# Strength proof according to the FKM-Guideline within Creo Simulate

Markus Kölbl | ITD | ZF Test Systems

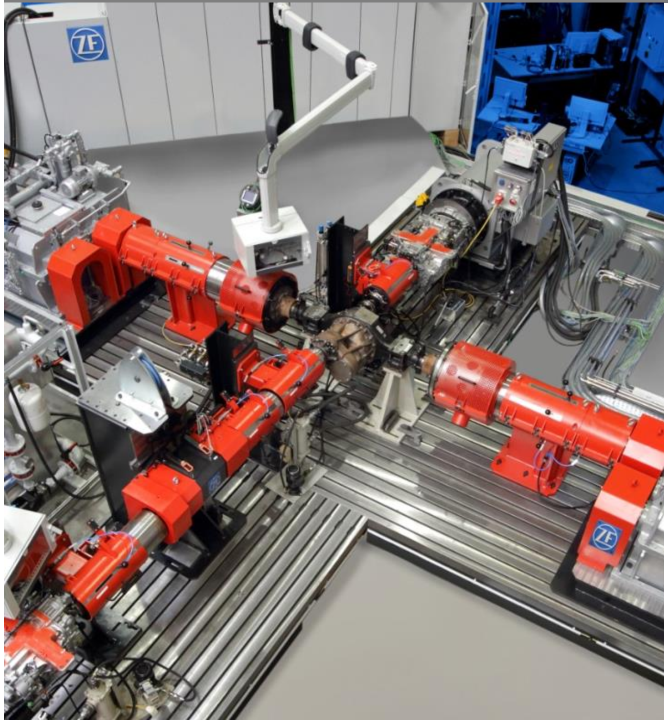
# Agenda

1. ZF Test Systems
2. Use from creo simulate in our business unit
3. Motivation
4. Installation
5. Example bending beam with notch
6. Implementation and Using femMeshFKM
7. Postprocessing in Creo

# 01
# ZF Test Systems

# Test Systems

## Facts




Sales 2016*: 86.8 million €

Growth 2002 - 2016 12 % p.a.

1000 test systems in 40 countries at 300 customers.

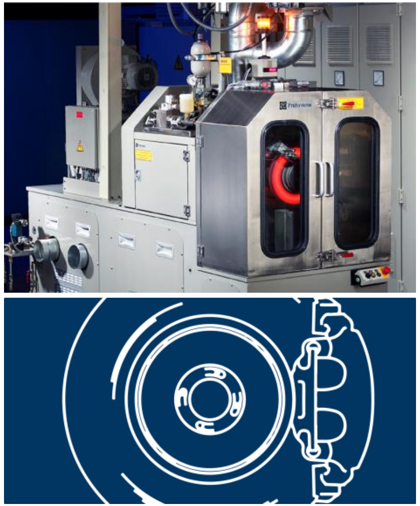
* non consolidated

# Test Systems

## Product Overview / Test Systems for...

Transmissions and Axles

Tires and Wheels



Brakes


Transmission Components and Oils

# 02

# Use from creo simulate in our business unit

# Use of Creo Simulate

## Static Analysis

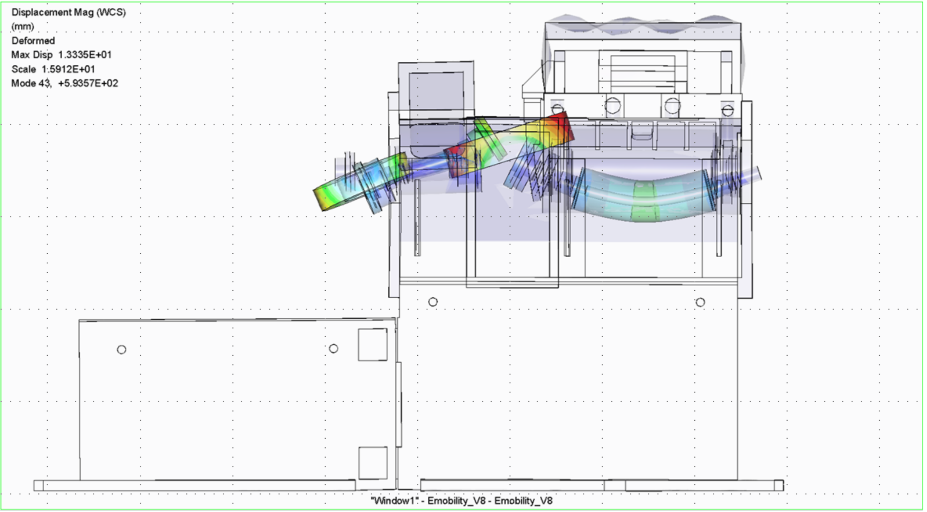
Structural-mechanical analyses with strength proof for:

- Measuring rims
- Load wheels and drums for tire and vehicle test benches
- High-strength screw connections
- highly stressed welded connections
- Flywheel masses under centrifugal load

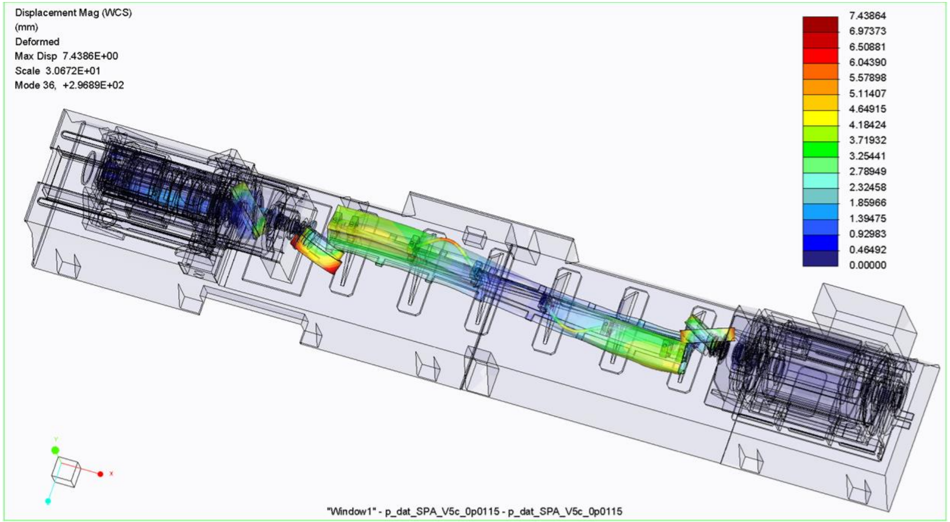
# Use of Creo Simulate

Modal analysis / Frequency analysis

Test bench for **E-drive components** rotor-dynamic drive train




Test bench for **rotary oscillation damper** in the converter gear rotordynamic drive train



