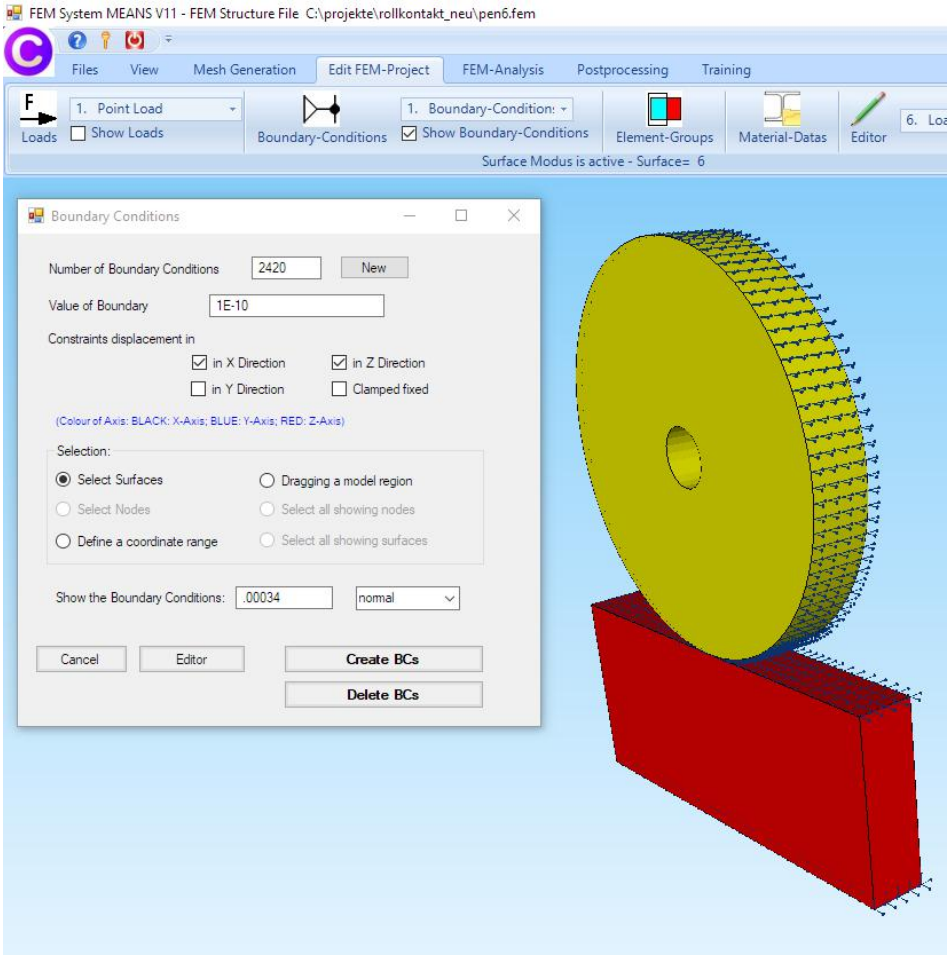
Clamped fixed

Select “Edit FEM-Project” and "Boundary Conditions" and clamped fixed surface 7 on the bottom.



