FEMFAT 2021 News and More

ECS Simulation Conference 2021

Gaier, ECS

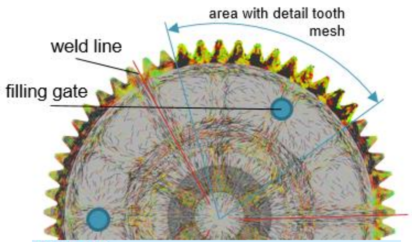
History of FEMFAT Releases, New Versioning

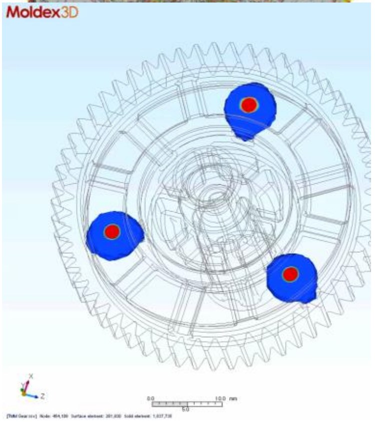
- About two new versions are released every year.
- Major release = **5.4**, Service release = **5.4a**, Bug fixes and new Features = **5.4.1**
- **FEMFAT 2021** (instead FEMFAT 5.5) will be released Q3 2021!

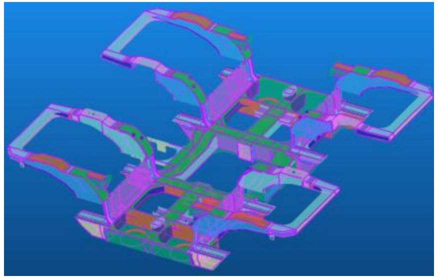
New Features FEMFAT 5.4.1 released June 2020 (1/2)

- Highlights
  - SPECTRAL: Assessment of short fiber reinforced plastics
  - Search local maximum/minimum values of damage/safety and output as group and/or table (for base material)
  - Support of multiple ABAQUS odb-versions (High efficiency without upgrade of the odb-file)

- VISUALIZER and WELD
  - Visualization of node characteristics
  - Add and rearrange multiple subwindows
  - Support of χMCF file format V3.0 for geometry-based weld definitions
  - Weld Seam Scanner improvements
  - Mark welding seams as checked/unchecked
  - Extended stress interpolation and averaging possibilities based on element nodal stresses for weld elements



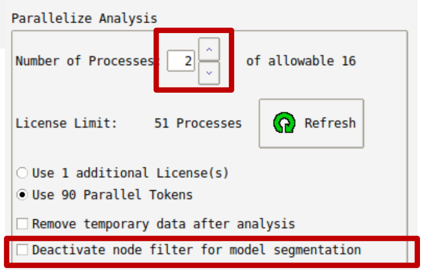




New Features FEMFAT 5.4.1 released June 2020 (2/2)

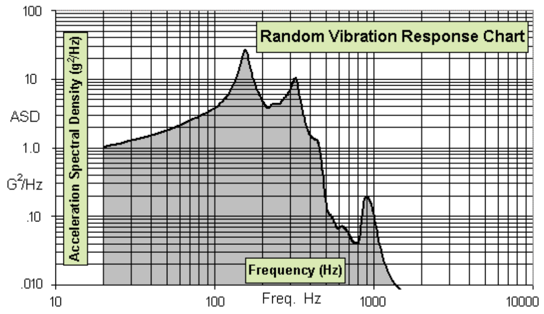
- **Methods**
  - Improved stress gradient influence parameters for short fiber reinforced plastic
  - SPECTRAL: Support of transfer functions for non-equidistant frequency steps
  - Support of RBE2-spider as inner elements of SPOT nugget
- **Interfaces**
  - Multi-material-file (ffd) for import of all needed materials with one click
  - Support of „Include“ with FE-structure interfaces for ABAQUS & NASTRAN
  - ABAQUS: Support of „centroidal stresses“ from odb
  - Dynamic memory allocation for groups from Medina
  - Support of H3D files from Radioss (HVTrans generated)
- **Textual and Graphical Output**
  - Output of used equivalent stresses in report file for ‚automatic‘ setting
  - Display of load spectra in the S-N diagram for „Detailed Results Group“
  - Rainflow-matrix Viewer: „Open“ menu to load a saved matrix
  - Rainflow-matrix Viewer: Pick the entries to show values
- **Performance**
  - PARALLEL: model splitting without filters active (speedup the splitting)
  - ChannelMAX: accelerated analysis because of improved stress superposition
  - Scratch-files for TransMAX and HEAT in HDF5 format (same as ChannelMAX)

```
MAT===
# Comment
    -1
   218
Dataset 218
    -1
    -1
   223
Dataset 223
    -1
MAT===
# Comment
    -1
   218
Dataset 218
    -1
    -1
   223
Dataset 223
    -1
```

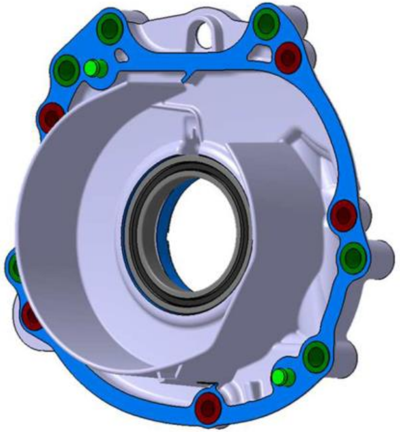


SPECTRAL: Assessment of Short Fiber Reinforced Plastics

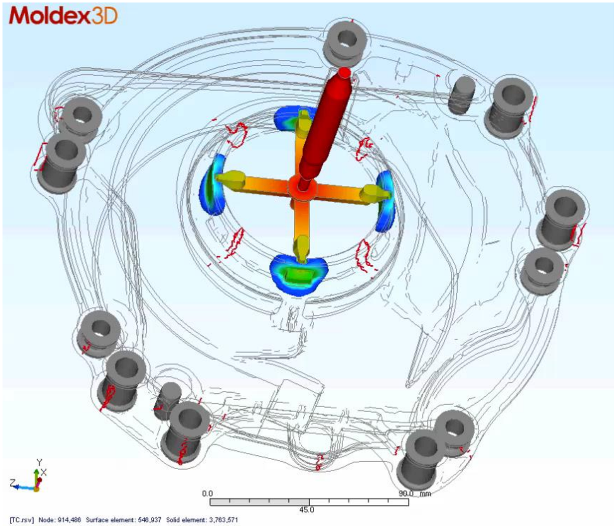
MAGNA



Load PSDs (Power Spectral Density)



Gear Box Cover



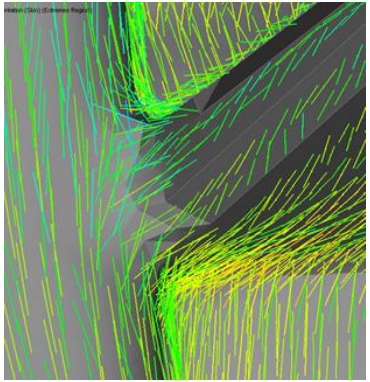
CAD

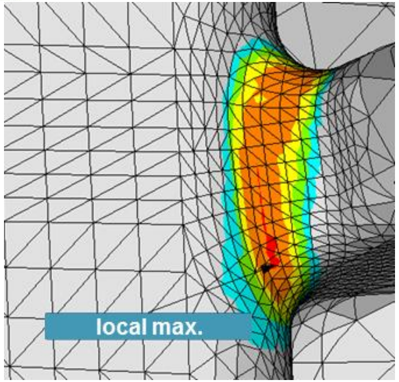
Filling Simulation (MoldFlow, Moldex3D)

Fiber Orientation & Material Mapping (Digimat-MAP)

Modal FE- Analysis Frequency Response (Nastran, ABAQUS)

Fatigue Analysis FEMFAT spectral

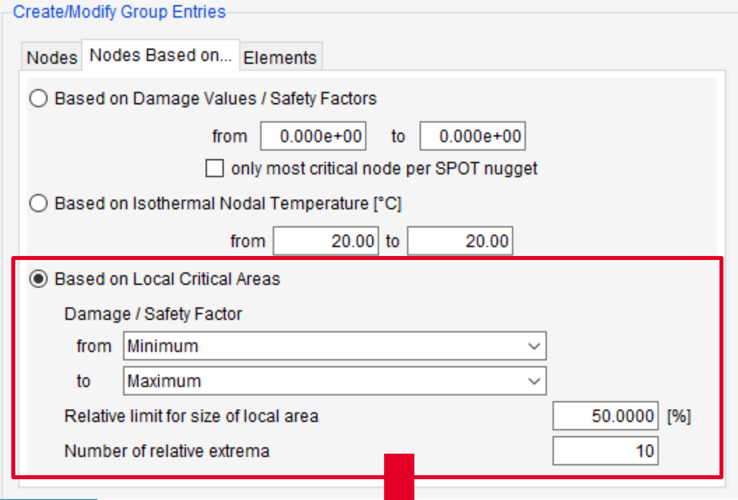


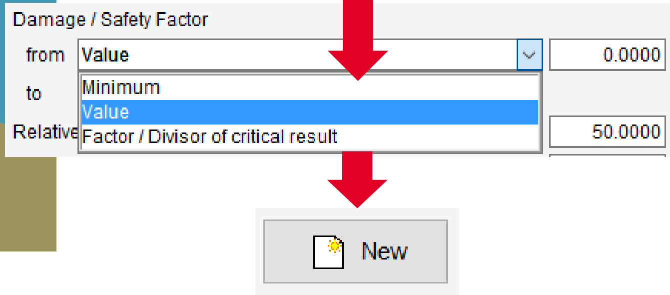


Grouping Function Based on Local Maximum/ Minimum Values of Damage/Safety (for Base Material)

INPUTS:

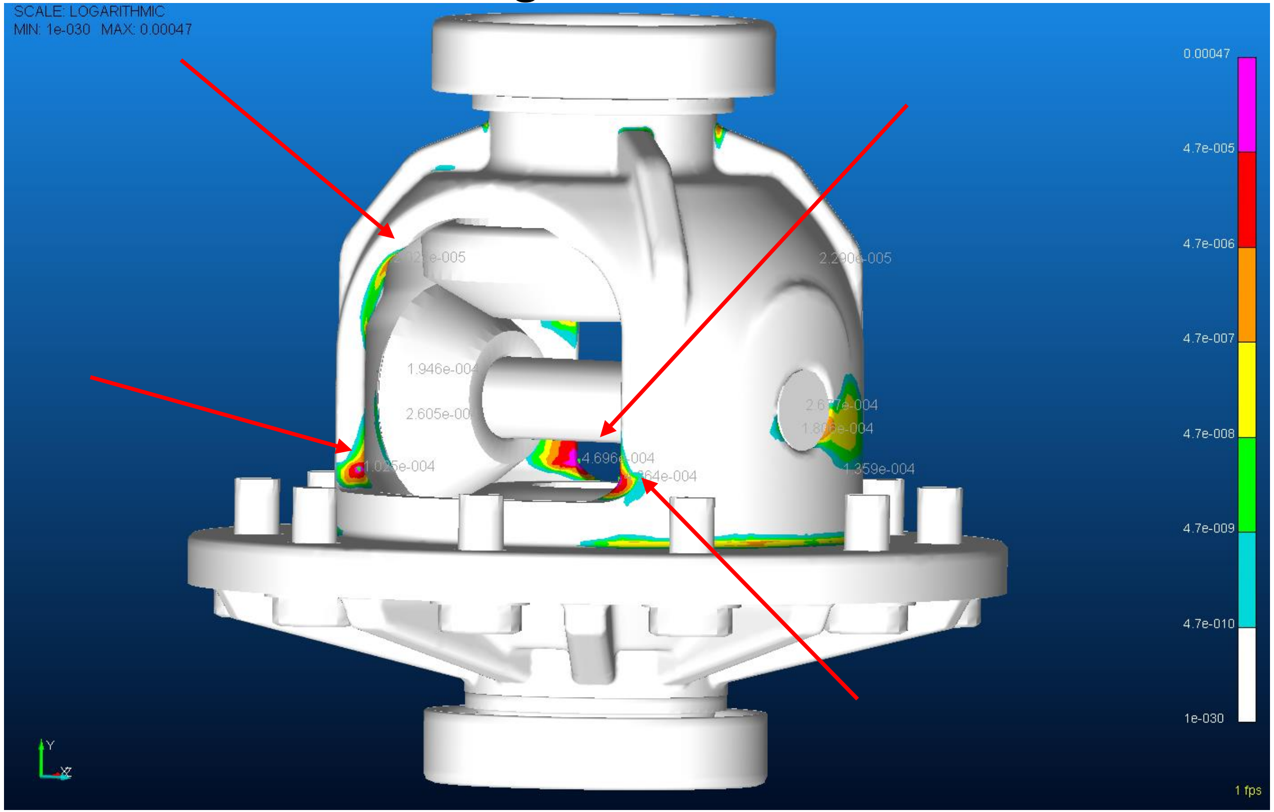
- Define range for damage / safety factor to be considered
- Relative filter limit for critical area size: e.g. 50% of local extrema
- Number of required extrema: e.g. 10