# 03
# Motivation

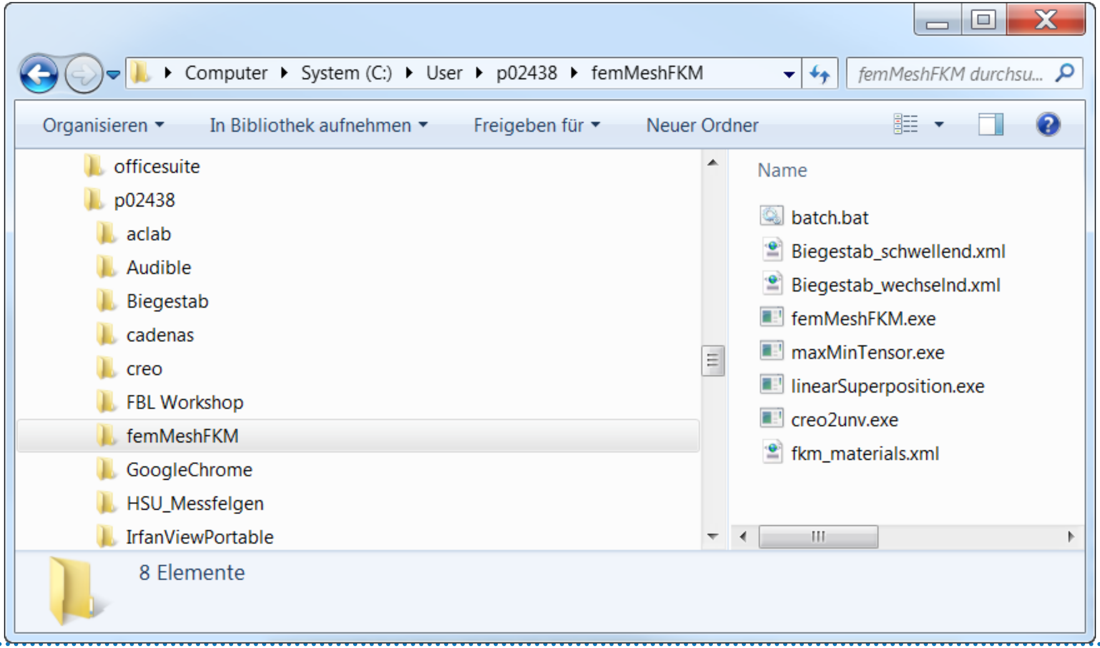
# Motivation

- The strength verification according to FKM with FEM results is time-consuming and requires a separate tool, e.g. KISSsoft.
- The strength verification was so far only carried out at individual points of the model. The selection of verification points is usually based on the equivalent stresses. The following influences can not or not sufficiently be considered.
  - Locally different limit value of strain or plastic notch factor
  - The location of the most critical combination of stress amplitude and mean stress
  - The local stress gradient
- femMeshFKM was developed from ZF for railway applications (IX), where FEM calculations are performed with Permas. Postprocessing was done in Hyperworks.
- For ZF Test Systems, femMeshFKM has been extended to use Creo Simulate data. The postprocessing can be done also in Simulate.

# 04
# Installation

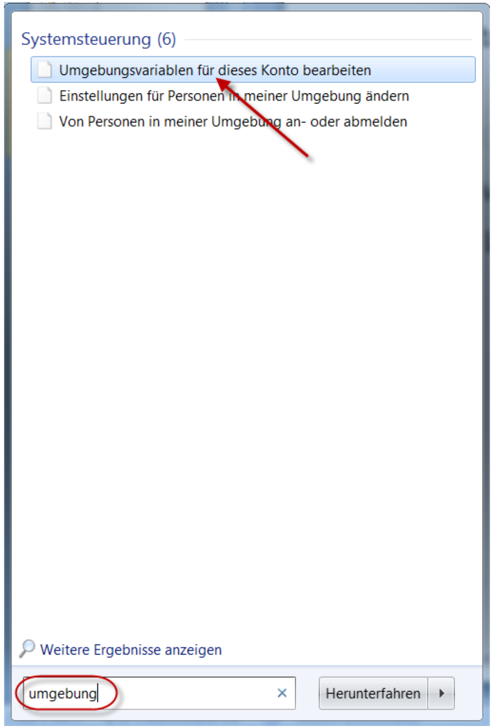
# Installation

- Copy the files from the program folder into a work directory
  in this example D:\Arbeitsverzeichn_CAD\femMeshFKM

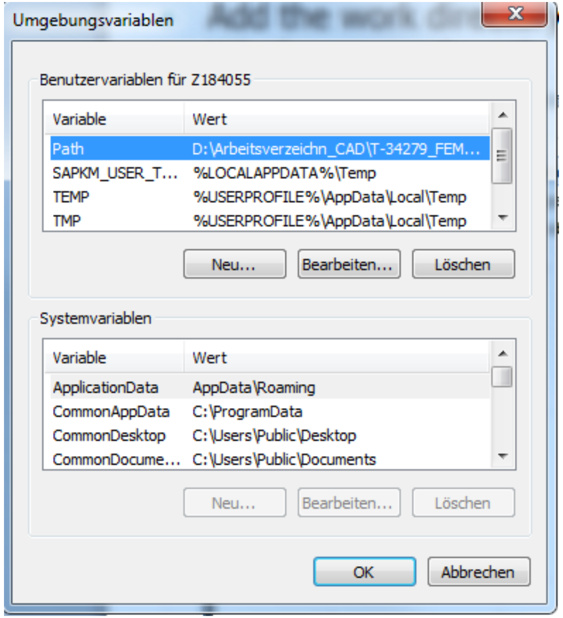


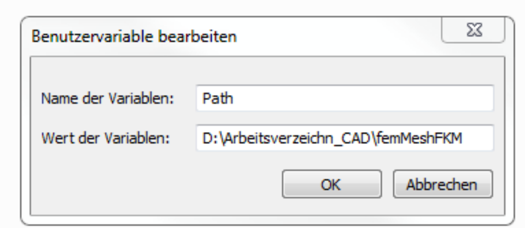
# Installation

- Add the work directory to the Windows search path




- Create a user-defined variable with the working directory path, if the variable is already existing, add it as the first entry and separate it with a semicolon



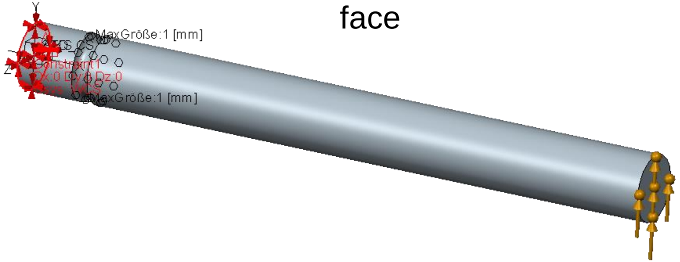



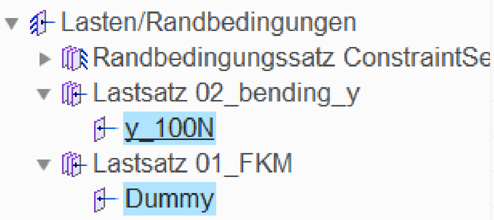
# 05
# Example bending beam with notch

# Example bending beam with notch

- Cylindrical beam with notch and uniaxial bending

Fixing at one end face




$$d := \sqrt[3]{\frac{100 \cdot mm^3 \cdot 32}{\pi}} = 10.0616\,mm$$

$$F_z := 100 \cdot N$$

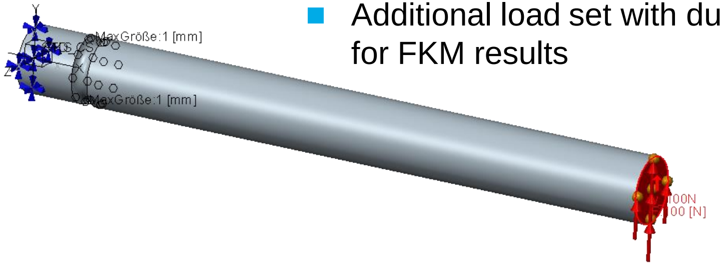
$$laenge := 100 \cdot mm$$

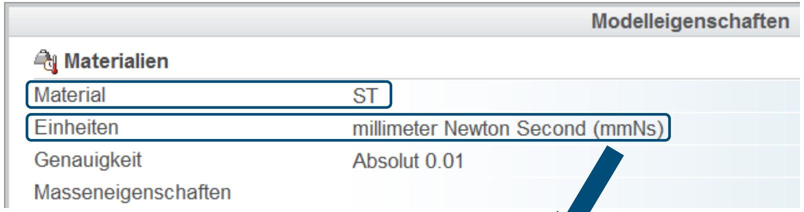
$$M_B := F_z \cdot laenge = 10\,N \cdot m$$

$$W_B := \frac{\pi}{32} \cdot d^3 = 100\,mm^3$$

$$\sigma_B := \frac{M_B}{W_B} = 100\,MPa$$

- One load set with bending in y direction
- Additional load set with dummy for FKM results