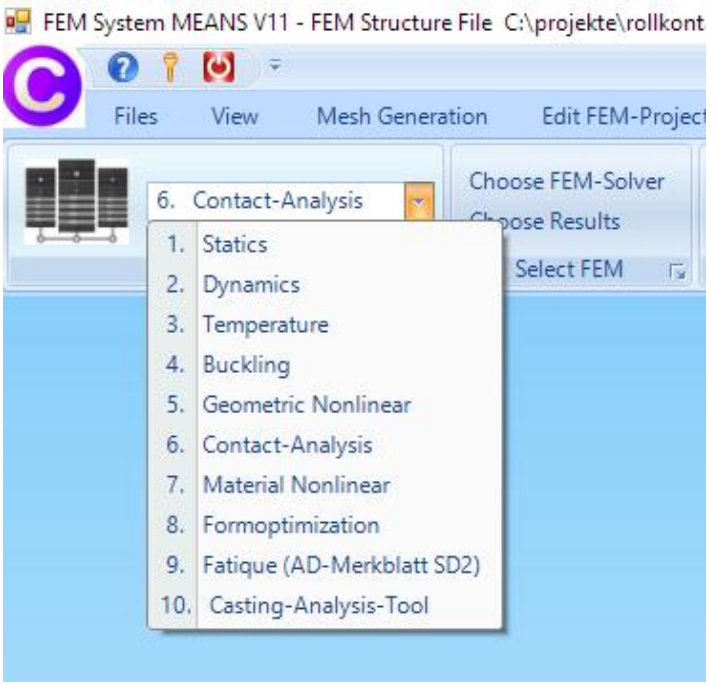
Boundary conditions in Z direction

In addition, surfaces 4 and 6 must not move in the Z direction.



FEM Analysis

