# **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/mm<sup>2</sup>.

#### 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                          | Rzyl            | = 50.00  | [mm]                 |
| Width contact surface<br><u>Material</u> | b <sub>KF</sub> | = 15   | [mm]                 |
| Young's modulus E                        | EMat            | = 210000.00  | [N/mm <sup>2</sup> ] |
| Load F prescribed                        | Fk              | = 214200   | [N]                  |
| Results                                  |                 |  |                      |
| semi contact widht                       | а               | = $1.520 \cdot (F_k \cdot r_{Zyl} / (b_{kf} \cdot E_{Mat}))^{(1/2)}$<br>= 2.803    | [mm]                 |
| max. contact pressure                    | σο              | = $0.418 \cdot (F_k \cdot E_{Mat} / (b_{kf} \cdot r_{Zyl}))^{(1/2)}$<br>= 3237.166 | [N/mm <sup>2</sup> ] |

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

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

#### 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.

| 💀 FEM-System MEANS V11 fuer DirectX11                                |   |
|--|---|
|  |   |
| Files View Mesh Generation Edit FEM-P                                | roject FEM-Analysis Postprocessing Training |
| without Mesh O Wireframe Light 10%     with Mash I Educate Hidday Li | - 1. Main View - 🔂 3. Line-Modus 🔽 🗆 Trac   |
| Hidden-Line O with Mesh  | Line Modus is active 2. Node-Modus          |
|  | 3. Line-Modus                               |

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

| Surface Nodes Lines     | Surface Nodes Lines   |
|-------------------------|---|
| Number of Nodes = 1506  | Node: 1 New   |
| Nodes V                 | X: 52.0137  |
| from: 1                 | Y: 91.9881  |
| until: 1506             | Z: 79.8258  |
| Show Nodes              | Create Nodes  |
| Knotenbereich erzeugen  | Create Nodes  |
| Create a Range of Nodes | Create Lines  |
| Surface Nodes           | Circle / Rectangle  |
| Edge Nodes Surface      | Automatical and a second second second  |
| Knotenbereich löschen   | Conv Bange of Nodes   |
| Delete Range of Nodes   | Lina Medera   |
| Knotenbereich ändern    | Unit Nodes  |
| Coordinate-Factor       | Check Nodes   |
| Node: 1 EDIT            | Mesh Generators   |
| X: 52.0137              | 2D Mesh Generator   |
| Y: 91.9881              | 3D Mesh Grid  |
| Z: 79.8258              | in the second |
| Node numbering          | EG= 1 V   |
| Element numbering       |   |
| Element groups          | Load DXF-Lines  |
| Node-Size:              | UNDO / REDO   |
| Value= .02              |   |
| Size= large 🗸           |   |
|                         | Linien-Modus beenden  |

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.

|     | Create Rectangle or Ci  |   |
|-----|-------------------------|---|
| +   | Lircle Retangle         | le la |
| 1   | New REDO                | 1   |
| 4   | Actual Element group: 1 |   |
| f - | Middle-Point            |   |
| f   | X direction: 0.00       | ······                                    |
| 1   | Y direction: 0.00       |   |
| Ī   | Z direction: 0.00       |   |
|     | Radius: 50              | +   |
| 1.  | Number of Nodes: 92     | 1   |
| ł   | Start Angel: 0          | /   |
| 1 A | Only Nodes 360          | /   |
| A.  | Create Arc              |   |
| Y Y | Cancel                  |   |
|     |                         |   |
|     |                         | /   |
| ×   |                         |   |

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

| Circle Retangle New REDO Actual Bernert group: 1        |           |   |
|---|-----------|---|
| Middle-Point<br>X direction: 0.00<br>Y direction: 0.00  |           |   |
| Z direction: 0.00 Radius: 7.5 Number of Nodes: 24       | $\bigcap$ |   |
| Start Angel:         0           Only Nodes         360 |           |   |
| Create Arc<br>Cancel                                    |           |   |
|   |           | / |

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.

| 💀 Create Rectangle or Ci — 🔲 🗙 |             |
|--------------------------------|-------------|
| Circle Retangle                |             |
| New REDO                       |             |
| Actual Element group: 2        |             |
| Rectangle-Point                |             |
| X direction: -60               | f           |
| Y direction: -100              |             |
| Z direction: 0.00              |             |
| Width: 120                     |             |
| Height: 49.99                  | $\lambda$ / |
| Create Rectangle               |             |
| Cancel                         |             |
| L                              |             |
|                                |             |
|                                |             |
|                                |             |
| Y                              |             |
| 4                              |             |
|                                |             |
| <b>x</b>                       |             |