Display of Nodes with Local Critical Areas in VISUALIZER

- Nodes with relative damage maxima



Output Table of Nodes with Local Critical Areas in the Report File

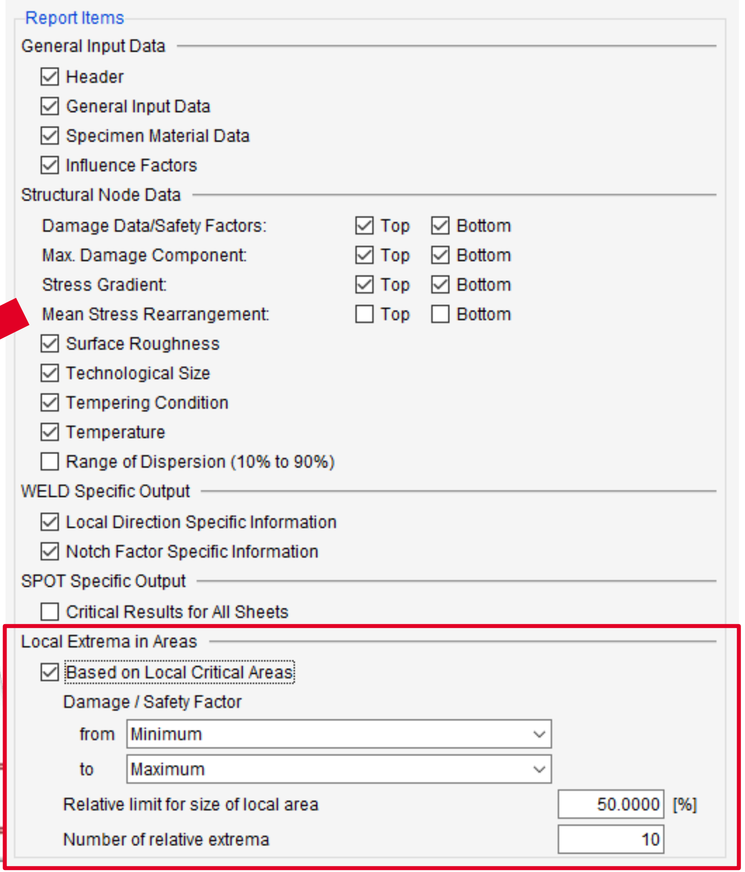
The entries are analogous to the group menu and are treated the same internally.

However, all variables are decoupled and independent of the entries in the group menu.

Critical Locations:

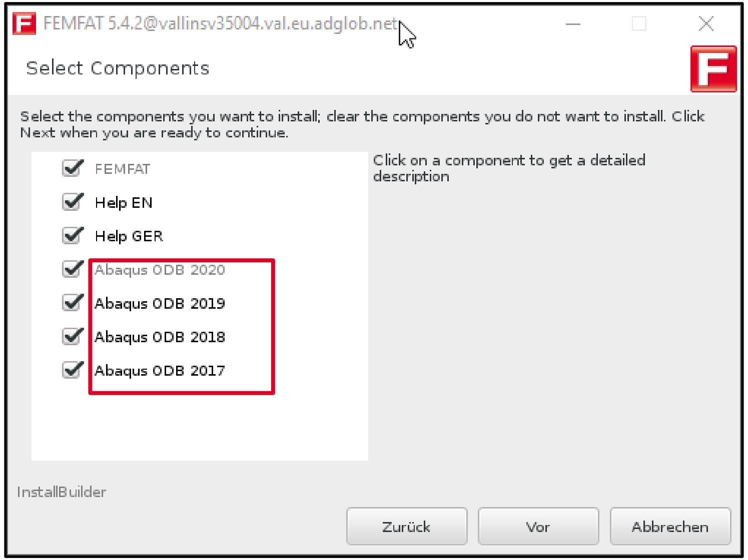
Location	NodeLab	Damage	Result Pos.	Rel.Str.Grad	Stress Ampl.	Mean Stress	LocFatigLim
1	1835	3.209e+04	top / surf	0.1144	1213.6919	-45.8876	321.0130
2	1851	1.378e+03	top / surf	0.1287	888.5620	31.1083	302.5340
3	8991	3.057e+02	bot / trans	0.3578	795.8170	10.7916	307.4100
4	9020	2.913e+02	top / surf	0.3342	801.9175	-10.2546	312.4611
5	1847	2.410e+01	top / surf	0.1292	642.0276	-23.5760	315.6582
6	2106	6.735e-02	top / surf	0.1112	368.7176	15.6678	306.2397
7	2124	3.477e-02	top / surf	0.1021	347.4174	15.2929	306.3297
8	2160	1.115e-02	top / surf	0.0969	313.7703	14.0856	306.6195
9	2149	7.136e-03	top / surf	0.1436	303.0617	14.4143	306.5406
10	2178	5.667e-03	top / surf	0.1222	299.1216	13.2567	306.8184

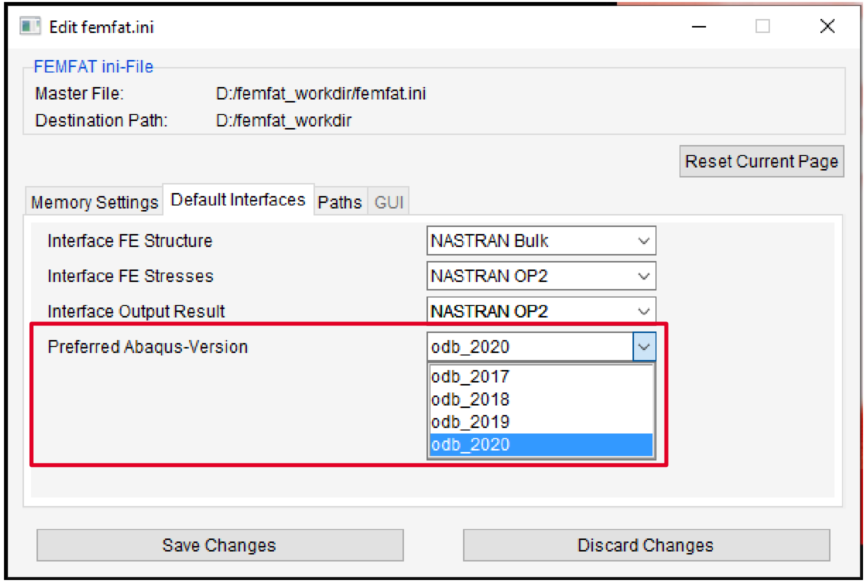
End - Critical Locations



Support of Multiple ABAQUS odb-Versions

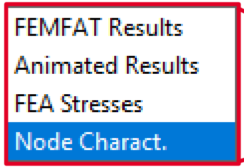
- FEMFAT recognizes automatically the version of an ABAQUS odb-File
- No time-consuming upgrade of the odb-file is necessary, if the version is installed
- During the installation process the needed ABAQUS versions can be selected
- At FEMFAT start preferred version can be selected in the drop-down box

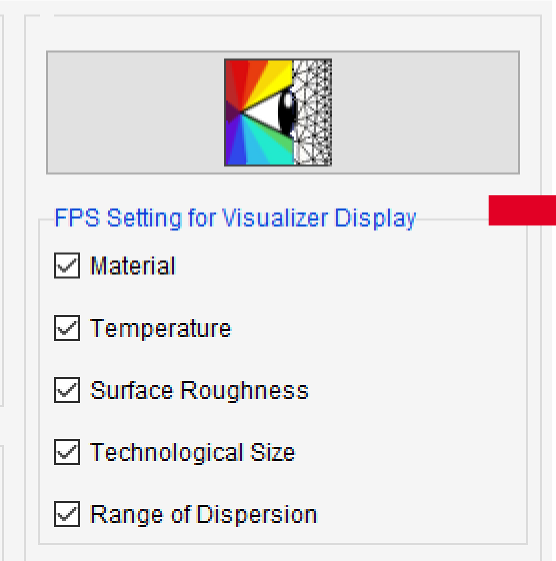


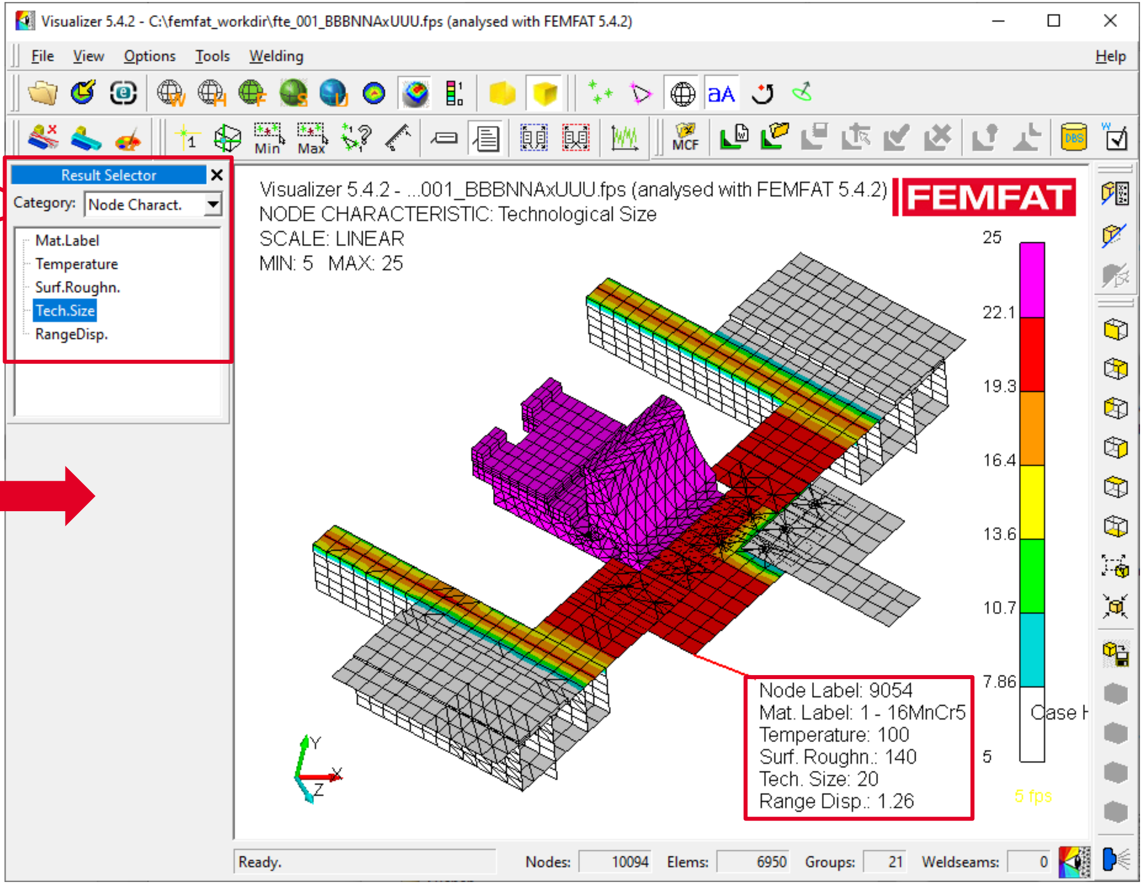


VISUALIZER: Visualization of Node Characteristics

Desired properties can be selected.