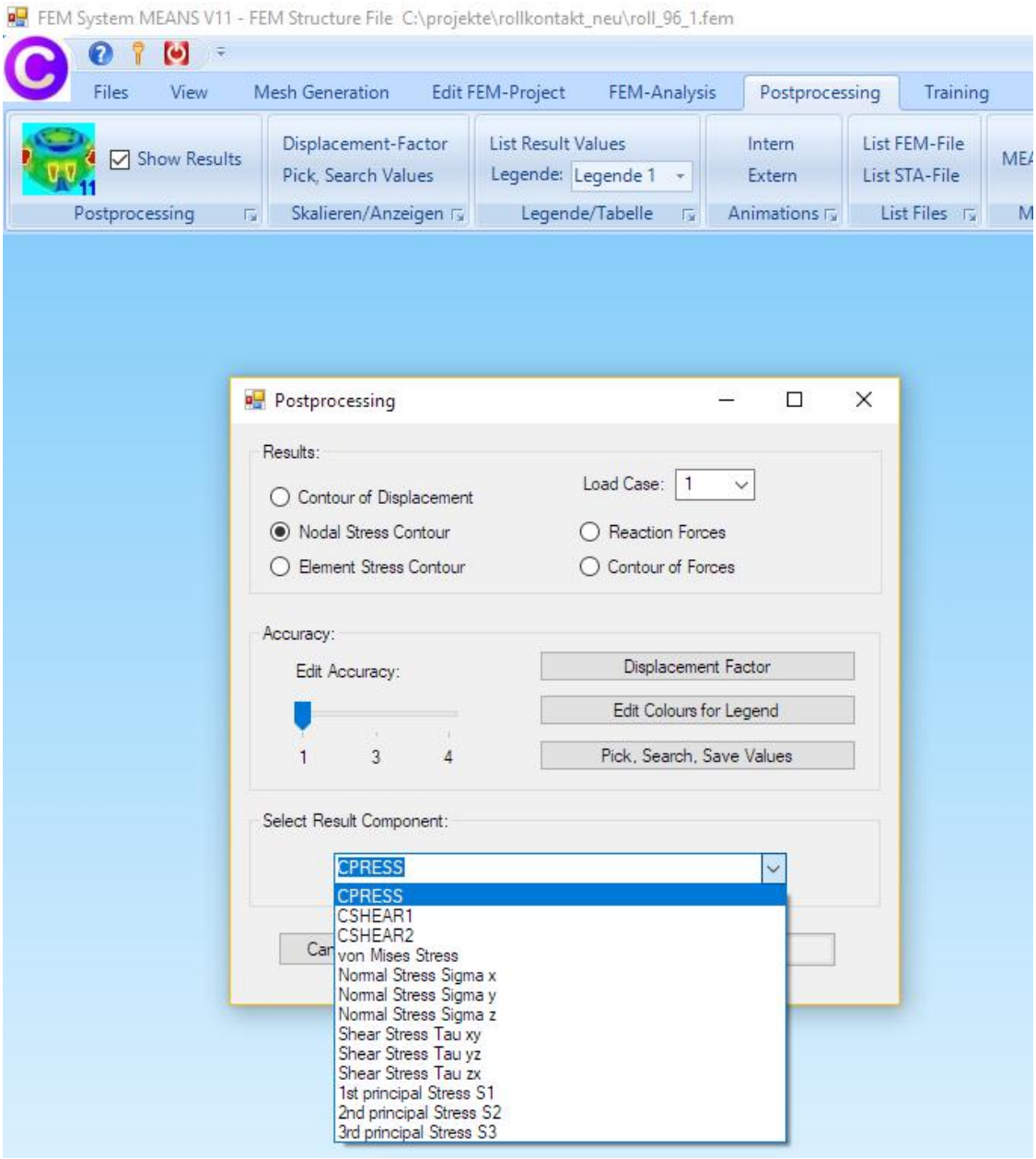
Select the "FEM Analysis" tab and "6. Contact Analysis" to start the contact analysis. Here also the master and slave contact surface can be exchanged.