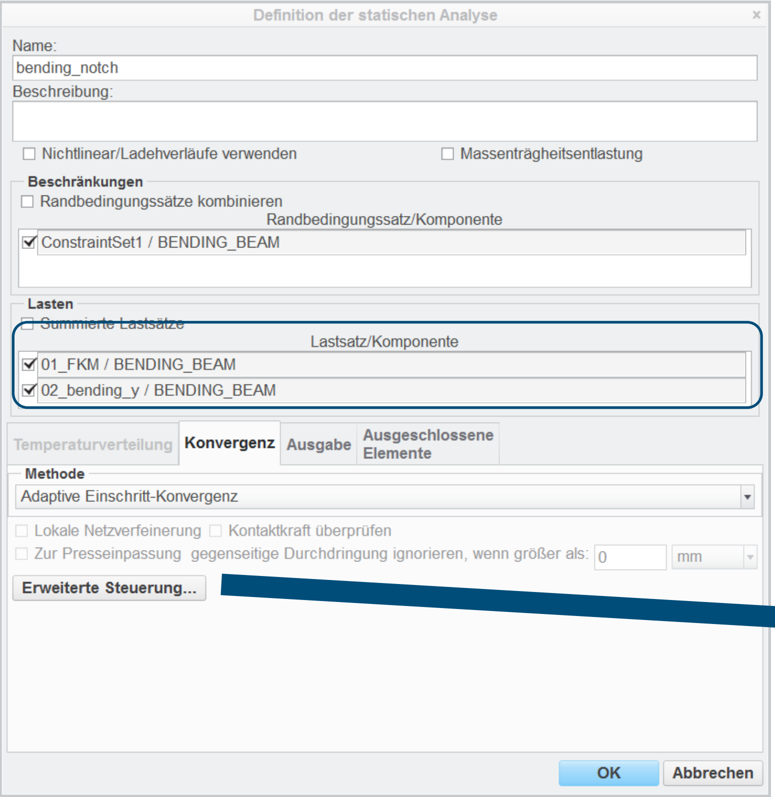


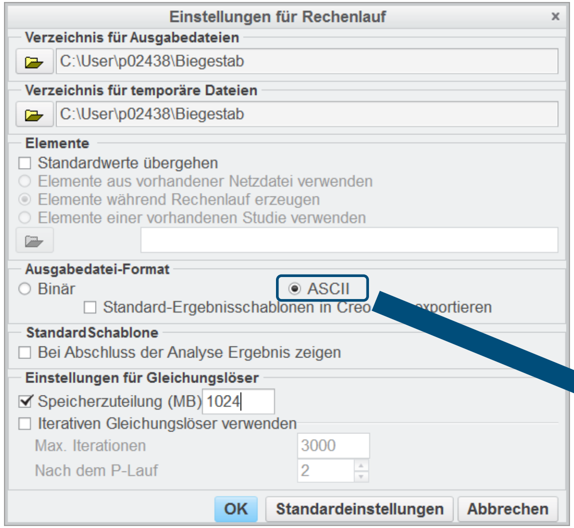




For stress results in in MPa

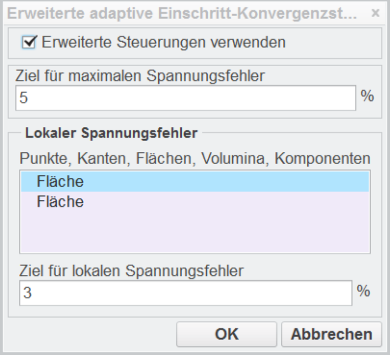
ZF start modell has millimeter Kilogram Sec mmKs