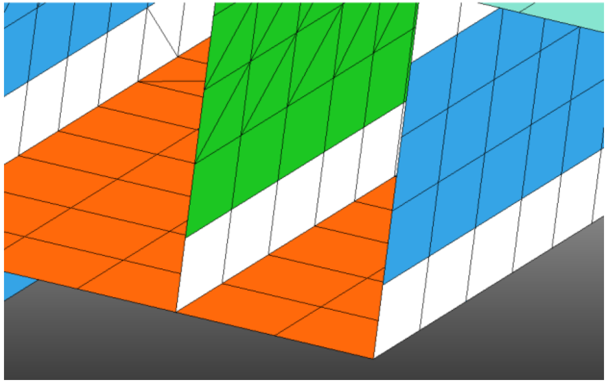


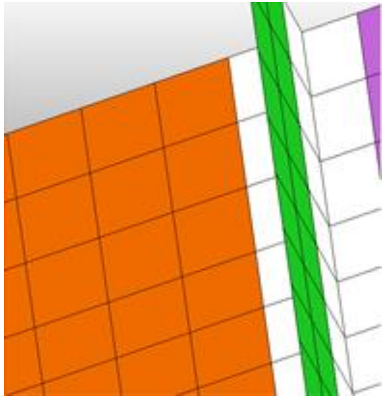


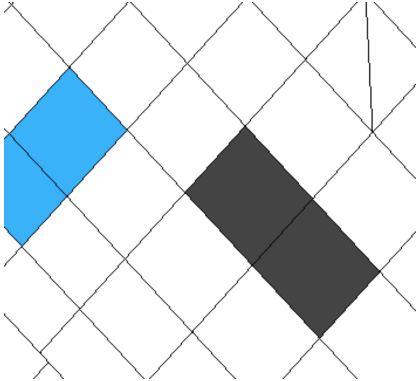
VISUALIZER: Visualization of Material Assignment

Special for element-based material display:

- Materials are displayed analogous to the PIDs
- Elements whose **nodes** have all been assigned **the same material** are colored in the corresponding material color
- Elements whose **nodes** have been assigned **different materials** are **colored white**
- Elements whose **nodes** have **not been assigned materials** are **colored gray**



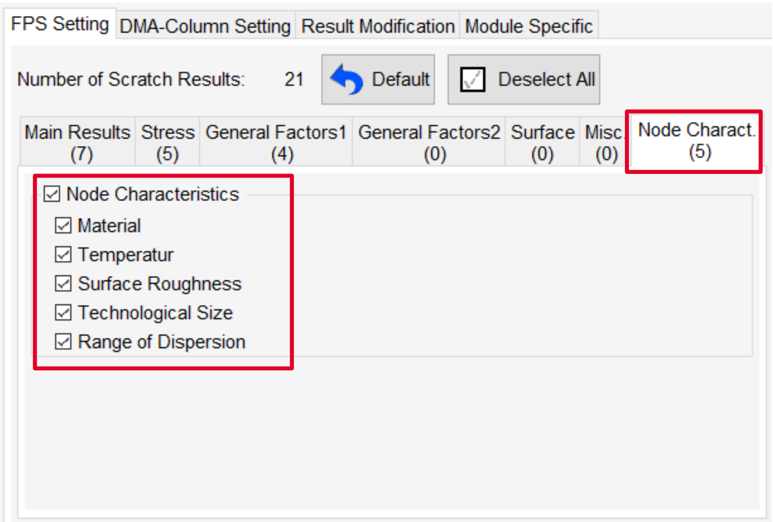


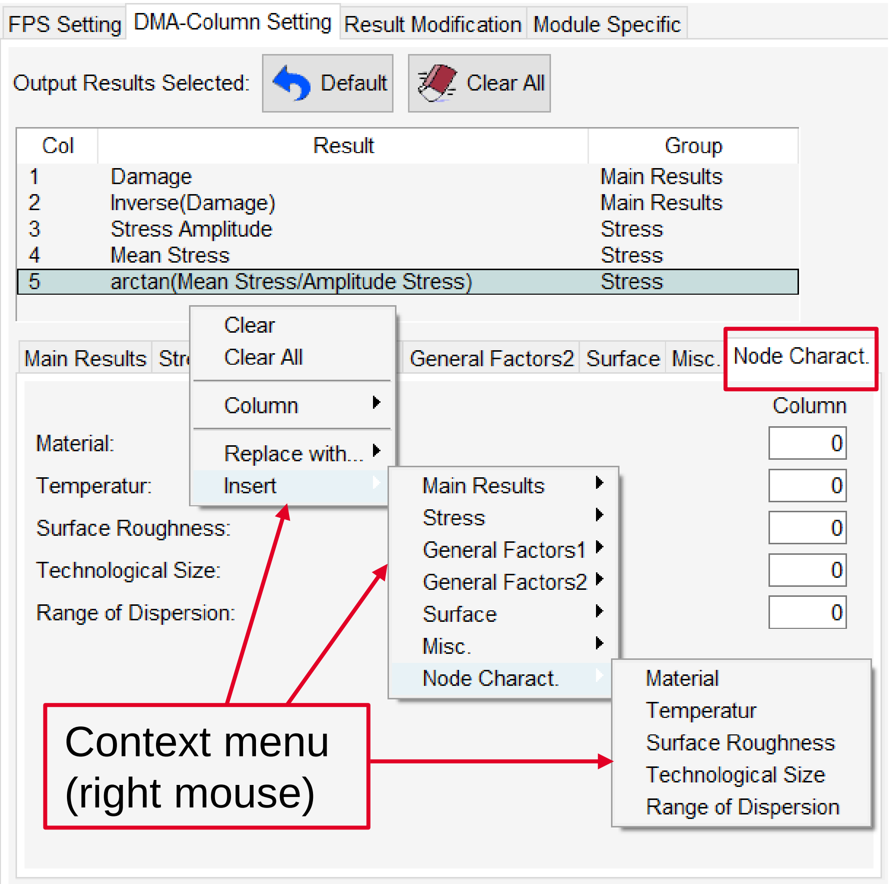


Output of Node Characteristics in the dma-File

Desired entries in the dma-file can be selected.

In the FPS Setting and DMA-Column Tabs the Node Characteristics and the functions "Replace" and "Insert" were extended accordingly.

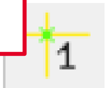




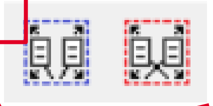
VISUALIZER: Add and Rearrange Multiple Subwindows

1. Select nodes
2. Add all subwindows
3. Rearrange (according to node or current positions)

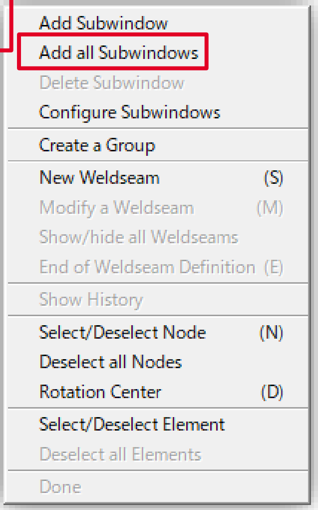
1.

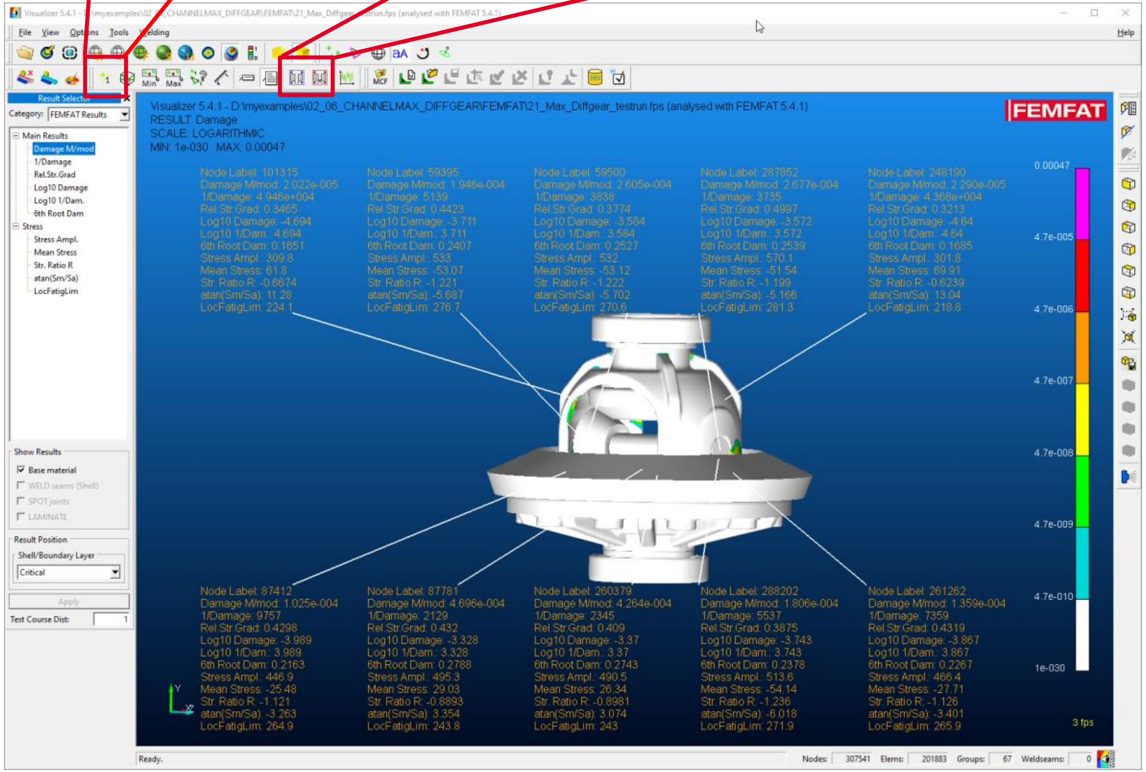


3.



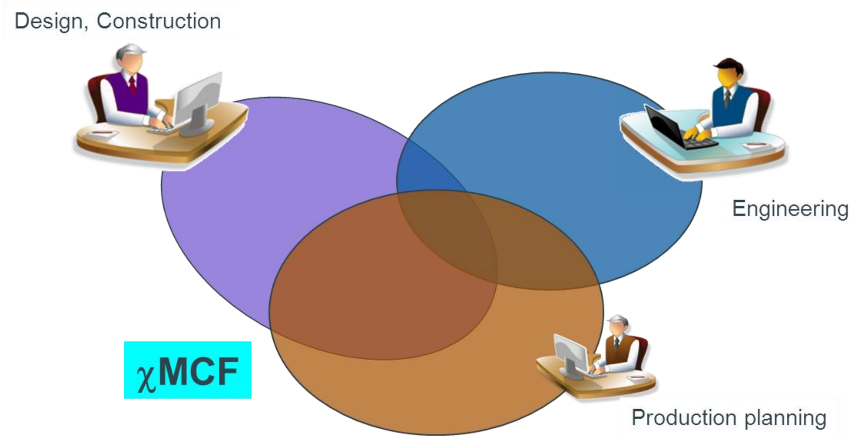
2.