Postprocessing

Select the "Postprocessing" tab and click on the icon to evaluate the results of contact analysis, these include

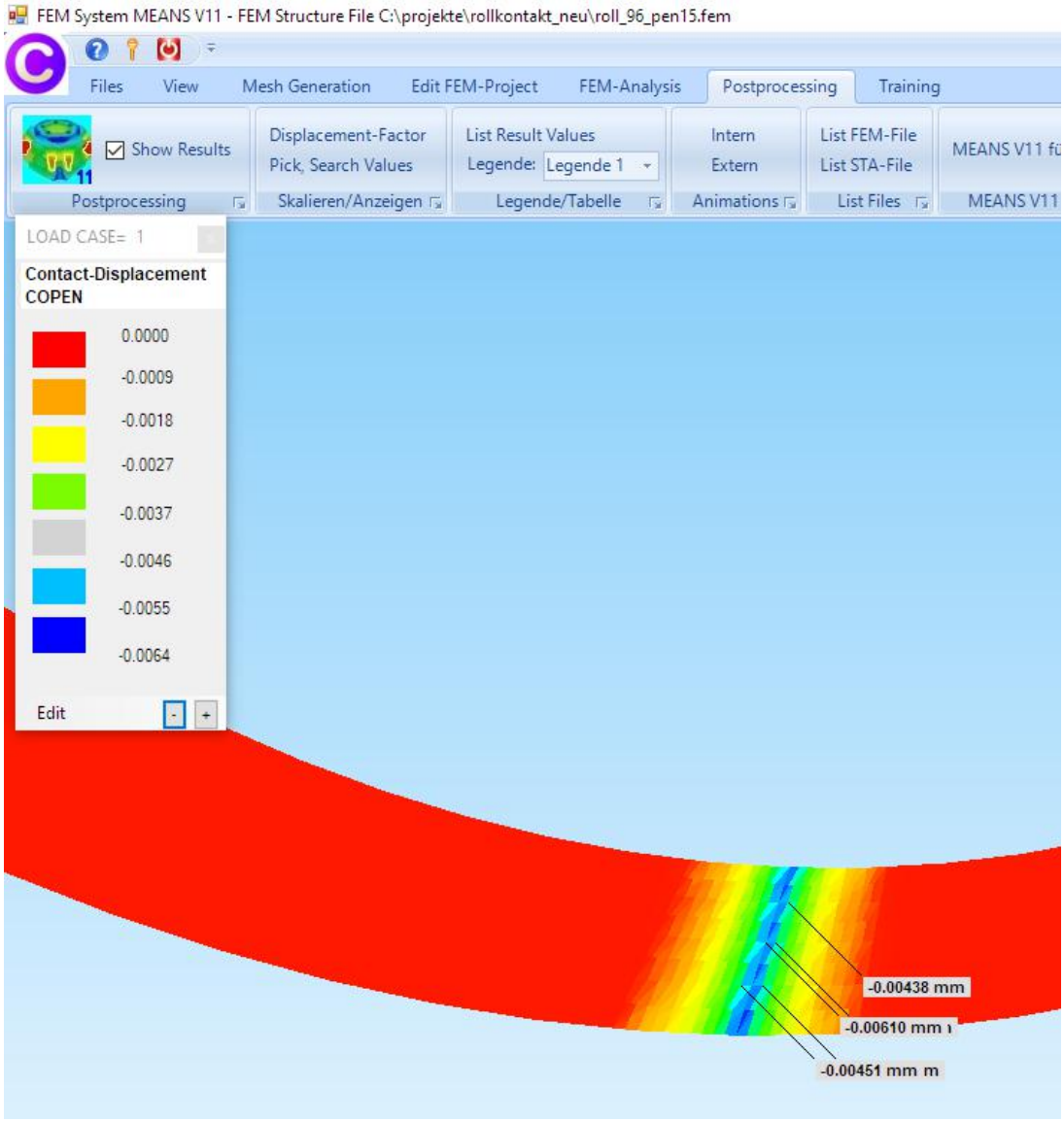
COPEN = Contact Displacements on the Slave surface
CPRESS = Contact Pressure on the Slave surface