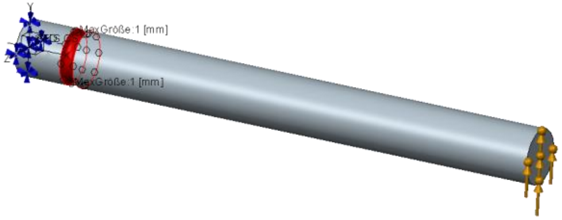
# Example bending beam with notch



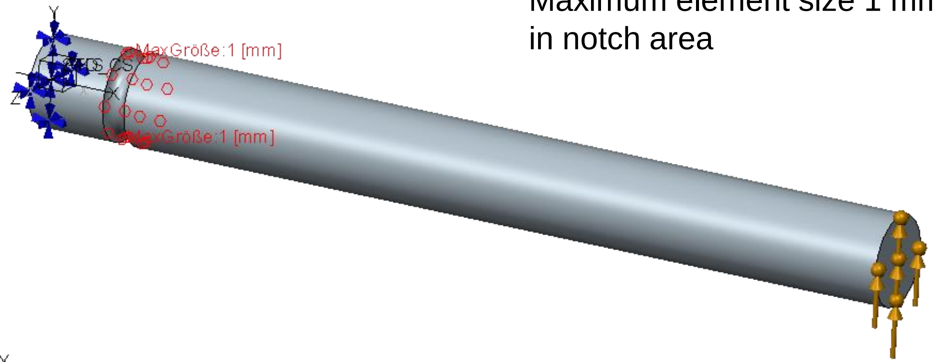

Necessary for postprocessing with femMeshFKM



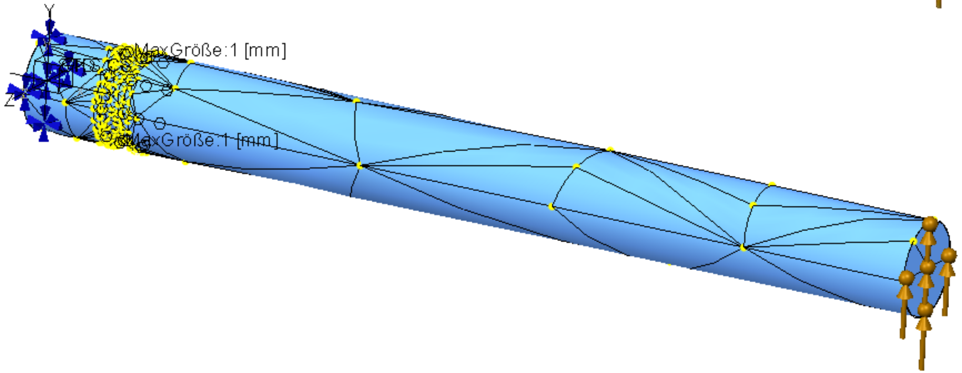



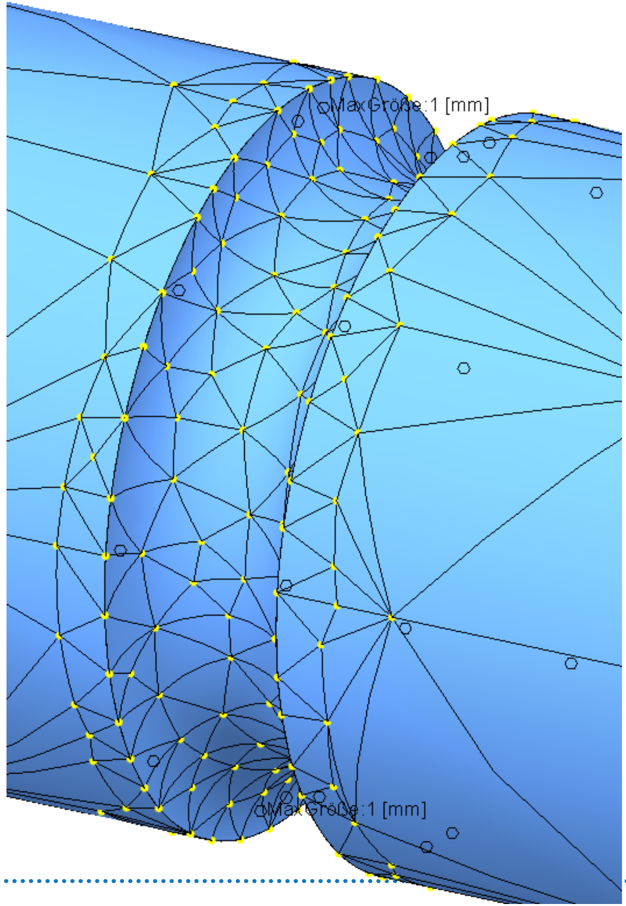

Reduced local stress error for notch area

# Example bending beam with notch

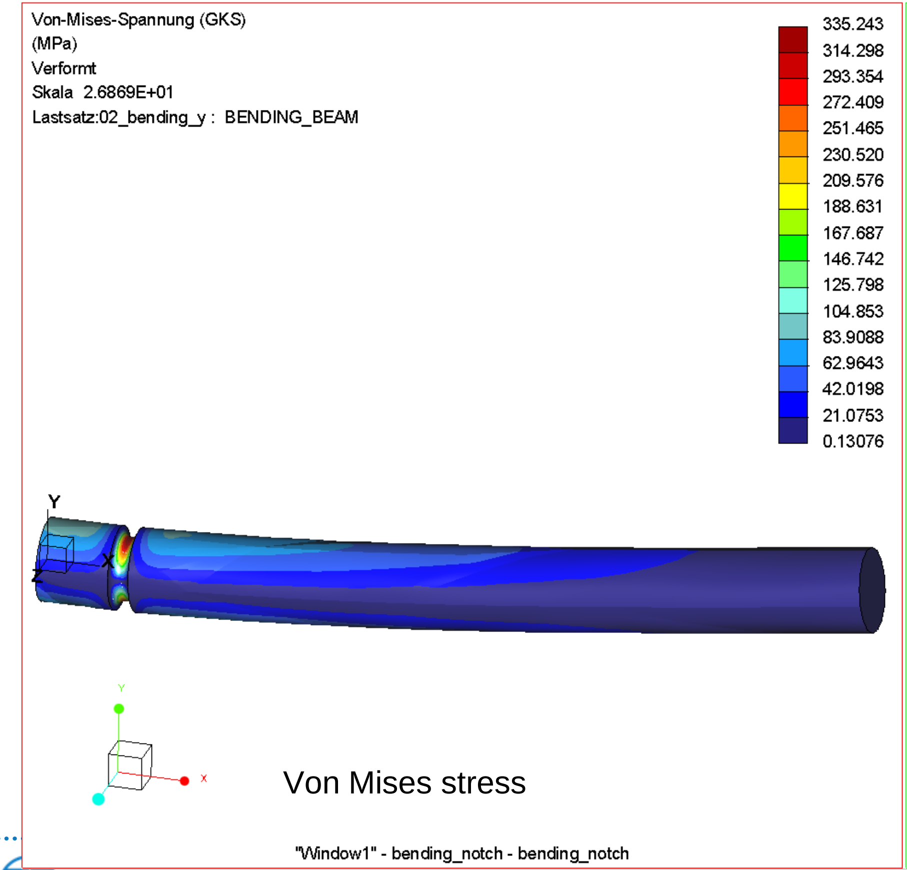


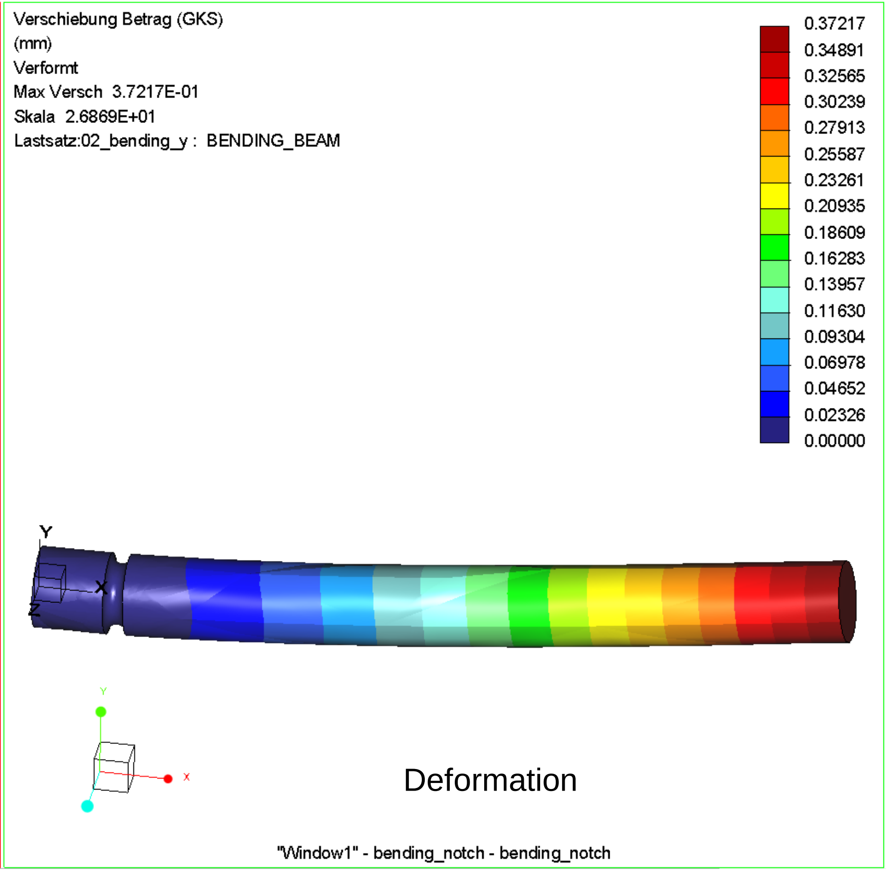

Maximum element size 1 mm
in notch area



# Example bending beam with notch



# Example bending beam with notch

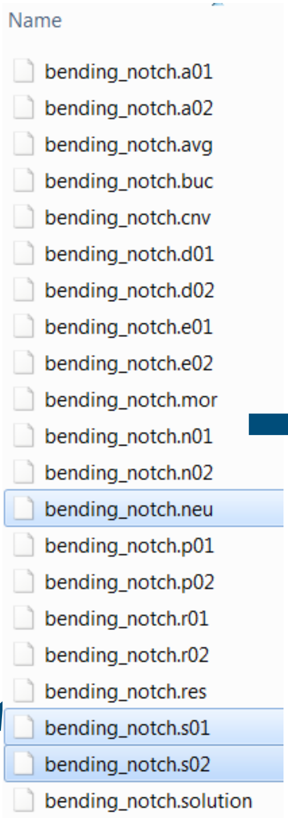
- Copy simulate results for postprocessing to femMeshFKM work directory

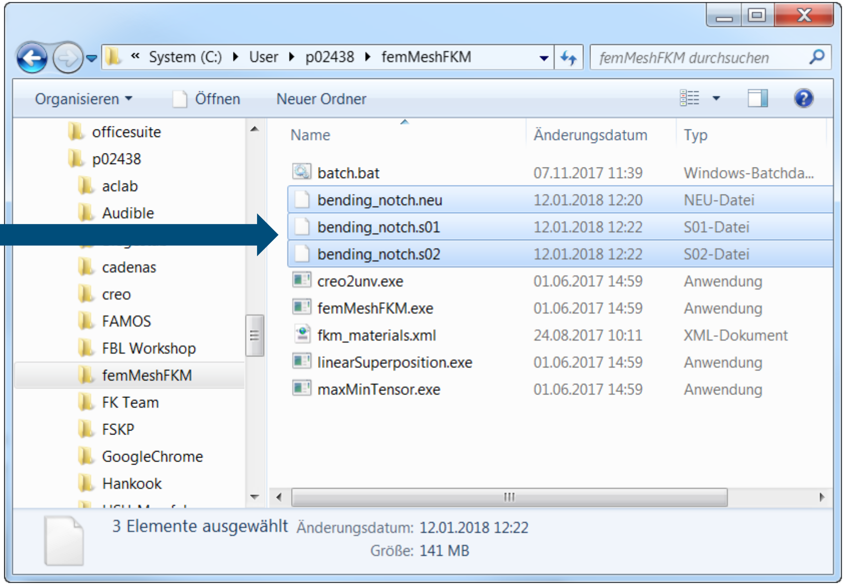
- Mesh geometry

- Stress results for each load set

- File extensions are assigned according to ASCII sorting of the names of the load sets

01_FKM => .s01
02_bending_y => .s02



# 06
# Implementation and Using femMeshFKM

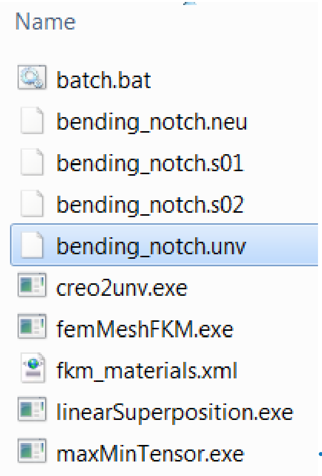
# Convert model from Simulate to UNV

batch.bat - Editor

Datei Bearbeiten Format Ansicht ?

```
@Echo off

cls
echo.
echo translate mesh definition from Creo to UNV
creo2unv.exe bending_notch.neu
cls

cls
echo.
echo create additional zero load set
linearSuperposition.exe 0.0 bending_notch.s02

cls
echo.
echo acalculate amplitude and mean stress at each node
maxMinTensor.exe -o bending_notch bending_notch_linSup.unv bending_notch.s02

cls
echo.
echo strength verification
femMeshFKM --creo_file bending_notch.s01 bending_notch.xml
cls


echo fertig
```