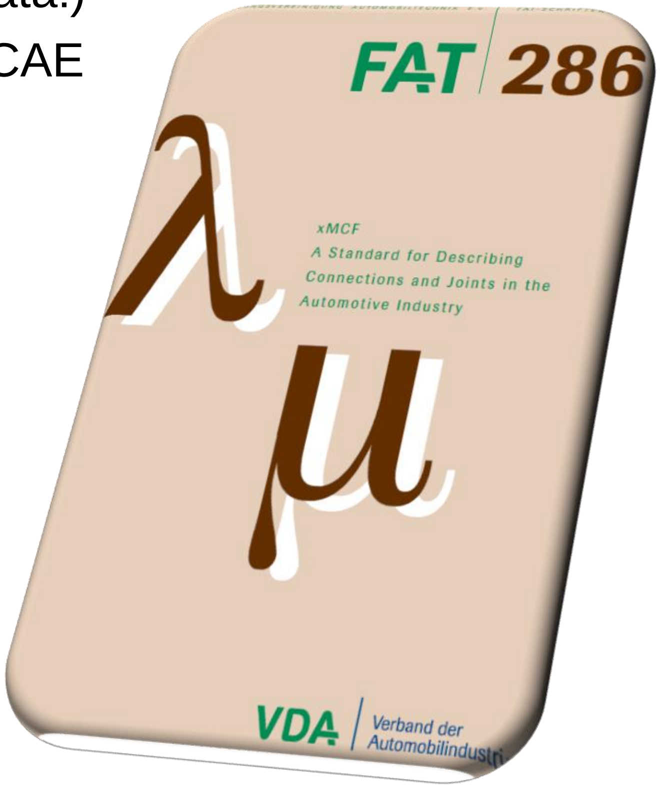




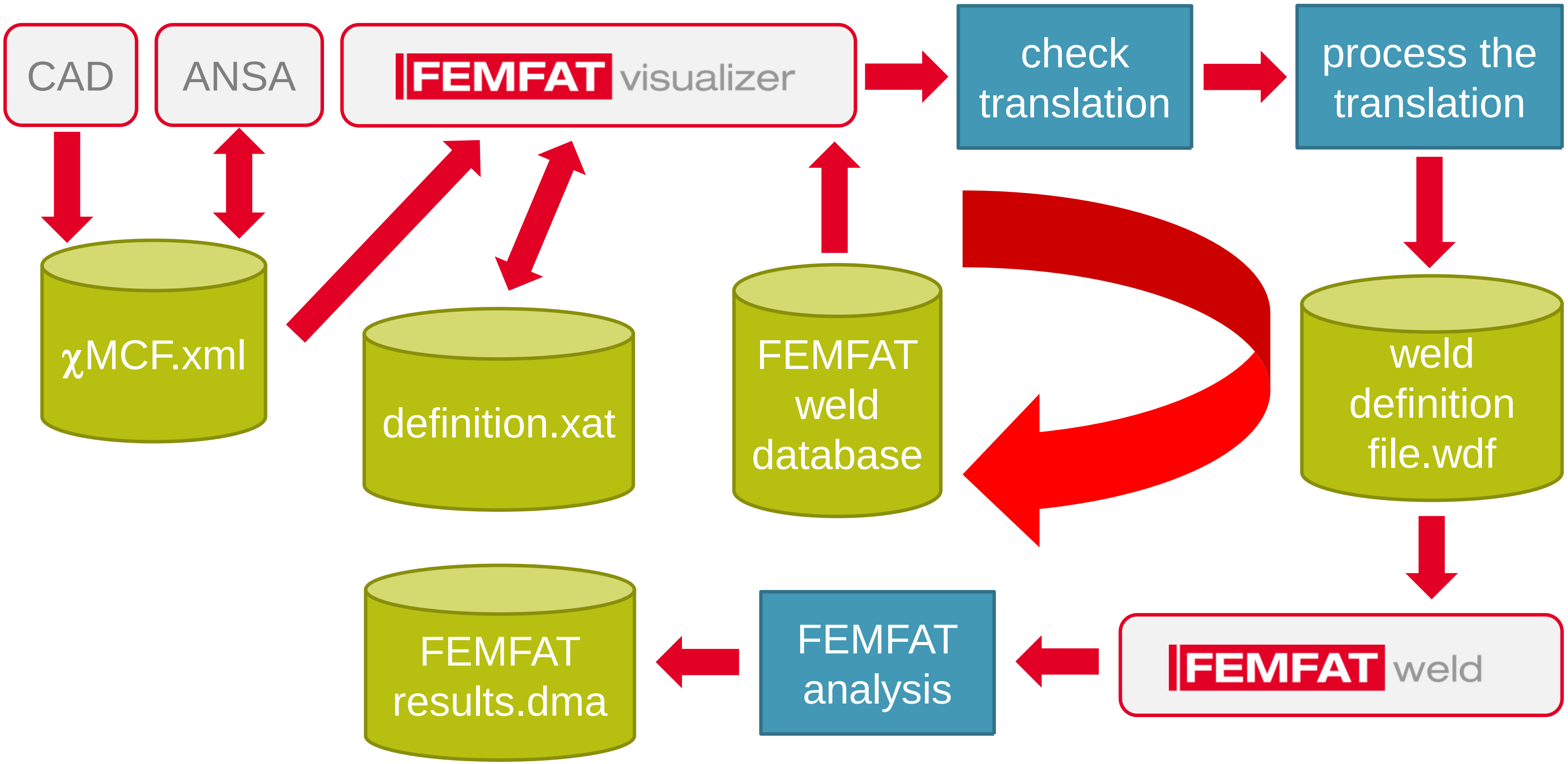
The Idea behind χMCF

- χMCF is a standardized XML file format for the geometry-based definition of joints (no definition based on FE data!)
- Used for exchange of joint definitions between CAD, CAE and CAM, e.g. for
  - Automated assembly of batch-meshed parts
  - Automatic programming of welding robots



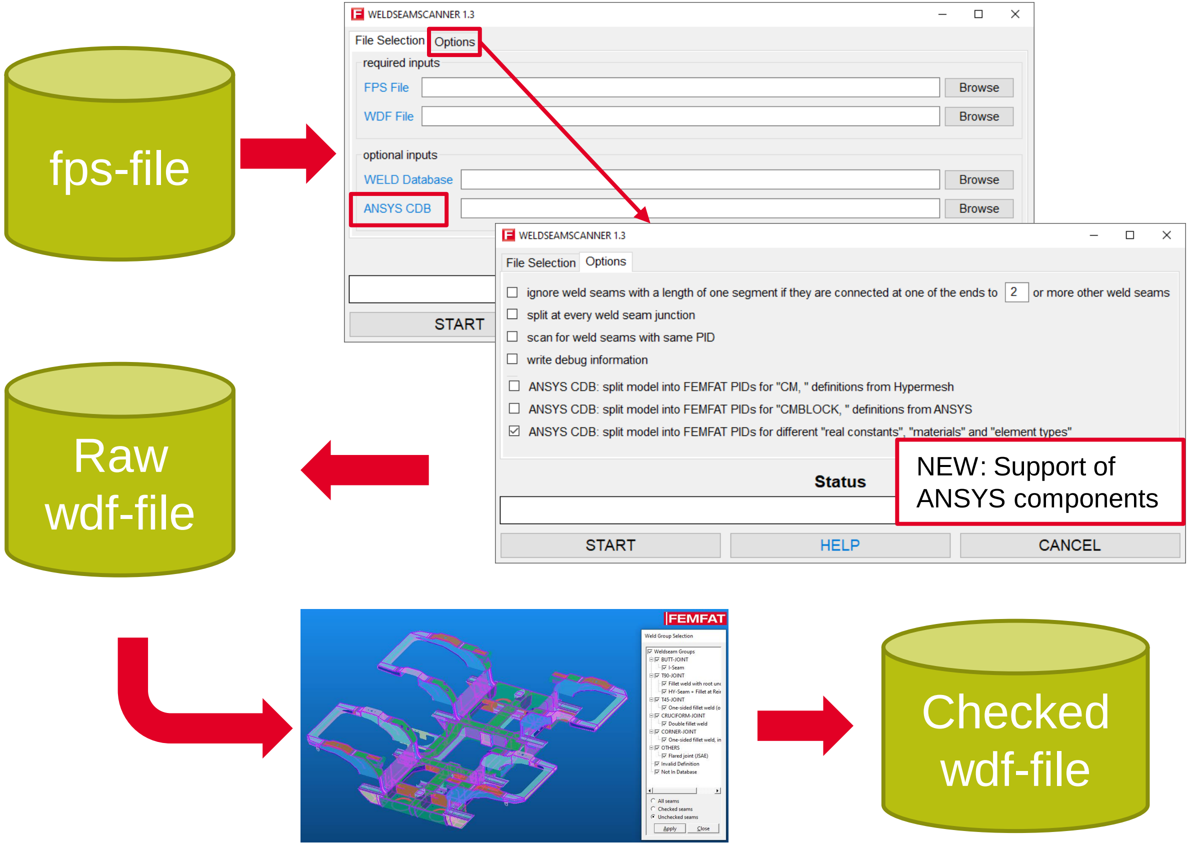


Workflow of Weld Definition via χMCF 3.0



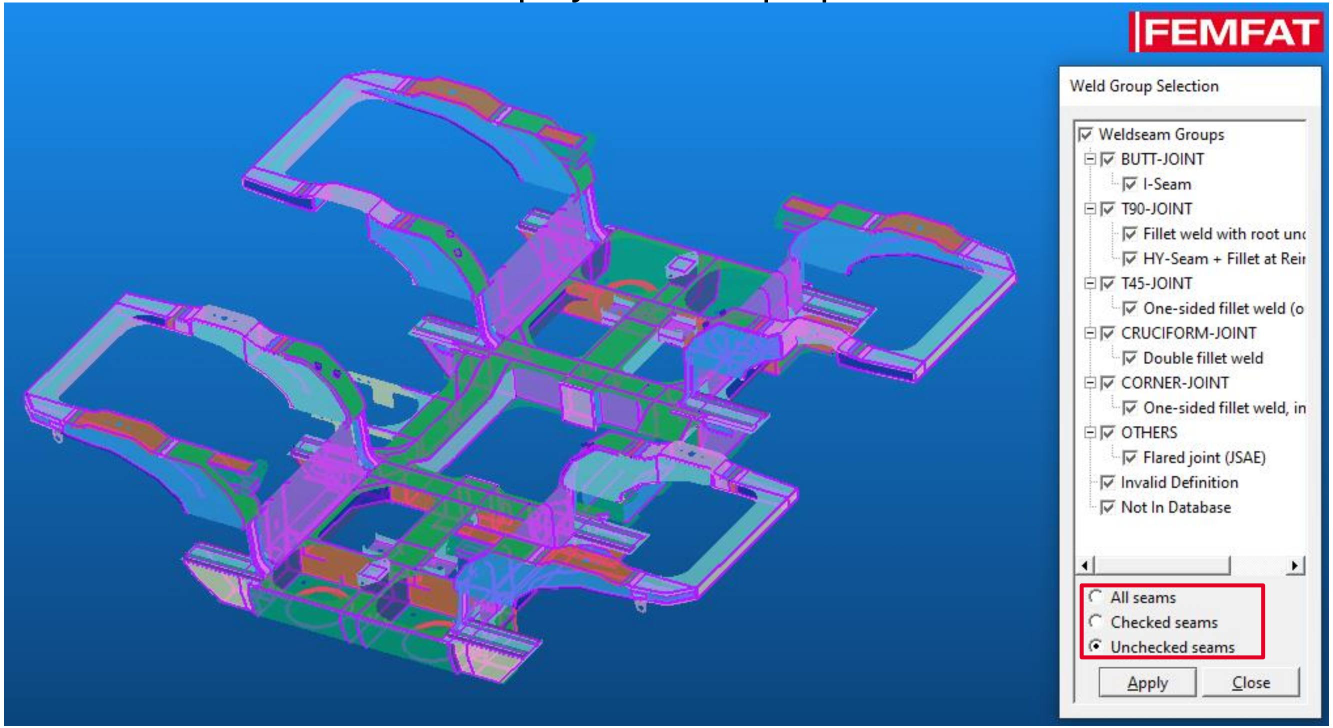
Workflow of Weld Definition using the Weld Seam Scanner, Improvements

- Generate a raw wdf-file with the Weld Seam Scanner (contains only paths, dummy node colors, no weld type)
- Raw wdf-file is read into VISUALIZER and here exact weld type, weld orientation, start/end nodes etc. can be checked and modified. The raw wdf-file cannot be used directly for a FEMFAT calculation!
- Save wdf-File
- Import in FEMFAT
- Start WELD analysis



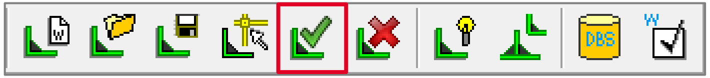
New Functionalities for the Checking of Welding Seams

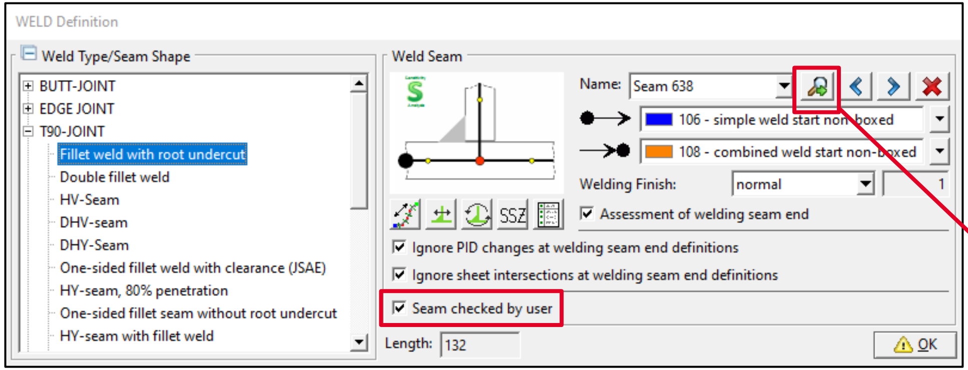
- Welding seams with missing information (e.g. welded side) that have been automatically generated (via xMCF, Weld Seam Scanner or automatic seam separation) are marked as "Unchecked" → Checking the weld definition is strongly recommended!
- In the VISUALIZER, such seams are displayed with a purple seam line.



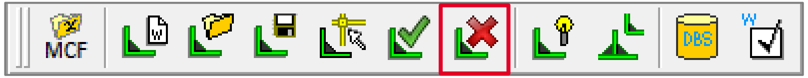
New Functionalities for the Checking of Welding Seams

- Mark a welding seam as checked/unchecked with the "Check/Uncheck Weldseam(s)" button from the "Welding" toolbar or with the "Seam checked by user" flag in the “WELD Definition" window.
- It is also possible to set several seams to checked with the frame selection.