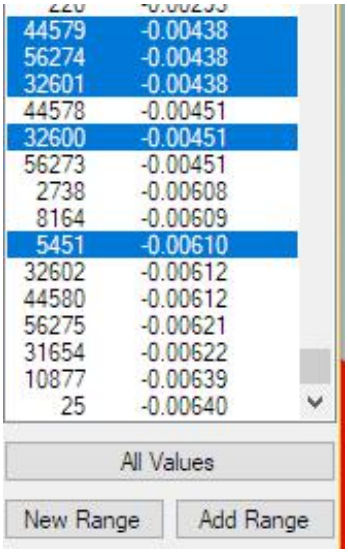
Select in the Surface-Mode menu “Show only Surfaces” to show only the Slave contact surface 4.

Select the icon to display the contact displacements COPEN on the slave contact surface.

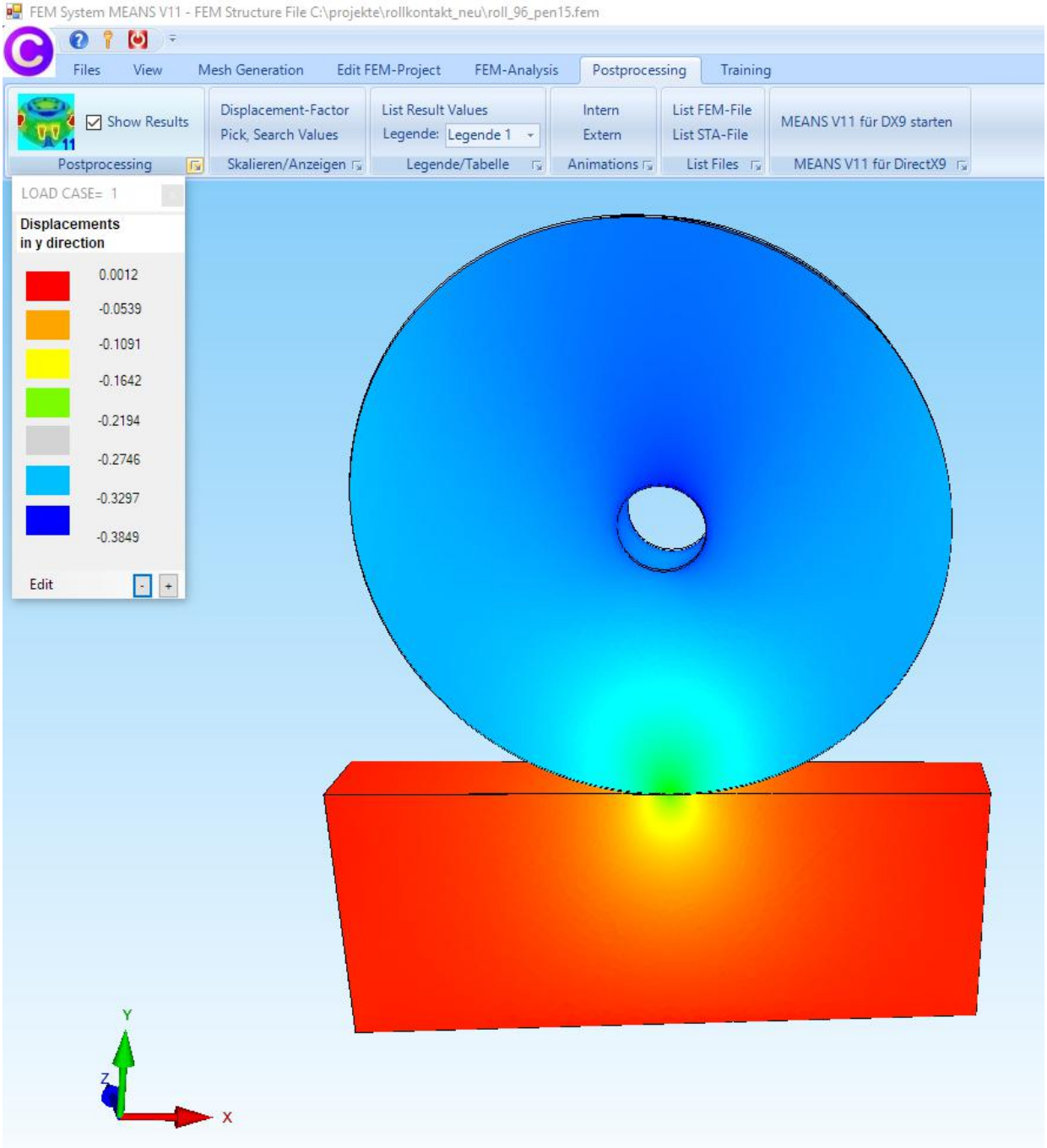
Select "Pick Search Values" and click on the listed displacements, if marked in blue, directly display the value with a line at the nodes.