#### 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:

| 🖳 2D-Netzgenerator                         | ( <u>245</u> ) | × |
|--|----------------|---|
| from Bement Group: 1 until Bement Group: 2 |                |   |
| Element Typ: TRI3S V                       |                |   |
| Mesh Density: 300 V                        |                |   |
| Snap Radius: .005                          |                |   |
| QUAD-Meshing                               |                |   |
| 3D-Extrusion                               |                |   |
| Nodes in Z Direction:<br>Z-Depth:          |                |   |
| Check Nodes Refine Mesh                    |                |   |
| Cancel Tutorial MESH GENERATIO             | N              |   |
|  |                |   |

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

| 🖉 Create Rectangle or Ci — 🔲 🗙 |   |
|--------------------------------|---|
| Circle Retangle                |   |
| New REDO                       |   |
| Actual Element group: 3        |   |
| Rectangle-Point                |   |
| X direction: -1                |   |
| Y direction: -49               |   |
| Z direction: 0.00              | 1 |
| Width: 2                       | / |
| Height: 5                      |   |
|                                |   |
| Create Rectangle               |   |
|                                |   |
|                                |   |
| Cancel                         |   |
|                                |   |
|                                |   |

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

| 💀 Create Rectangle or Ci — 🔲 X                   |  |
|--|--|
| New REDO   |  |
| Actual Element group: 4                          |  |
| X direction: -1                                  |  |
| Y direction:     -56       Z direction:     0.00 |  |
| Width: 2<br>Height: 5                            |  |
| Create Rectangle                                 |  |
| Cancel   |  |
| Y  |  |
| <b>†</b>   |  |
| ×  |  |

| In Line-Mode, select the "2D Mesh Generator" and "Refine Mesh" me | nu |
|---|----|
|---|----|

| 🛃 2D Mesh Generation                         |   | × |
|--|---|---|
| from Element Group: 1 until Element Group: 4 |   |   |
| Element Typ: TRI3S ~                         |   |   |
| Mesh Density: 300 ~                          |   |   |
| Snap Radius: 5E-06                           |   |   |
| QUAD-Meshing                                 |   |   |
| JD-Extrusion                                 |   |   |
| Nodes in Z Direction: 5                      |   |   |
| Z-Depth: 15                                  |   |   |
| Check Nodes Refine Mesh                      |   |   |
| Cancel Tutorial MESH GENERATIO               | N |   |
|  |   |   |

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.



## 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.

| FEM-P | Project                                 | FEM-Analysis Postp   | processing Tra                                   | ining   |        |  |
|-------|---|--|--|---|--------|--|
| tions | 1. Bou                                  | undary-Condition: •<br>w Boundary-Conditions<br>Surface Modus is acti                | Element-Groups<br>ive - Surface= 8               | Material-Datas  | Editor | 6. Loads     Image: Temperature       1. Element Groups     Temperature       2. Element Nodes     Temperature       3. Nodal Coordinates     Image: Material Datas       4. Material Datas     Soundary-Conditions       6. Loads     Form Optimization       8. Delete |
|       | Edit Load                               | ds   |  | - 0   | ×      |  |
|       | Nr                                      | Node   | FHG  | Value   | ^      |  |
|       | 1                                       | 93   | 2  | -751.6  |        |  |
| -     | 2                                       | 94   | 2  | -751.6  |        |  |
|       | 3                                       | 95   | 2  | -751.6  |        | Edit Load Case — 🗆 X   |
|       | 4                                       | 96   | 2  | -751.6  |        | · · · · · · · · · · · · · · · · · · ·  |
|       | 5                                       | 97   | 2  | -751.6  |        | Actual Load Case: 1 🗸 K  |
|       | 6                                       | 98   | 2  | -751.6  |        | Factor 751.6   |
|       | 7                                       | 99   | 2  | -751.6  |        |  |
|       | 8                                       | 100  | 2  | -751.6  |        | multiply     O divide  |
|       | 9                                       | 101  | 2  | -751.6  |        | ◯ add ◯ replace  |
|       | 10                                      | 102  | 2  | -751.6  |        |  |
|       | 11                                      | 103  | 2  | -751.6  |        | 2010EL   |
|       | 12                                      | 104  | 2  | -751.6  |        | CANCEL   |
|       | ad Case:<br>mber of Lo<br>I<br>D<br>Pre | 3 < ads/Load Case: 285 L Vew Load Case elete Load Case Load Factor ssure->Point Load | Load Car oad Type:     1     Com     Convert Ter | ees: 3<br>Point Load<br>bine Load Cases<br>opy Load Case<br>operature to a Load C<br>Change FHG | ase    |  |
|       |   |  | ок   |   |        |  |