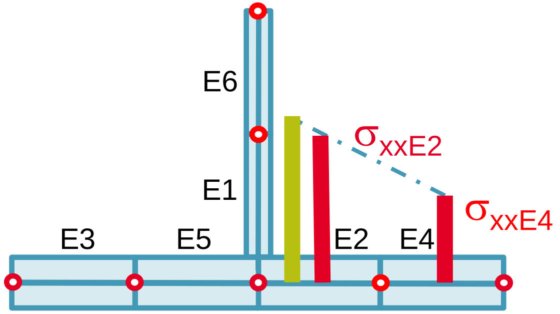
Also new:

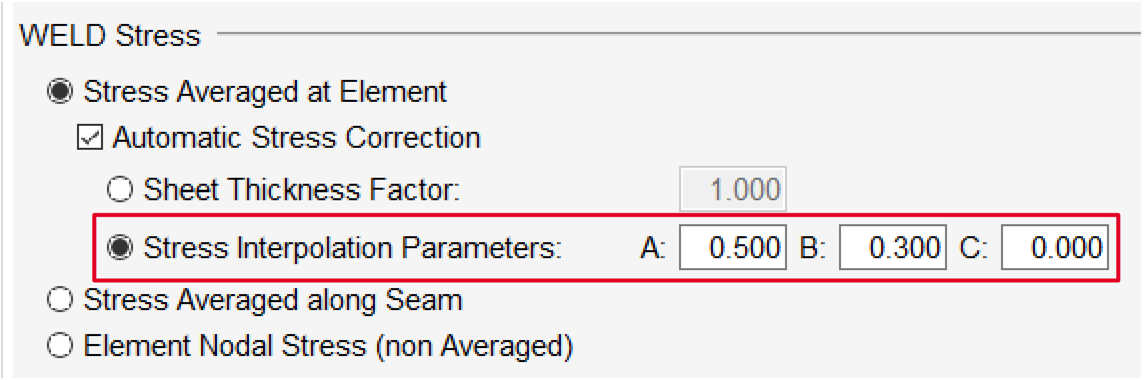


- The "Delete Weldseam(s)" - button now has a frame selection with which several visible seams can be deleted at the same time.
- The pivot point is automatically set to the center of the currently displayed seam

WELD: New Parameter in Stress Interpolation Method for Automatic Stress Correction

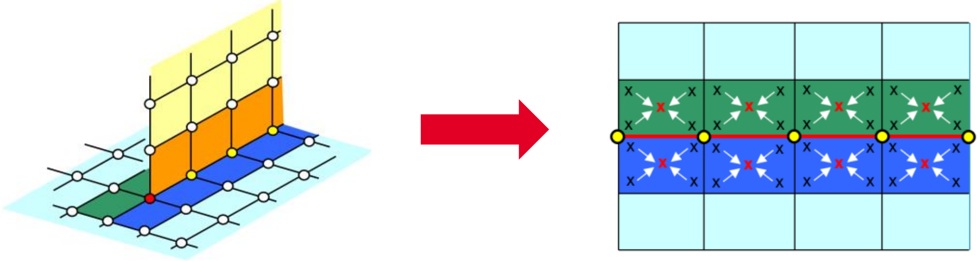
- Extended stress interpolation method: $D_{Assesspoint} = a\ t_{Neighbor} + b\ t_{Evalulated} + c$
- Stress interpolation parameter can be entered in the GUI and in the WELD database



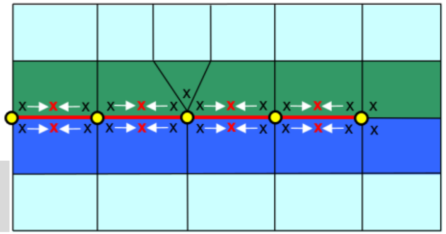


WELD: Extended Stress Averaging Possibilities

1) By default, element nodal stresses will be averaged

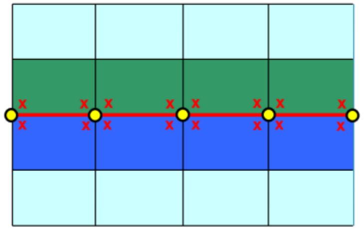


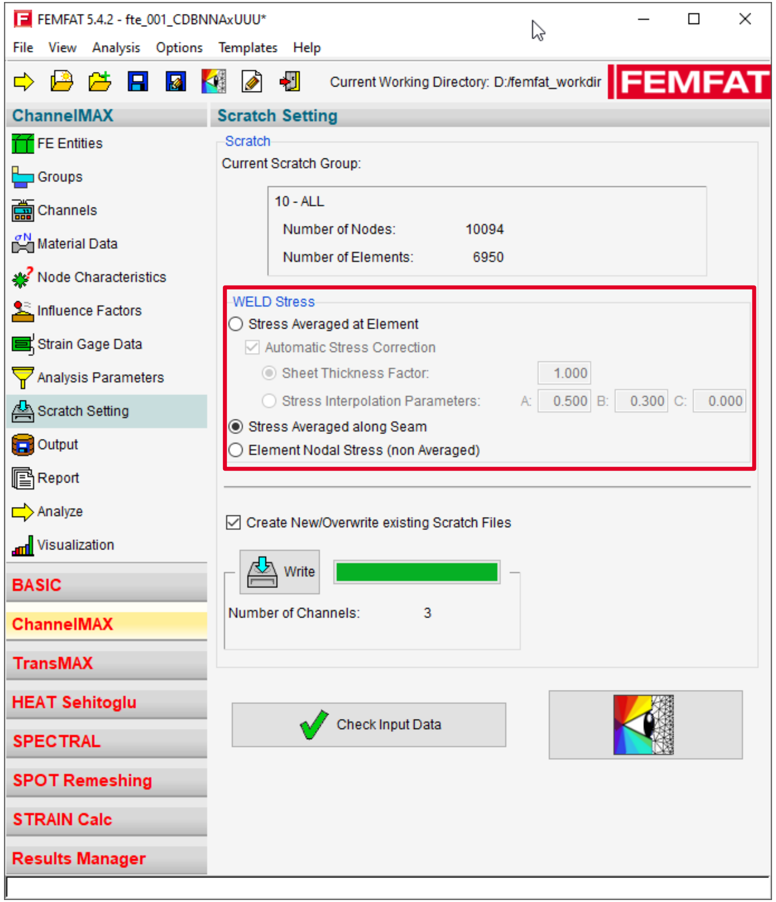
2) Averaging along the welding seam



x … Element nodal stresses (Cubic stresses)
x … Averaged stresses

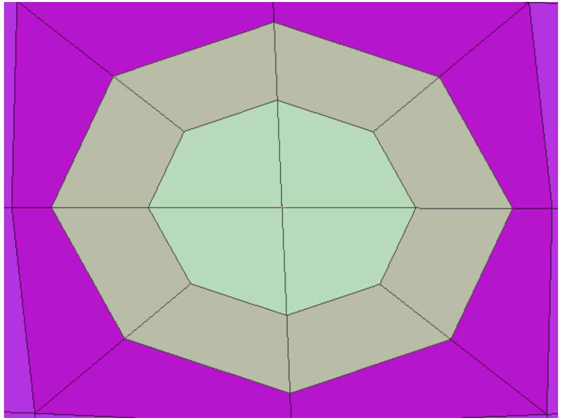
3) Usage of element nodal (cubic) stresses directly



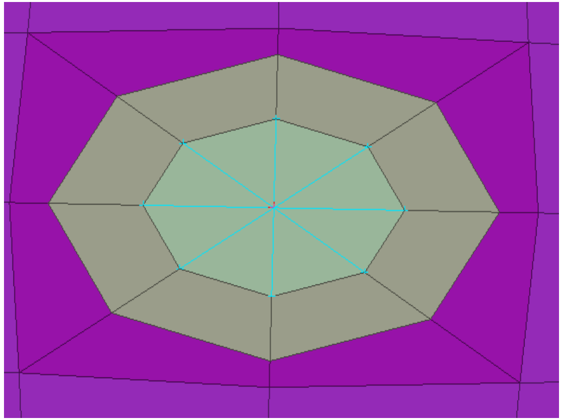


RBE2-Spider as Inner Elements of SPOT Nuggets

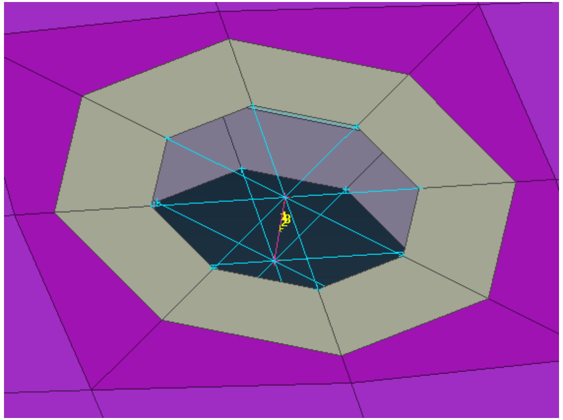
- Following modelling types will be supported in FEMFAT spot:
  - Added the RBE2 in the center of the nugget (modelling variant 1)
  - Removed inner shell elements and added RBE2 (modelling variant 2)



Standard model



Standard model + RBE2 (variant 1)

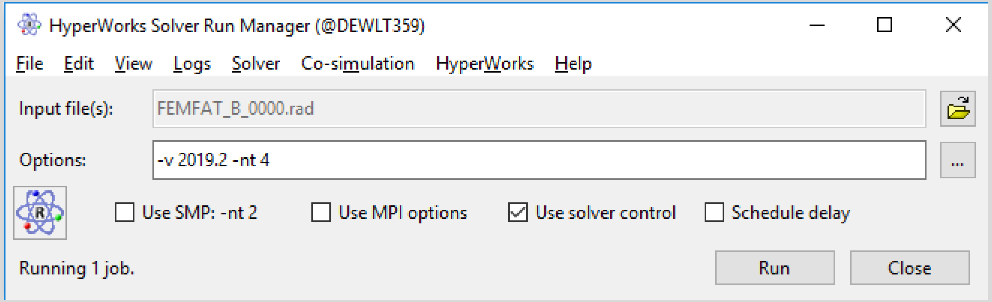


Replaced inner shell with RBE2 (variant 2)

Creation Options of H3D Files in Radioss

- Supported in FEMFAT:

Starting RADIOSS and getting H3D using HV-Trans:

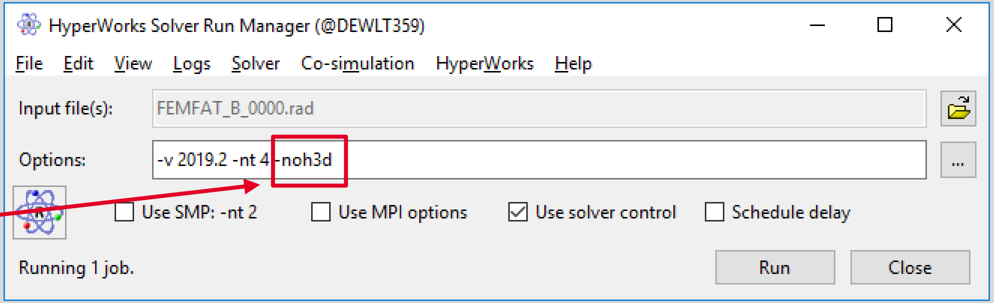


- **NOT** Supported in FEMFAT:

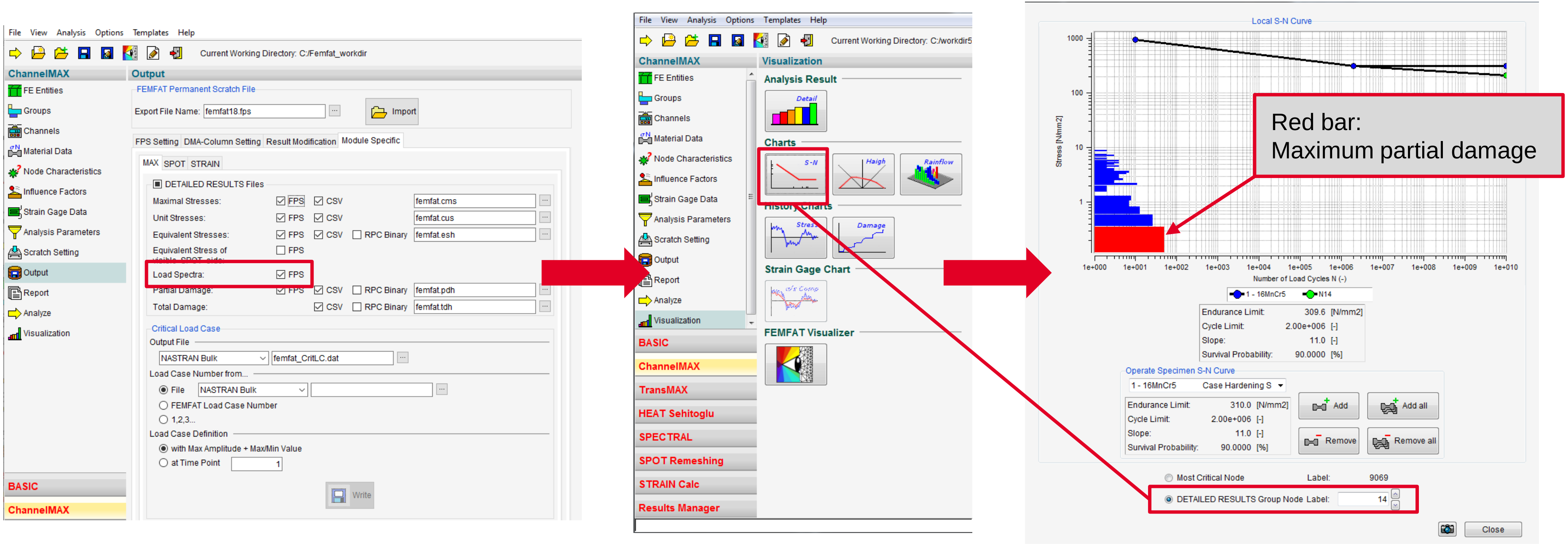
Using "-noh3d", the Solver Run Manager is forced NOT to start the HV-Trans script after the run is finished.

The H3D file is generated by RADIOSS itself anyway.