- Convert the Simulate mesh to the format UNV
- Execute the program **creo2unv.exe** with batch file. Use the file ending with **.neu** from the Simulate result folder as parameter
- **creo2unv.exe** creates a file with the same name ending in **.unv**



# Create additional load cases by linear superposition

- With the **linearSuperposition.exe** program, load sets can be scaled or superimposed.

batch.bat - Editor

Datei Bearbeiten Format Ansicht ?

```
@Echo off

cls
echo.
echo translate mesh definition from Creo to UNV
creo2unv.exe bending_notch.neu
cls

cls
echo.
echo create additional zero load set
linearSuperposition.exe 0.0 bending_notch.s02

cls
echo.
echo acalculate amplitude and mean stress at each node
maxMinTensor.exe -o bending_notch bending_notch_linSup.unv bending_notch.s02

cls
echo.
echo strength verification
femMeshFKM --creo_file bending_notch.s01 bending_notch.xml
cls


echo fertig
```

Generates a new stress file **_linsup** in the formats **.unv** and **.hwascii** scaled with the specified factor **0.0** from the specified stress file **bending_notch.s02**

Name

- batch.bat
- bending_notch.neu
- bending_notch.s01
- bending_notch.s02
- bending_notch.unv
- bending_notch_linSup.hwascii
- bending_notch_linSup.unv
- creo2unv.exe
- femMeshFKM.exe
- fkm_materials.xml
- linearSuperposition.exe
- maxMinTensor.exe

# Create additional load cases by linear superposition

- With the **linearSuperposition.exe** program, load sets can be scaled or superimposed.

batch.bat - Editor

Datei Bearbeiten Format Ansicht ?

```
@Echo off

cls
echo.
echo Lassaetze ueberlagern
echo.
linearSuperposition.exe Felge_9_5x20_01.s02 Felge_9_5x20_01.s03 Felge_9_5x20_01.s04

cls
echo.
echo Modell von Creo nach UNV uebersetzen (fuer Anzeige in HyperView)
echo.
creo2unv.exe Felge_9_5x20_01.neu
creo2unv.exe Felge_9_5x20_01.s01 Felge_9_5x20_01.s02 Felge_9_5x20_01.s03 Felge_9_5x20_01.s04
cls

echo.
echo Maximale Amplitude und Mittelspannung an jedem Knoten bestimmen
maxMinTensor.exe -o Felge_9_5x20 Felge_9_5x20_01_02.unv Felge_9_5x20_01_linSup.unv


cls
echo.
echo Dauerfestigkeitsnachweis nach FKM
femMeshFKM --creo_file Felge_9_5x20_01.s01 FKM_Felge_9_5x20.xml
echo fertig
```

Creates from the specified stress files
**Felge_9_5x20_01.s02**
**Felge_9_5x20_01.s03**
**Felge_9_5x20_01.s04** new stress files **_linsup** in the formats **.unv** and **.hwascii** with a linear superposition for each node

Name

- Felge_9_5x20_01.unv
- Felge_9_5x20_01_01.hwascii
- Felge_9_5x20_01_01.unv
- Felge_9_5x20_01_02.hwascii
- Felge_9_5x20_01_02.unv
- Felge_9_5x20_01_03.hwascii
- Felge_9_5x20_01_03.unv
- Felge_9_5x20_01_04.hwascii
- Felge_9_5x20_01_04.unv
- Felge_9_5x20_01_linSup.hwascii
- Felge_9_5x20_01_linSup.unv

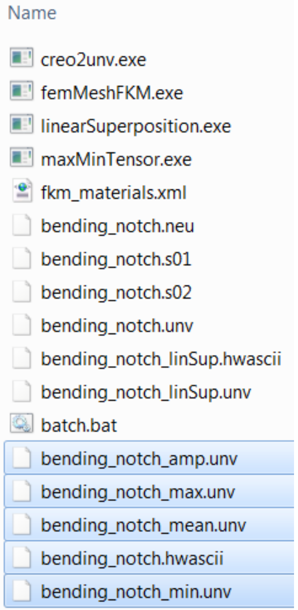
# Calculation of amplitude and mean stress

- For the strength verification, the amplitude and the associated mean stress must be calculated at each node.
- **maxMinTensor.exe** generates with the specified name a file with the ending **.hwascii** and **_amp.unv** (amplitudes), **_mean.unv** (mean stress), **_max.unv** (maximum stress), **_min.unv** (minimum stress)

batch.bat - Editor

Datei Bearbeiten Format Ansicht ?

```
@Echo off

cls
echo.
echo translate mesh definition from Creo to UNV
creo2unv.exe bending_notch.neu
cls

cls
echo.
echo create additional zero load set
linearSuperposition.exe 0.0 bending_notch.s02

cls
echo.
echo acalculate amplitude and mean stress at each node
maxMinTensor.exe -o bending_notch bending_notch_linSup.unv bending_notch.s02

cls
echo.
echo strength verification
femMeshFKM --creo_file bending_notch.s01 bending_notch.xml
cls


echo fertig
```



# Material properties

- Material database **fkm_materials.xml**:
  - Create material with data from the FKM guideline according to the comments in the database.

fkm_materials.xml - Editor

Datei Bearbeiten Format Ansicht ?

```
<?xml version="1.0" ?>
<!-- Materials database for use with femMeshFKM
 Entries for each material:
        name="EN-GJS-400-15"
        number="0.7040"
        type="GJS"          {GJS, AluGuss}
        R_m_N="400"         Zugfestigkeit, Kapitel 5
        R_p_N="250"         Streckgrenze, Kapitel 5
        A="15"              Bruch {GJS, GJL, GJM, Stahl, GS, AluGuss, AluKnet}
        f_W_sigma="0.34"    Zugdruckwechselfestigkeitsfaktor, Tabelle 4.2.1
        f_W_tau="0.65"      Schubwechselfestigkeitsfaktor, Tabelle 4.2.1
        a_G="0.05"          Konstante für Stützzahl, Tabelle 4.3.2
        b_G="3200"          Konstante für Stützzahl, Tabelle 4.3.2
        a_R_sigma="0.16"    Konstante für Rauheitsfaktor, Tabelle 4.3.5
        R_m_N_min="400"     Minimale Zugfestigkeit für Rauheitsfaktor, Tabelle 4.3.5
        a_M="0.35"          Konstante für Mittelspannungsempfindlichkeit, Tabelle 4.4.1
        b_M="0.08"          Konstante für Mittelspannungsempfindlichkeit, Tabelle 4.4.1
        N_D_sigma="1e6"     Knickpunktzyklenzahl, Tabelle 4.4.3
        k_sigma="5"         Neigungsexponent, Tabelle 4.4.3
        N_D_II_sigma="1e6"  Knickpunktzyklenzahl, Tabelle 4.4.3
        k_II_sigma="5"      Neigungsexponent, Tabelle 4.4.3
        K_NL_E="1.0"        Faktor für GJL, Tabelle 4.3.8 (default: K_NL_E=1.0)
        K_A="1.0"           Anisotropiefaktor, Kap. 3.2.1.5
        d_eff_N_m="60"      Konstante für Größenfaktor bzgl. Rm, Tabelle 3.2.2
        a_d_m="0.15"        Konstante für Größenfaktor bzgl. Rm, Tabelle 3.2.2
        d_eff_N_p="60"      Konstante für Größenfaktor bzgl. Rp, Tabelle 3.2.2
        a_d_p="0.15"        Konstante für Größenfaktor bzgl. Rp, Tabelle 3.2.2
        f_sigma="1.3"       Druckfestigkeitsfaktor, Tabelle 3.2.5
        f_tau="0.65"        Schubfestigkeitsfaktor, Tabelle 3.2.5
        K_NL_Zug="1.0"      Faktor für GJL, Tabelle 3.3.2 (default: K_NL_Zug=1.0)
        K_NL_Druck="1.0"    Faktor für GJL, Tabelle 3.3.2 (default: K_NL_Druck=1.0)
```


```
<Material
    name="42CrMo4_QT"
    number="1.7225"
    type="Stahl"
    R_m_N="1100"
    R_p_N="900"
    A="10"
    f_W_sigma="0.45"
    f_W_tau="0.577"
    a_G="0.5"
    b_G="2700"
    a_R_sigma="0.22"
    R_m_N_min="400"
    a_M="0.35"
    b_M="-0.1"
    N_D_sigma="1e6"
    k_sigma="5"
    N_D_II_sigma="1e6"
    k_II_sigma="5"
    K_NL_E="1.0"
    K_A="1.0"
    d_eff_N_m="16"
    a_d_m="0.3"
    d_eff_N_p="16"
    a_d_p="0.4"
    f_sigma="1.0"
    f_tau="0.577"
    K_NL_Zug="1.0"
    K_NL_Druck="1.0"
/>
```