## **Create Boundary Conditions**

#### **Clamped fixed**

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

| File       View       Mesh Generation       Edit FEM-Project       FEM-Analysis       Postprocessing       Training         I.       Point Load       Boundary-Conditions       Is       Benerat-Concupitions       Is       Element-Groups       Material-Datas       Editor         I.       Boundary-Conditions       Is       Benerat-Concupitions       Is       Element-Groups       Material-Datas       Editor         I.       Boundary-Conditions       Is       Element-Groups       Material-Datas       Editor         I.       Boundary-Conditions       Is       Element-Groups       Material-Datas       Editor         I.       Boundary-Conditions       Is       Is       Is       Is       Is       Is         I.       Boundary Conditions       400       New       Is       Is <t< th=""><th>Files       View       Mech Generation       Edit FEM-Project       FEM-Analysis       Postprocessing       Taning            <ul> <li>Point Laad</li> <li>Bounday-Condition</li> <li>Bounday-Conditions</li> <li>Bounday-Con</li></ul></th><th>FEM System MEANS V11 - FEM Str</th><th>ucture File C:\projekte\ro</th><th>ollkontakt_neu\pen6.fem</th><th></th><th></th><th></th></t<> | Files       View       Mech Generation       Edit FEM-Project       FEM-Analysis       Postprocessing       Taning <ul> <li>Point Laad</li> <li>Bounday-Condition</li> <li>Bounday-Conditions</li> <li>Bounday-Con</li></ul> | FEM System MEANS V11 - FEM Str  | ucture File C:\projekte\ro   | ollkontakt_neu\pen6.fem  |                |   |                |
|---|--|---|--|--|----------------|---|----------------|
| I. Point Load Boundary-Conditions I. Boundary-Co  | 1. Point Load       Boundary-Conditions         1. Boundary-Conditions       1. Boundary-Conditions         1. Boundary Conditions       1. Boundary-Conditions         2. Beam-Joint       2. Beam-Joint         3. Editor       1. Boundary Conditions         Value of Boundary Conditions       490         Were of Boundary Conditions       1. Develon         In Y Direction       In 2 Direction         In Y Direction       Camped fixed         Constraints displacement in       In 2 Direction         In Y Direction       Camped fixed         Selecton:       Order of Rowing surfaces         Selecton:       Design a model region         Selector:       Deside BCs  | Files View Mesh @   | eneration Edit FEM-I   | Project FEM-Analysis   | Postprocessing | Training  |                |
| Boundary Conditions   | Boundary Conditions     Aundeer of Boundary Conditions     Idea of Boundary     Id  | ads ✓ Show Loads  | Boundary-Conditions  | <ol> <li>Boundary-Conditio</li> <li>Boundary Conditio</li> <li>Beam-Joint</li> <li>Editor</li> </ol> | ns Element-Gro | Material-Datas E  | ditor 6. Loads |
| Constraints displacement in   in X Direction   in Y Direction   Colour of Axis: BLACK: X-Axis: BLUE: Y-Axis: RED: Z-Axis:   Selection:   Select Surfaces   Dragging a model region   Select Nodes   Select all showing nodes   Define a coordinate range   Select all showing surfaces   Show the Boundary Conditions:   Doublete BCs   | Constraints displacement in     In X Direction        In Y Direction        Colour of Axis: BLACK: X-Axis; BLUE: Y-Axis; RED: Z-Axis     Selection:     Select Nodes        Select Nodes        Select All showing nodes        Select Nodes        Select all showing surfaces        Show the Boundary Conditions:     0034     normal     Cancel   Edtor   Create BCs   | Boundary Conditions Number of Boundary Conditions Value of Boundary 1E                  | 480 New  | ×  |                |   |                |
| <ul> <li>Select Surfaces</li> <li>Dragging a model region</li> <li>Select Nodes</li> <li>Select all showing nodes</li> <li>Define a coordinate range</li> <li>Select all showing surfaces</li> <li>Show the Boundary Conditions: .00034</li> <li>nomal</li> <li>Cancel</li> <li>Editor</li> <li>Create BCs</li> <li>Delete BCs</li> </ul>   | <ul> <li>Select Surfaces</li> <li>Dragging a model region</li> <li>Select Nodes</li> <li>Select all showing nodes</li> <li>Define a coordinate range</li> <li>Select all showing surfaces</li> <li>Show the Boundary Conditions: .00034</li> <li>normal</li> <li>Cancel</li> <li>Editor</li> <li>Create BCs</li> <li>Delete BCs</li> </ul>   | Constraints displacement in<br>in ><br>Colour of Axis: BLACK: X-Axis; BLU<br>Selection: | ( Direction in Z D<br>' Direction Clamp<br>E: Y-Axis; RED: Z-Axis)   | Direction<br>Ded fixed   |                |   |                |
| Show the Boundary Conditions: .00034 nomal  Cancel Editor Create BCs Delete BCs   | Show the Boundary Conditions: 00034 normal   | Select Surfaces     Select Nodes     Define a coordinate range                          | Dragging a model n     Select all showing n     Select all showing n | region<br>nodes<br>surfaces  |                |   |                |
| Cancel Editor Create BCs Delete BCs   | Cancel Editor Create BCs Delete BCs  | Show the Boundary Conditions:   | .00034 norm  | nal 🗸  | mile           |   |                |
| Delete BCs  | Delete BCs   | Cancel Editor   | Create   | e BCs  | States.        |   |                |
|   |  |   | Delete   | e BCs  |                |   |                |
|   |  |   |  |  |                | the second se |                |
|   |  | Y   |  |  |                |   |                |