Starting RADIOSS for getting H3D using native export:

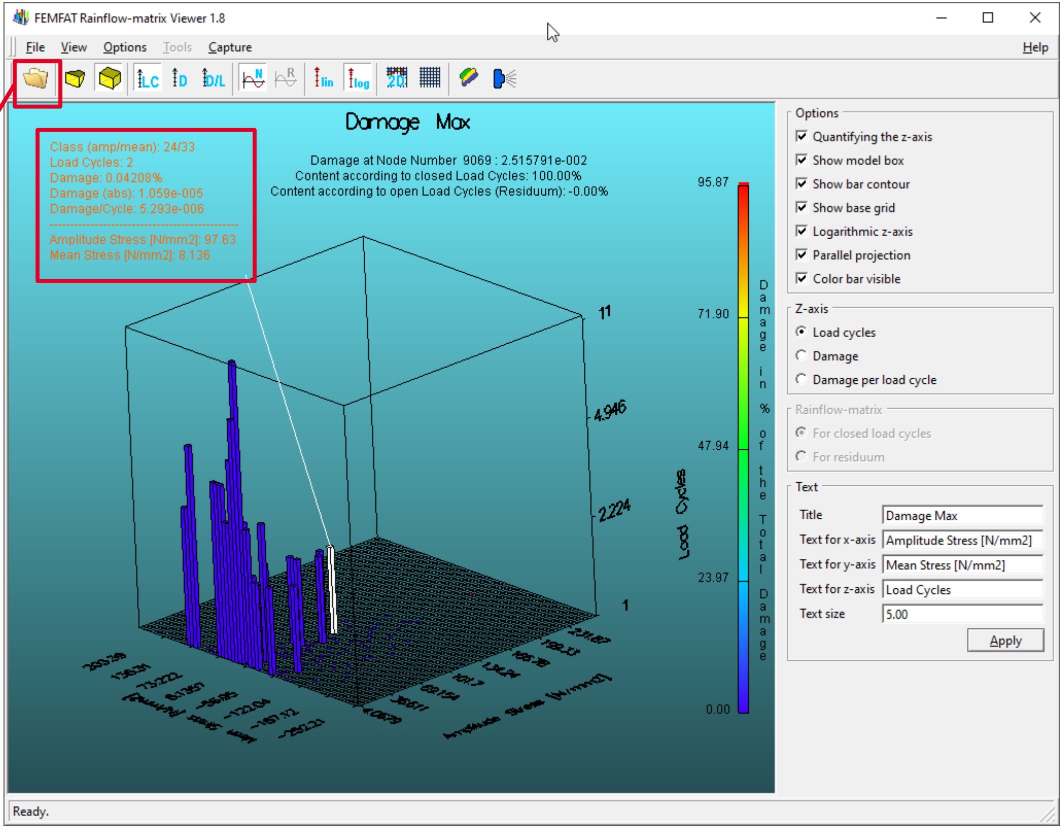


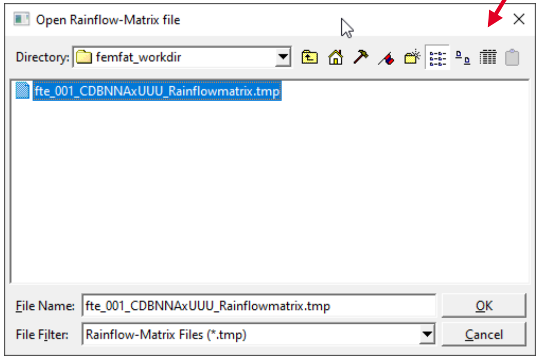
Display of Load Spectra for “Detailed Results” Group



New Functions in Rainflow-matrix Viewer

- The rainflow.exe can also be executed standalone.
- Via file dialog a tmp-file can be opened.
- Click on a bar to display a subwindow

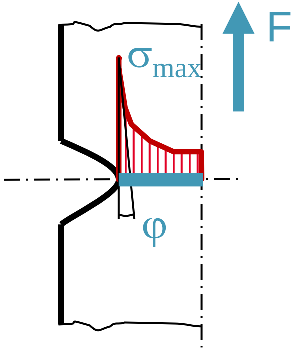




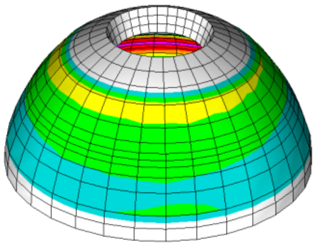
New Features of FEMFAT 5.4.2 released February 2021

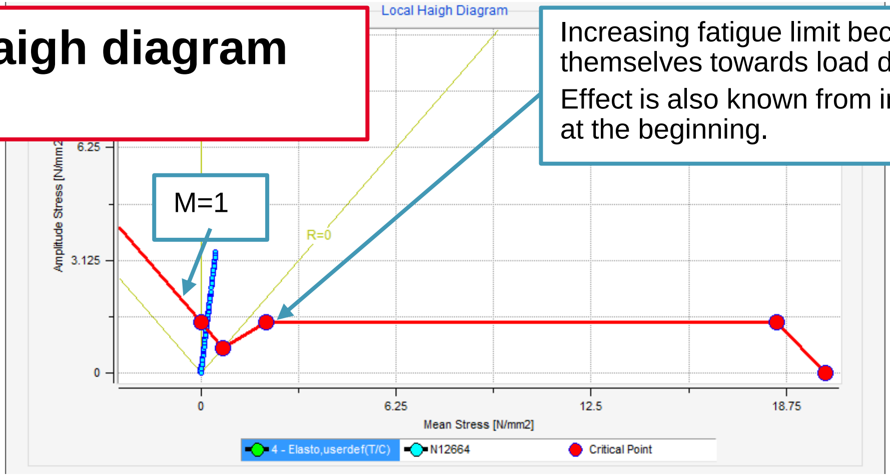
- **New stress gradient analysis method**
- Results Manager: multiple fps-file selection
- Results Manager: Enhanced ‘Formula’ combination method
- SPECTRAL: Output of equivalent stress PSDs as ASCII CSV file
- Dynamic memory allocation for load histories
- Import of averaged-at-nodes stresses (and strains for HEAT) from ABAQUS odb
- New material class and material data for elastomers (natural rubber) for shore hardness 40 to 60.

$$\chi' = \frac{1}{\sigma_{max}} \left(\frac{d\sigma}{dx} \right)$$



User defined Haigh diagram for elastomers



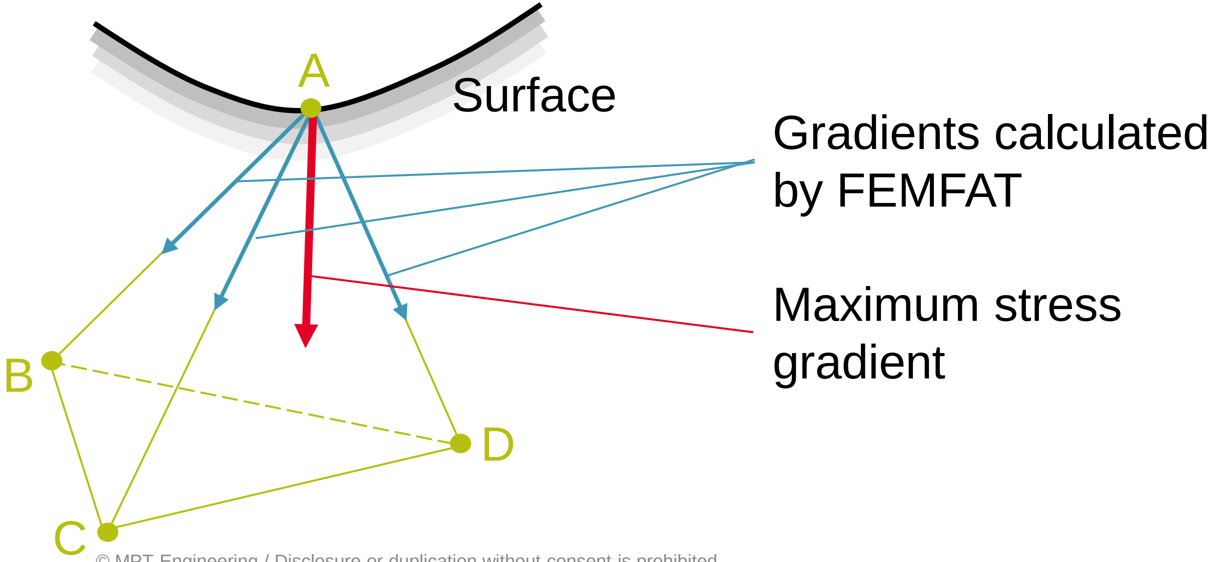


Increasing fatigue limit because of strain crystallization: Chains orient themselves towards load direction and block cracks.

Effect is also known from inflating balloons, which needs more power at the beginning.

New Stress Gradient Analysis Method

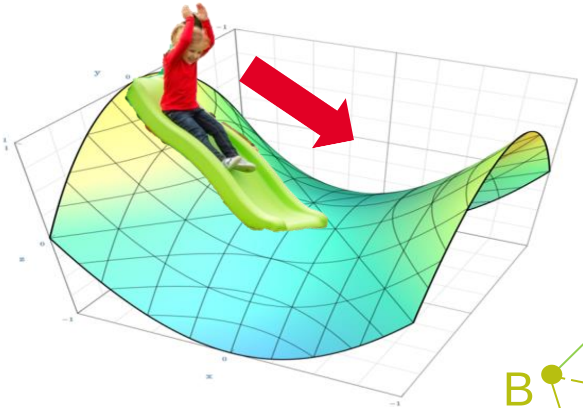
- **What was the problem?**
  - A complex shaped structure is usually meshed automatically with quadratic tetrahedrons
  - FEMFAT calculates the stress gradient along Finite Element edges
  - The maximum stress gradient is usually perpendicular to the surface
  - But often there are no Finite Element edges perpendicular to the surface
    → inaccurate results!

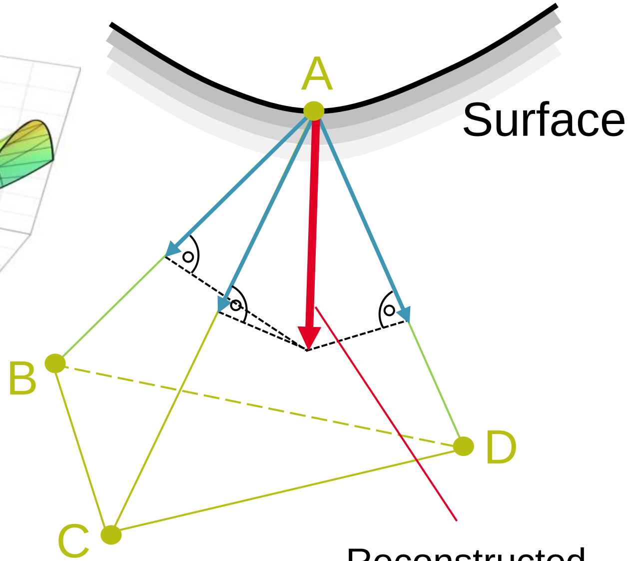


New Stress Gradient Analysis Method

- **Our solution**
  - A stress gradient is a vector pointing in the direction of the largest stress decrease, which can be reconstructed by its components along the Finite Element edges:

$$\vec{\chi} = -\vec{\nabla}\sigma_{Mises} = -\begin{pmatrix} \frac{\partial \sigma_{Mises}}{\partial x} \\ \frac{\partial \sigma_{Mises}}{\partial y} \\ \frac{\partial \sigma_{Mises}}{\partial z} \end{pmatrix}$$

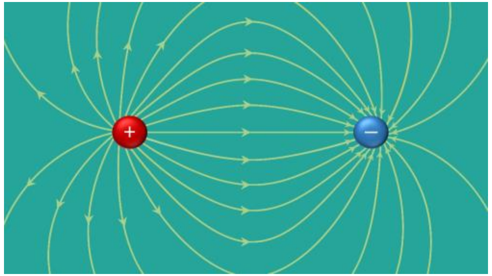




Reconstructed maximum stress gradient

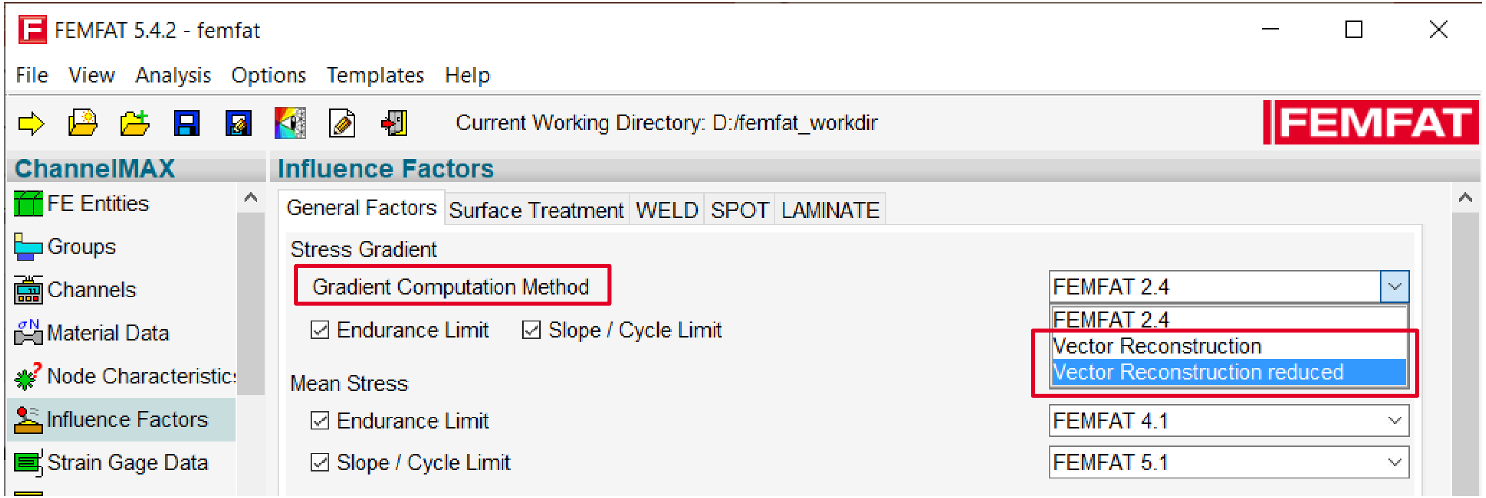
  - Analogon: Static electric field:

$$\vec{E} = -\vec{\nabla}\varphi$$



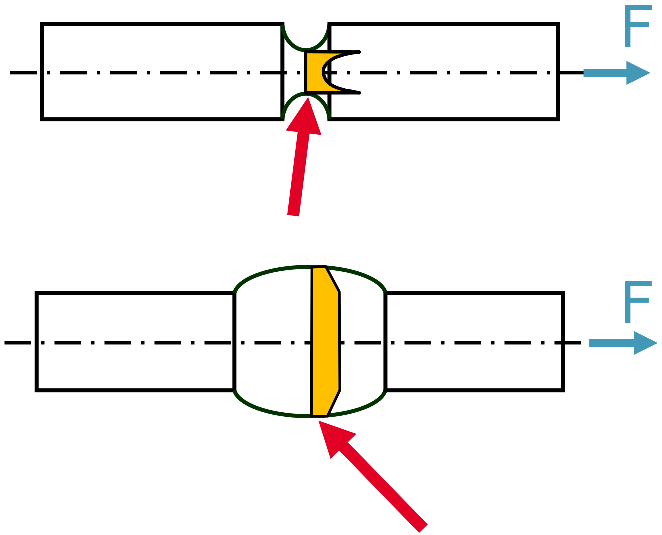
New Stress Gradient Analysis Method

- Additional improvements
  - Usage of superimposed stresses in ChannelMAX
    → Ensures the invariance of the stress gradient with respect to the analysis type (static mounting or inertia relief)
  - Equivalent method to TransMAX. Same results are obtained for same loading.
  - ***Vector Reconstruction reduced***: Accelerated analysis with reduced time steps for superposition, but mostly same accuracy



New Stress Gradient Analysis Method

- Further improvements
  - Only Finite Element edges with decreasing stresses are considered for gradient reconstruction, because there is no support effect for bulges („negative“ notches).



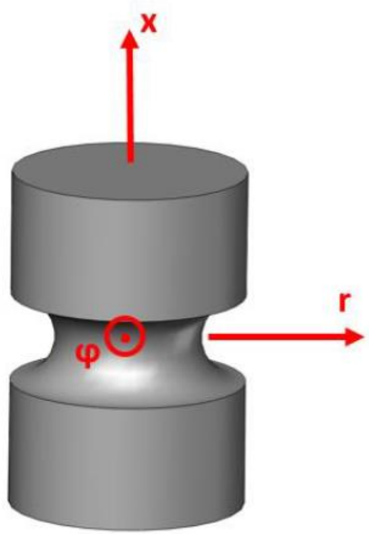
Decreasing stress → gradient influence

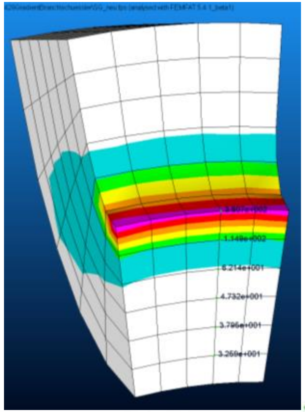
Increasing stress → NO gradient influence