# FKM-Verification Input

- For the FKM strength verification create input file **.xml**. The mesh, the amplitude stress, medium stress and material database that were previously generated are specified.
- Optionally, the maximum stress can be specified for static verification. By default, the static verification is performed with the high stress and the low stress (from the mean and amplitude stress).
- R_z: average roughness Sec. 4.3.1.4
- d_eff: effective diameter Sec. 3.2.1.4
- K_V: Surface treatment factor Sec. 4.3.3
- K_S: Coating factor Sec. 4.3.4
- N: Number of cycles Sec. 4.4.3.4
- J_D: Total safety factor for fatigue strength verification Sec. 4.5.3
- J_ges: Total safety factor for fatigue strength verification Sec. 3.5.2

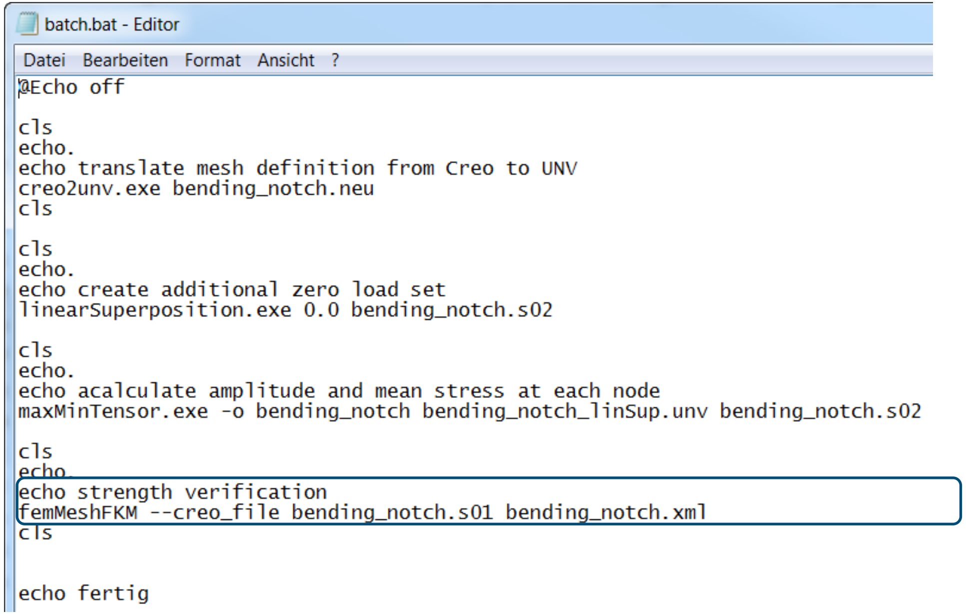
bending_notch.xml - Editor

Datei Bearbeiten Format Ansicht ?

```
<?xml version="1.0" ?>
<FkmInput>
    <GeneralSettings
        mesh_file = "bending_notch.unv"
        amplitude_stress_file = "bending_notch_amp.unv"
        mean_stress_file = "bending_notch_mean.unv"
        material_database_file="fkm_materials.xml"
    />
    <DefaultFKMSettings
        material_name="42CrMo4_QT"
        R_z="25"
        d_eff="10"
        K_V="1.0"
        K_S="1.0"
        N="1e6"
        j_D="1.5"
        j_ges="1.5"
        K_p=""
    />
</FkmInput>
```

- K_p: Plastic notch factor Sec. 3.3.1.1, table 1.3.2 for cross-sectional shap stresses or estimation according to (3.3.10) and (4.3.3) if no data is given

# FKM Strength verification

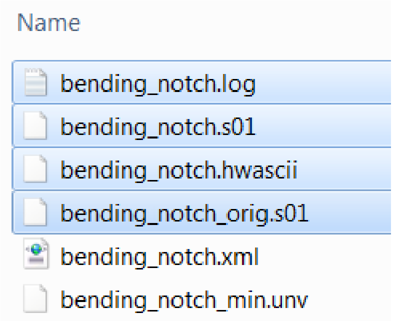


batch.bat - Editor

Datei Bearbeiten Format Ansicht ?

```
@Echo off

cls
echo.
echo translate mesh definition from Creo to UNV
creo2unv.exe bending_notch.neu
cls

cls
echo.
echo create additional zero load set
linearSuperposition.exe 0.0 bending_notch.s02

cls
echo.
echo acalculate amplitude and mean stress at each node
maxMinTensor.exe -o bending_notch bending_notch_linSup.unv bending_notch.s02

cls
echo.
echo strength verification
femMeshFKM --creo_file bending_notch.s01 bending_notch.xml
cls


echo fertig
```

- The program starts with the input file as parameter
- With the same name as the input file, femMeshFKM creates a file with the extension **.hwascii** and a logfile **.log**
- When running femMeshFKM, you can overwrite an existing stress file ***.s0x** with the result of the strength verification by using the **--creo_file** option. The original stress file is saved with the suffix **_orig.s0x** in the same folder.



# Postprocessing in Creo

- When running femMeshFKM, you can overwrite an existing stress file ***.s0x** with the result of the strength verification by using the **--creo_f**ile option. The original stress file is saved with the suffix **_orig.s0x** in the same folder.

**femMeshFKM.exe --creo_file MyAnalysis.s01 config.xml**

- The ***.s0x** file created in this way will be copied to the unused load case **01_FKM** of Creo Simulate and can be displayed with the Simulate Postprocessor.
- The results of the strength verification are written to the file as stresses and strains. The table indicates the translation:

**Cyclic analysis**

| | | |
|---|---|---|
| $s_{xx}$ | a_BK_sigma_v | cyclic degree of utilization |
| $s_{yy}$ | j_act_cyclic | actual factor of cyclic safety |
| $s_{xy}$ | N_allow | admissible number of load cycles |
| $s_{zz}$ | s_a | amplitude stress |
| $s_{yz}$ | s_m | mean stress |
| $s_{xz}$ | G | normalized stress gradient |