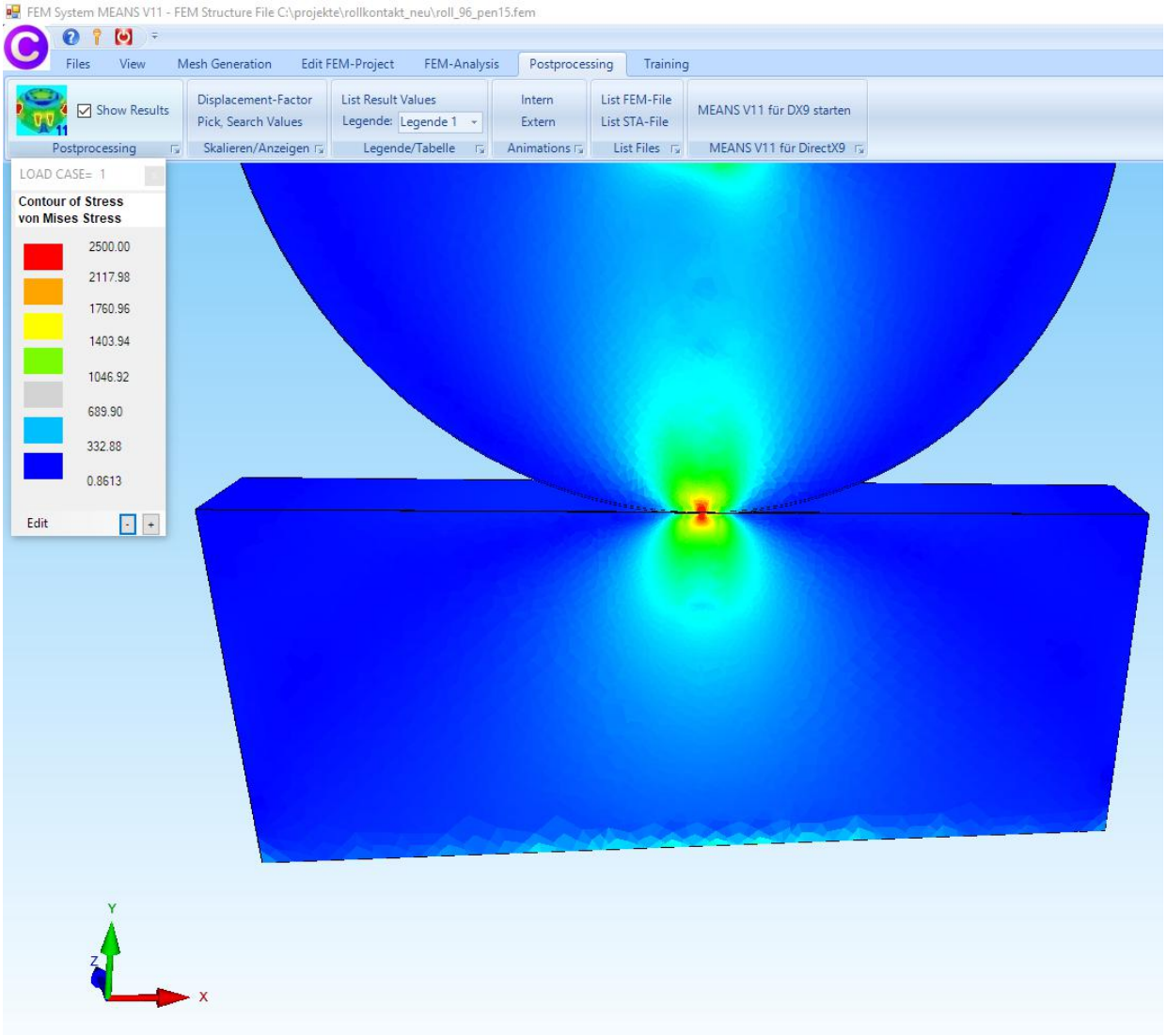
Pick, Search Result Values:



Select in the Surface-Mode menu “Show all Surfaces” and choose the icon to display the displacements in y direction.