## 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 **with** to evaluate the results of contact analysis, these include

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

| FEM System MEAN | JS V11 - FEM S | tructure File(                 | C:\projekte\r                  | ollkontakt               | _neu\roll_96_1       | .fem             |              |                      |   |
|-----------------|----------------|--------------------------------|--------------------------------|--------------------------|----------------------|------------------|--------------|----------------------|---|
| Files Vi        | ÷<br>ew Mesh   | Generation                     | Edit FEM-                      | Project                  | FEM-Analy            | sis Postpro      | ocessing     | Training             | , |
| Show            | Results Pi     | splacement-F<br>ck, Search Val | actor Lis<br>ues Le            | st Result V<br>egende: L | 'alues<br>egende 1 👻 | Intern<br>Extern | List<br>List | FEM-File<br>STA-File |   |
| Postprocessin   | g 🗔 S          | kalieren/Anze                  | eigen 🕞                        | Legende                  | e/Tabelle 🕞          | Animations       | s Ta Li      | st Files 🕞           |   |
|                 |                |                                |                                |                          |                      |                  |              |                      |   |
|                 |                |                                |                                |                          |                      |                  |              |                      |   |
|                 | 🖳 Po           | stprocessing                   |                                |                          |                      | - 0              | ×            |                      |   |
|                 | Resu           | ılts:                          |                                |                          |                      |                  |              |                      |   |
|                 | 0              | Contour of Disp                | lacement                       |                          | Load Case: 1         | ~                |              |                      |   |
|                 | ۲              | Nodal Stress C                 | ontour                         | (                        | Reaction Fo          | rces             |              |                      |   |
|                 | 0              | Element Stress                 | Contour                        | (                        | Contour of F         | orces            |              |                      |   |
|                 |                |                                |                                |                          |                      |                  |              |                      |   |
|                 | ACCL           | racy:<br>Edit Accuracy:        |                                |                          | Displacen            | ent Factor       | 1            |                      |   |
|                 |                | Call / Woodrady.               |                                | -                        | Edit Colours         | for Legend       |              |                      |   |
|                 |                |                                | 1                              | -                        | Diel: Comete         | Caus Values      | _            |                      |   |
|                 |                | 1 3                            | 4                              |                          | FICK, Search         | , save values    |              |                      |   |
|                 | Selec          | ct Result Comp                 | onent:                         |                          |                      |                  |              |                      |   |
|                 |                | CPRESS                         | 2                              |                          |                      |                  |              |                      |   |
|                 | _              | CPRESS                         | 1                              |                          |                      |                  |              |                      |   |
|                 |                | Car von Mise                   | 2<br>s Stress                  |                          |                      |                  |              |                      |   |
|                 |                | Normal St                      | ress Sigma x                   |                          |                      |                  |              |                      |   |
|                 | -              | Normal St                      | ress Sigma z                   |                          |                      |                  |              |                      |   |
|                 |                | Shear Str                      | ess Tau yz                     |                          |                      |                  |              |                      |   |
|                 |                | 1st princip                    | bal Stress S1                  |                          |                      |                  |              |                      |   |
|                 |                | 3rd princi                     | pai Stress S2<br>pal Stress S3 |                          |                      |                  |              |                      |   |