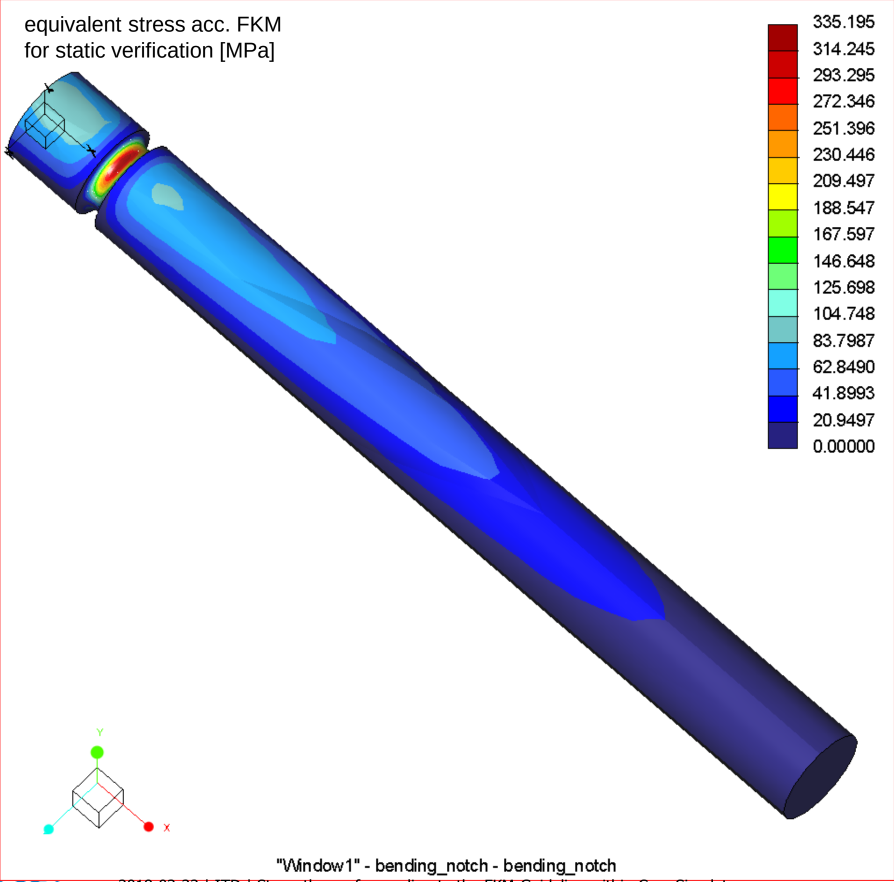
**Static analysis**

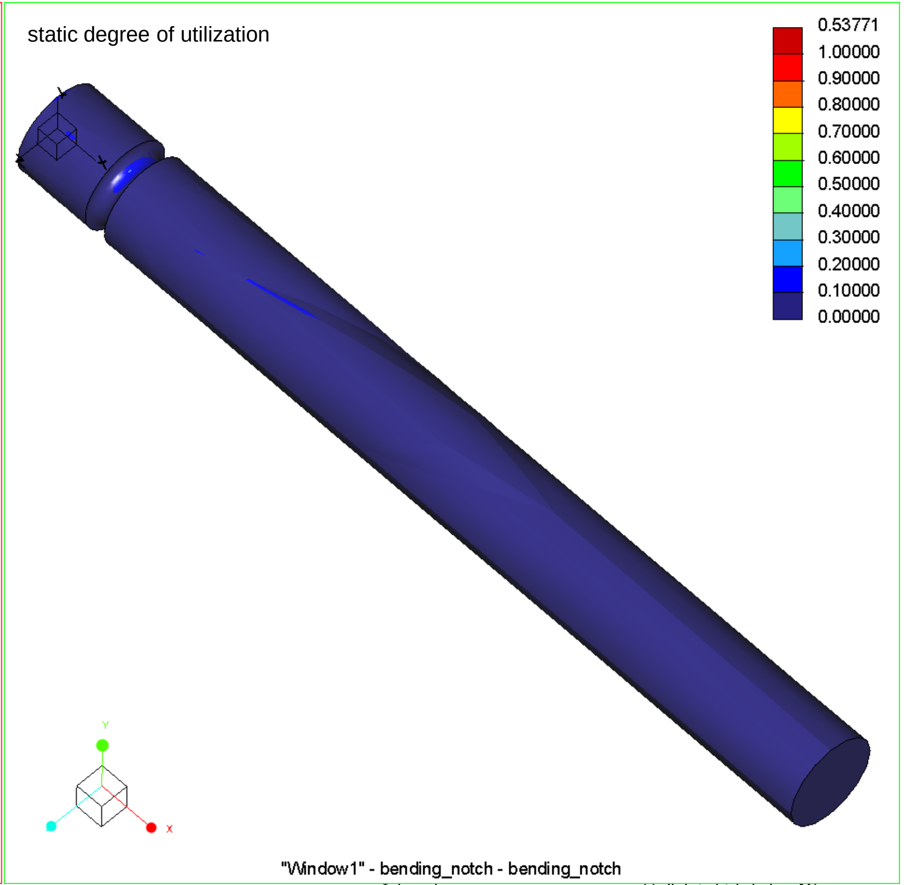
| | | |
|---|---|---|
| $\varepsilon_{xx}$ | a_SK | static degree of utilization |
| $\varepsilon_{yy}$ | j_act_static | actual factor of static safety |
| $\varepsilon_{xy}$ | n_pl | plastic limit acc. to (3.3.7) |
| $\varepsilon_{zz}$ | K_p | plastic limit acc. to (3.3.8) |
| $\varepsilon_{yz}$ | sigma_v | equivalent stress for static analysis |
| $\varepsilon_{xz}$ | a_BK_sigma_1 | major principle stress for static analysis |

# 07

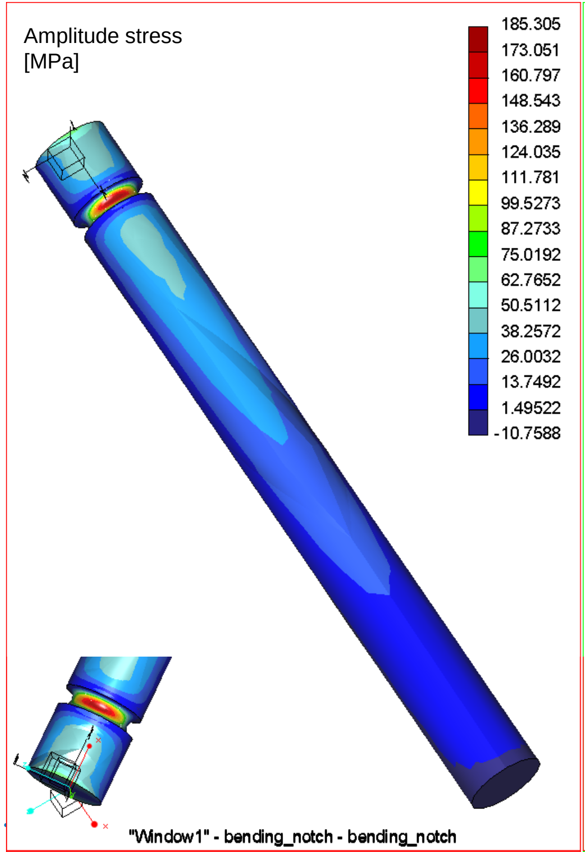
# Postprocessing in Creo

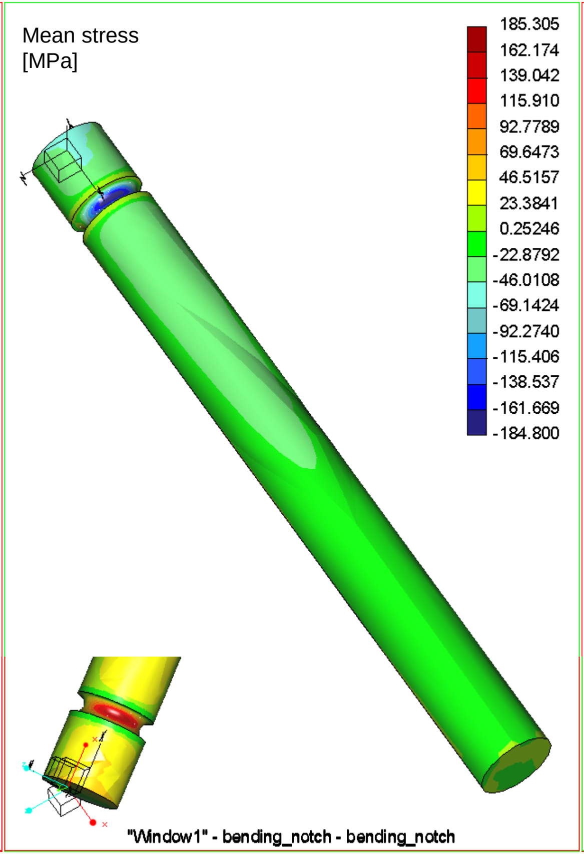
# Postprocessing in Creo
## Static Verification