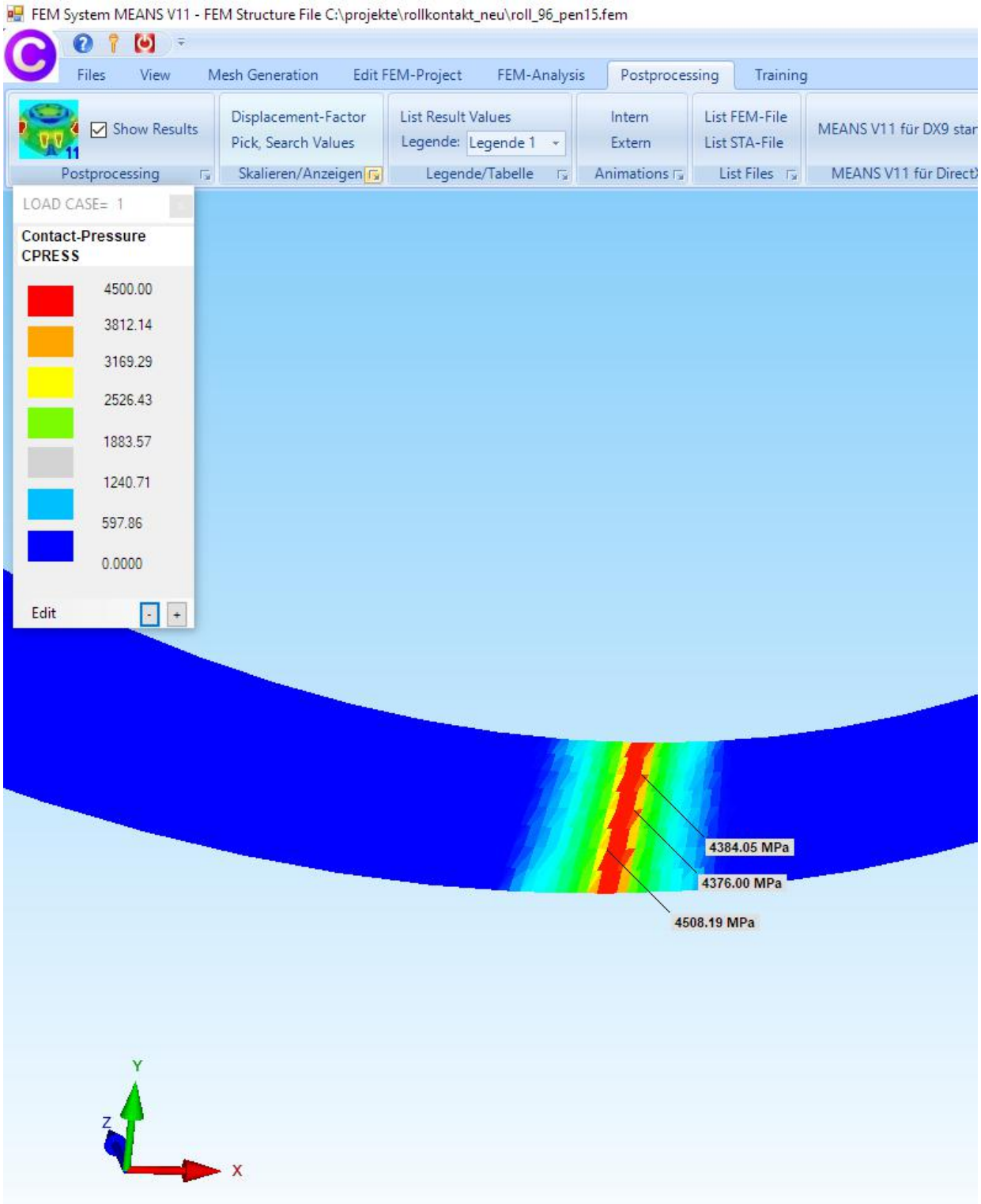


Select the Icon to display the v.Mises-Stress



Select in the Surface-Mode menu “Show only Surfaces” to show only the Slave contact surface 4.

Select the icon to display the contact pressure CPRESS on the slave contact surface.


FEM System MEANS V11 - FEM Structure File C:\projekte\rollkontakt_neu\roll_96_pen15.fem
Files
View
Mesh Generation
Edit FEM-Project
FEM-Analysis
Postprocessing
Training
Show Results
Displacement-Factor
Pick, Search Values
List Result Values
Legende: Legende 1
Intern
Extern
List FEM-File
List STA-File
MEANS V11 für DX9 star
Postprocessing
Skalieren/Anzeigen
Legende/Tabelle
Animations
List Files
MEANS V11 für Direct
LOAD CASE= 1
Contact-Pressure
CPRESS
4500.00
3812.14
3169.29
2526.43
1883.57
1240.71
597.86
0.0000
Edit
4384.05 MPa
4376.00 MPa
4508.19 MPa
Y
Z
X

Literatur

1. Zur Finite-Element-Modellierung des stationären Rollkontakts vonRad und Schiene von der Fakultät Bauingenieurwesen der Technischen Universität Dresden von SABINE DAMME aus Dresden. Berichte des Instituts für Mechanik und Flächentragwerke Heft4 (2006)

2. Kapitel 11 - FEM-Analyse eines Rad-Schienen Rollkontaktes mit MEANS V10 (2014)

3. Part 16 - Rolling Contact of a Rail-Wheel-Model with MEANS V10 (2014)