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:

| 220     | U.UU2JJ    |       |
|---------|------------|-------|
| 44579   | -0.00438   |       |
| 56274   | -0.00438   |       |
| 32601   | -0.00438   |       |
| 44578   | -0.00451   |       |
| 32600   | -0.00451   |       |
| 56273   | -0.00451   |       |
| 2738    | -0.00608   |       |
| 8164    | -0.00609   |       |
| 5451    | -0.00610   |       |
| 32602   | -0.00612   |       |
| 44580   | -0.00612   |       |
| 56275   | -0.00621   |       |
| 31654   | -0.00622   |       |
| 10877   | -0.00639   |       |
| 25      | -0.00640   | ~     |
|         | All Values |       |
| New Ran | ige Add    | Range |

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



🙀 FEM System MEANS V11 - FEM Structure File C:\projekte\rollkontakt\_neu\roll\_96\_pen15.fem

Select the Icon

to display the v.Mises-Stress

🚆 FEM System MEANS V11 - FEM Structure File C:\projekte\rollkontakt\_neu\roll\_96\_pen15.fem 0 1 🖸 = Files View Mesh Generation Edit FEM-Project FEM-Analysis Postprocessing Training Displacement-Factor List Result Values Intern List FEM-File MEANS V11 für DX9 starten 🍳 🗹 Show Results Pick, Search Values Legende: Legende 1 - Extern List STA-File 11 Skalieren/Anzeigen 🗊 Legende/Tabelle 🗊 Animations 🗊 List Files 🗊 MEANS V11 für DirectX9 🗊 Postprocessing LOAD CASE= 1 Contour of Stress von Mises Stress 2500.00 2117.98 1760.96 1403.94 1046.92 689.90 332.88 0.8613 Edit +

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



surface.

with to display the contact pressure CPRESS on the slave contact

| C 0 1 🛛 👘                  |  |  |                      |                            |                       |
|----------------------------|--|--|----------------------|----------------------------|-----------------------|
| Files View M               | lesh Generation Edit Fl                    | EM-Project FEM-Analysis                    | Postprocessing       | g Training                 |                       |
| Show Results               | Displacement-Factor<br>Pick, Search Values | List Result Values<br>Legende: Legende 1 - | Intern L<br>Extern L | st FEM-File<br>st STA-File | MEANS V11 für DX9 sta |
| Postprocessing 🕞           | Skalieren/Anzeigen                         | Legende/Tabelle 🕞                          | Animations 🕞         | List Files 🖓               | MEANS V11 für Direct  |
| LOAD CASE= 1               |  |  |                      |                            |                       |
| Contact-Pressure<br>CPRESS |  |  |                      |                            |                       |
| 4500.00                    |  |  |                      |                            |                       |
| 3812.14                    |  |  |                      |                            |                       |
| 3169.29                    |  |  |                      |                            |                       |
| 2526.43                    |  |  |                      |                            |                       |
| 1883.57                    |  |  |                      |                            |                       |
| 1240.71                    |  |  |                      |                            |                       |
| 597.86                     |  |  |                      |                            |                       |
| 0.0000                     |  |  |                      |                            |                       |
| Edit +                     |  |  |                      |                            |                       |
|                            |  |  |                      |                            |                       |
|                            |  |  |                      |                            |                       |
|                            |  |  |                      |                            |                       |
|                            |  | 16   |                      |                            |                       |
|                            |  |  |                      |                            |                       |
|                            |  |  |                      | 384.05 MPa                 |                       |
|                            |  |  |                      | 76.00 MPa                  |                       |
|                            |  |  |                      |                            |                       |
|                            |  |  | 4508.1               | 9 MPa                      |                       |
|                            |  |  |                      |                            |                       |
|                            |  |  |                      |                            |                       |
| V                          |  |  |                      |                            |                       |
| 4                          |  |  |                      |                            |                       |
| z                          |  |  |                      |                            |                       |
|                            | - X  |  |                      |                            |                       |
|                            | 57A)                                       |  |                      |                            |                       |

🖳 FEM System MEANS V11 - FEM Structure File C:\projekte\rollkontakt\_neu\roll\_96\_pen15.fem

#### 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)