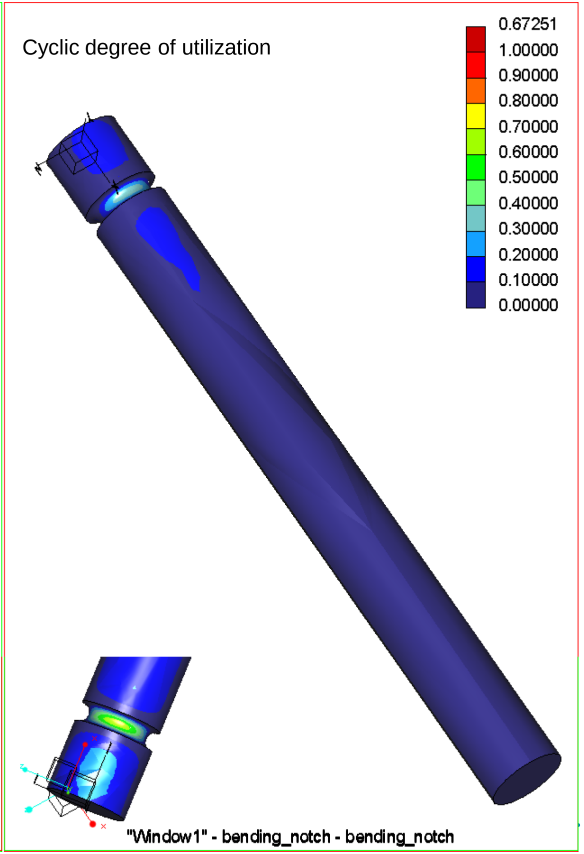


# Postprocessing in Creo
## Fatigue strength verification – swelling load



# Postprocessing in Creo
## Fatigue strength verification – alternating load

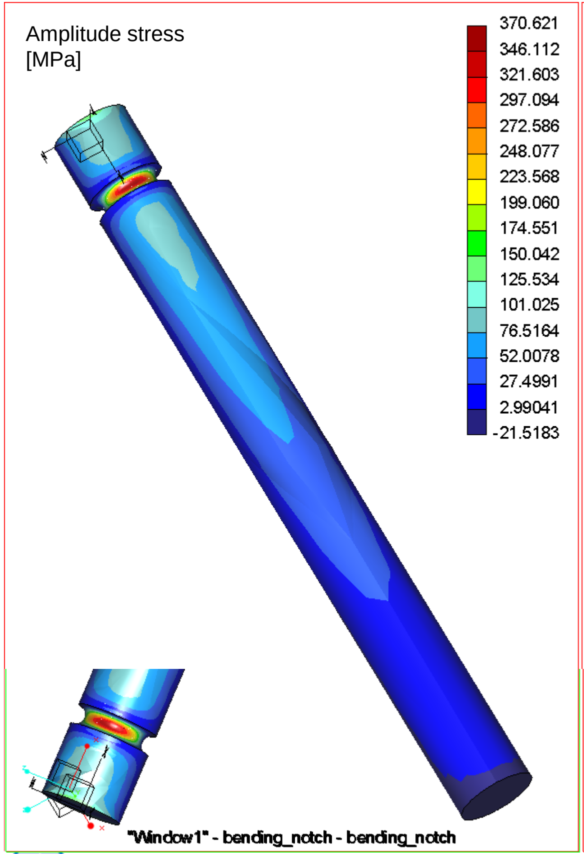
- And the same for an alternating load?
- Create instead of the zero load set an additional load set with inverted sign

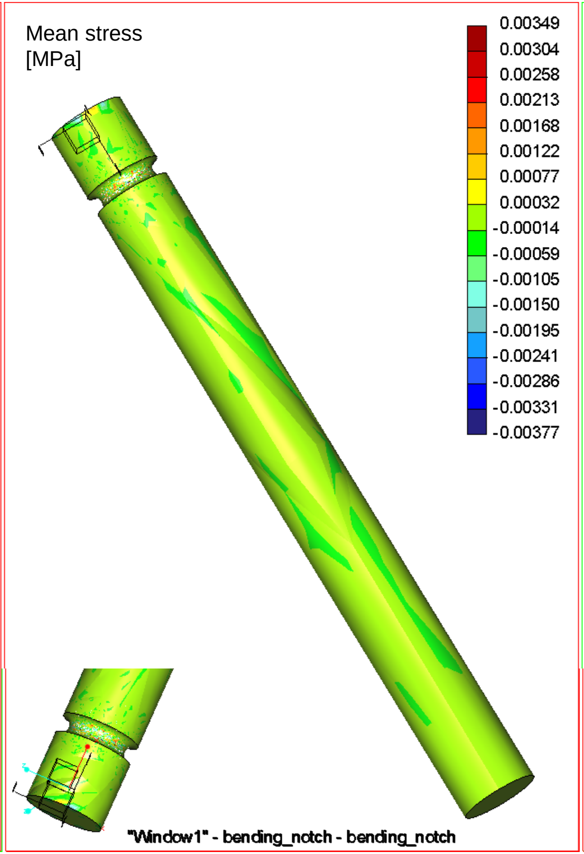
batch.bat - Editor

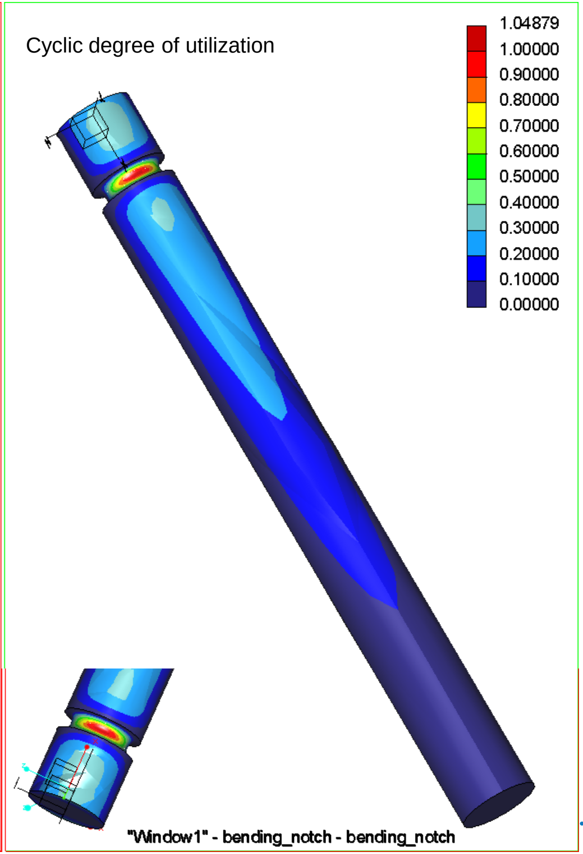
Datei Bearbeiten Format Ansicht ?

```
@Echo off

cls
echo.
echo translate mesh definition from Creo to UNV
creo2unv.exe bending_notch.neu
cls

cls
echo.
echo create additional zero load set
linearSuperposition.exe -1.0 bending_notch.s02

cls
echo.
echo acalculate amplitude and mean stress at each node
maxMinTensor.exe -o bending_notch bending_notch_linSup.unv bending_notch.s02

cls
echo.
echo strength verification
femMeshFKM --creo_file bending_notch.s01 bending_notch.xml
cls


echo fertig
```

# Postprocessing in Creo
## Fatigue strength verification – alternating load



# Documentation of the verification

| 01_Creo | 02_CreoUNV | 03_MaxMinTensor | 04_FKM |
|---|---|---|---|
| **Used Simulate results** | **With creo2unv and linSup created data** | **With MaxMinTensor created data** | **With femMeshFKM created data and according input data** |
| bending_notch.neu<br>bending_notch.s02<br>bending_notch_orig.s01 | bending_notch.unv<br>bending_notch_linSup.hwascii<br>bending_notch_linSup.unv | bending_notch.hwascii<br>bending_notch_amp.unv<br>bending_notch_max.unv<br>bending_notch_mean.unv<br>bending_notch_min.unv | batch.bat<br>bending_notch.log<br>bending_notch.s01<br>bending_notch.xml<br>fkm_materials.xml |

- Next to the Simulate results folder, create a **FKM** folder with the following subfolders

  01_Creo
  02_CreoUNV
  03_MaxMinTensor
  04_FKM

- Create and archive slides with the most important results

# Restrictions, open points

- Currently only 3D elements are supported. 2D elements and 1D elements (beams) are not supported.
- The use of the ASCII format is significantly slower in the calculation of the analysis and in the post-processing than the standard binary format.
- FemMeshFKM is used outside of Simulate without an own user interface.
- The viewing of the deformed state is not possible for the results of femMeshFKM.

# Thank you for participation