New Stress Gradient Analysis Method

- Stress gradient progress at notch root into the depth

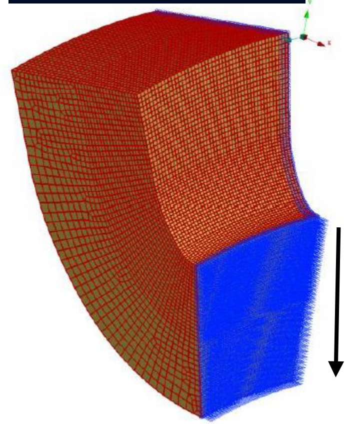
Specimen





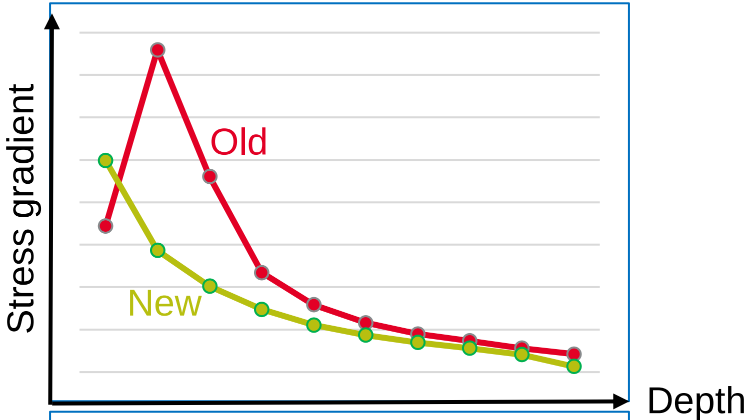
Rough mesh

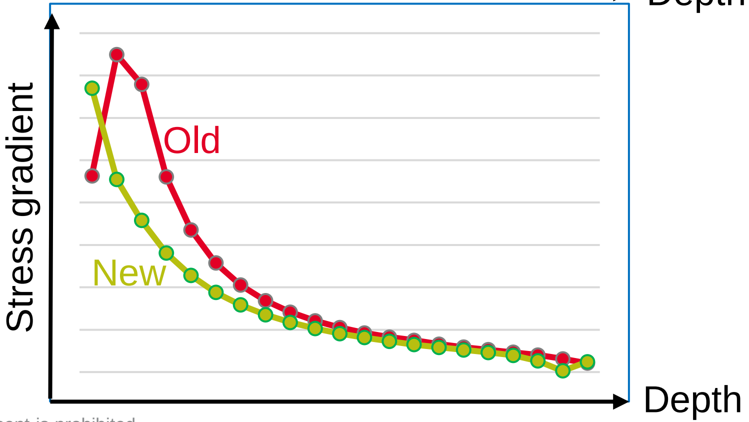
Depth



Fine mesh

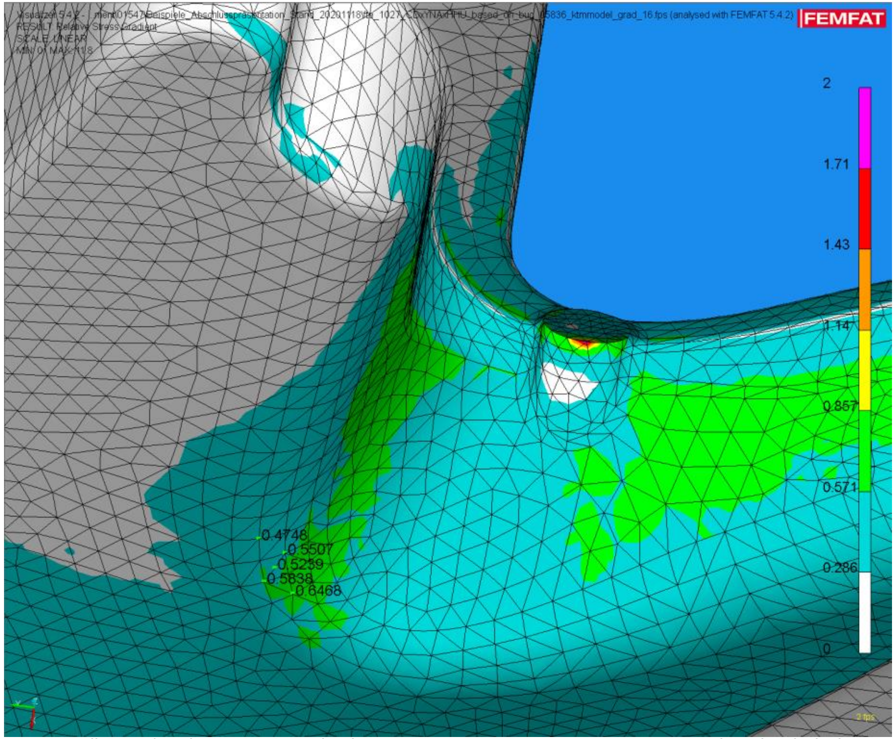
Depth



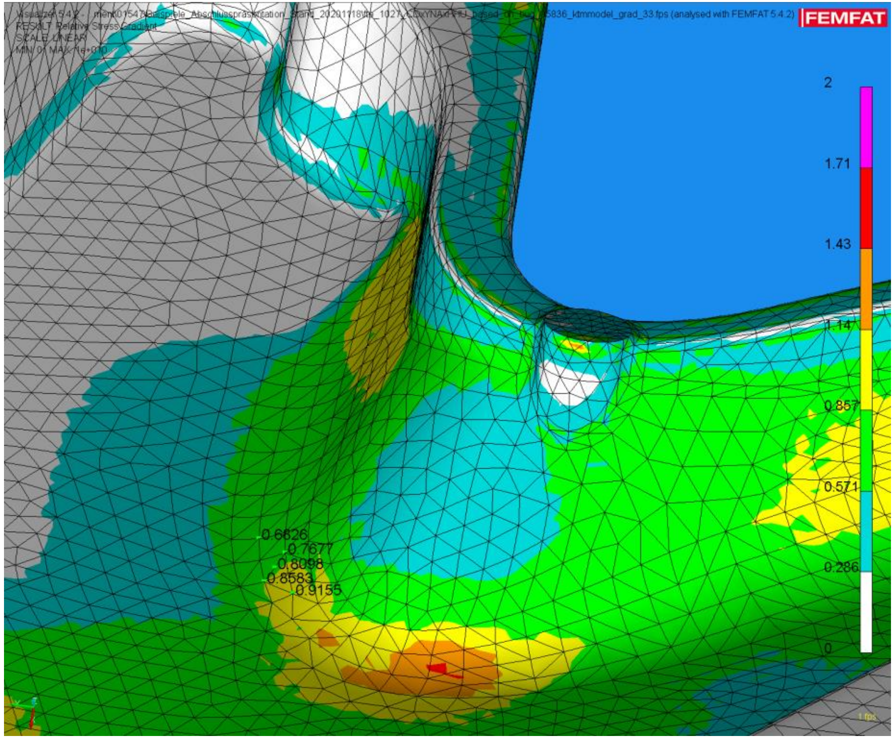


New Stress Gradient Analysis Method, Example

- Stress gradient result, old method

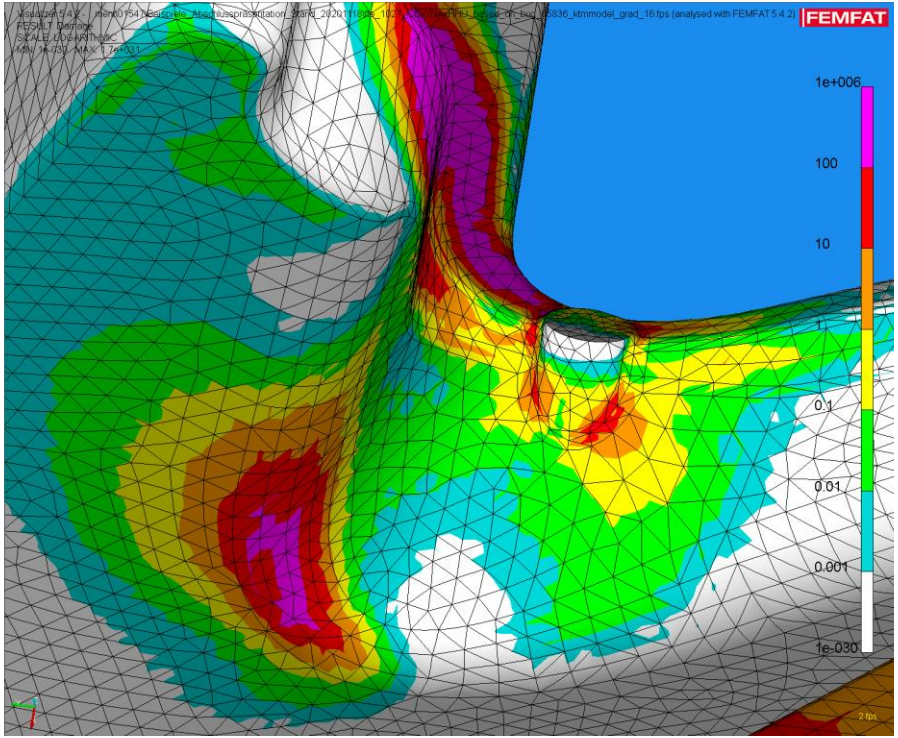


- Vector reconstruction

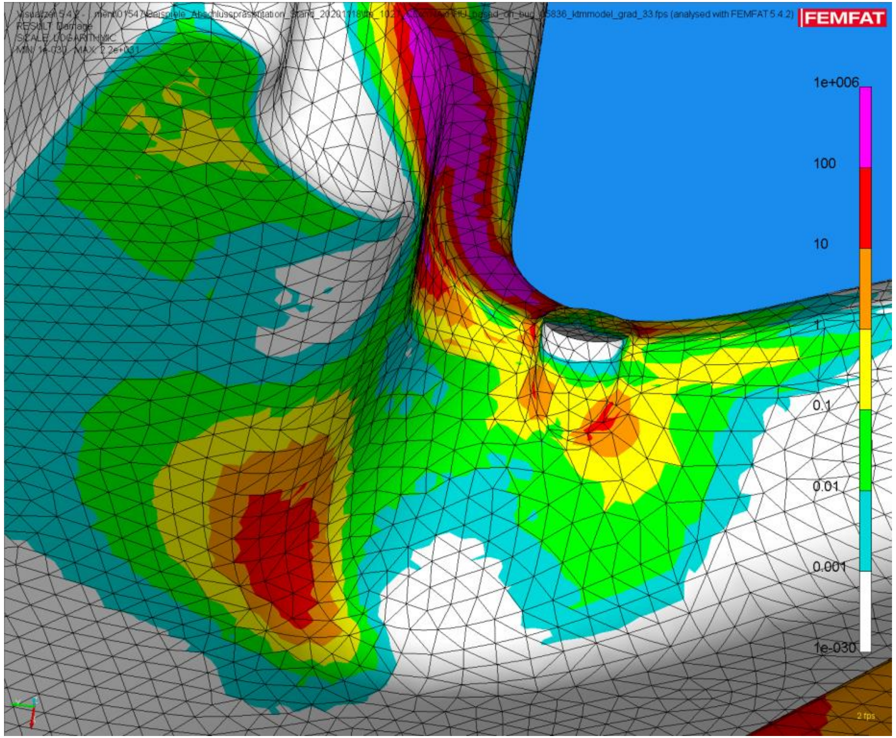


New Stress Gradient Analysis Method, Example

- Damage result, old method

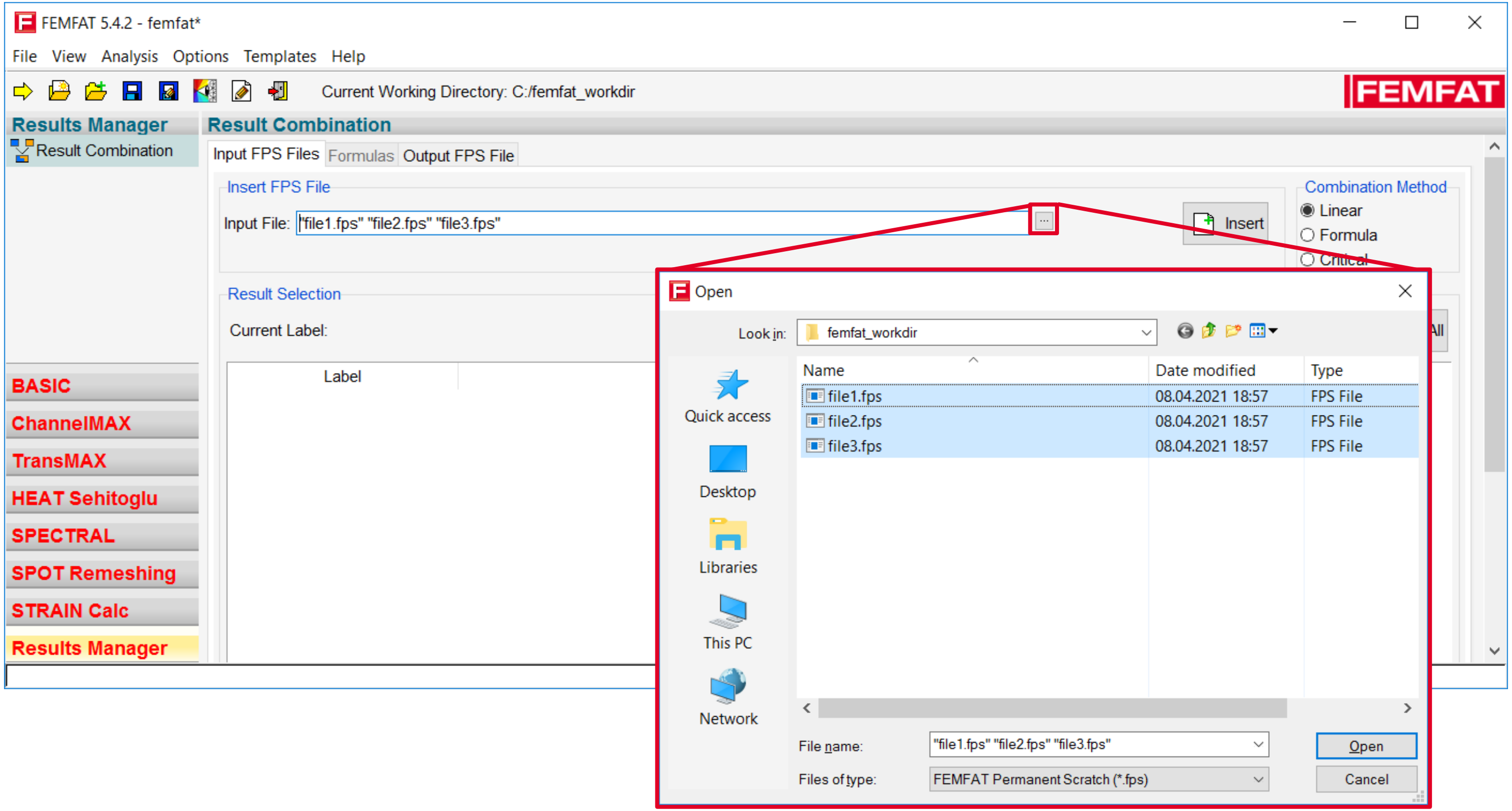


- Damage distribution, vector reconstruction



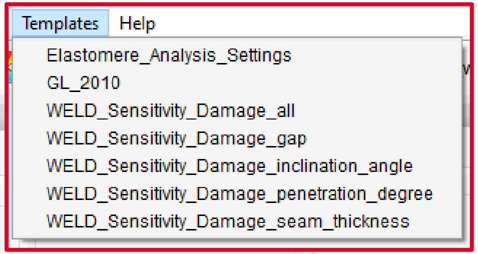
Results Manager: Select Multiple fps-Files

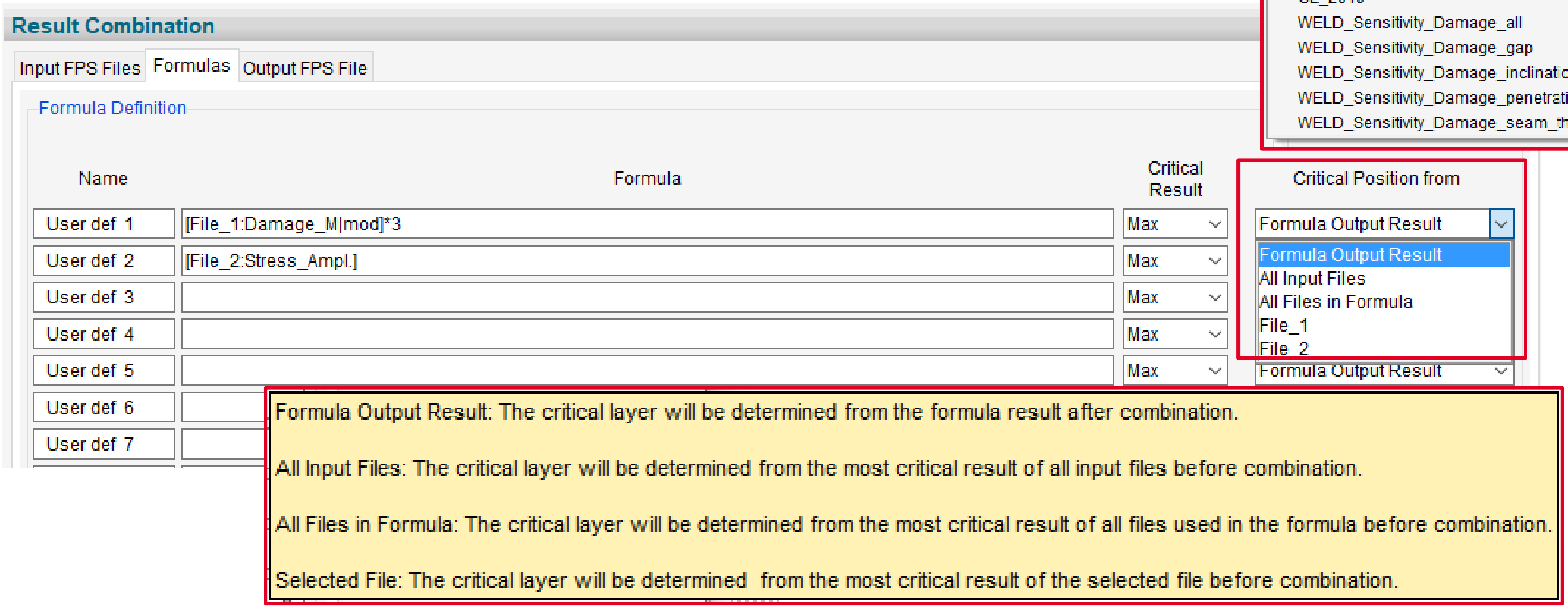
- Select more than one fps-file in the selection dialog



Results Manager: Enhanced 'Formula' Combination Method

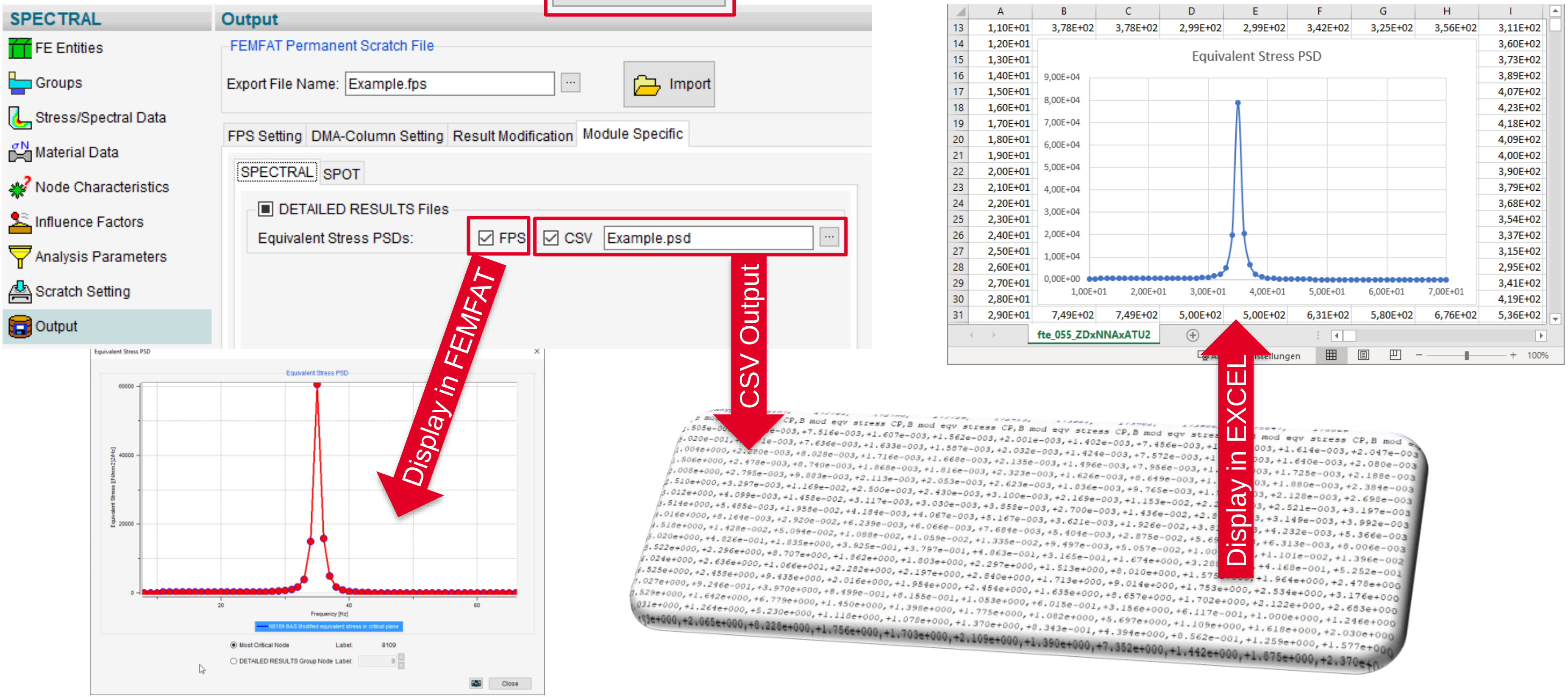
- The user can control the determination of the critical assessment point (e.g. at welding seams: weld toe, weld root, top or bottom, etc.).
- Useful functionality for WELD sensitivity analysis.





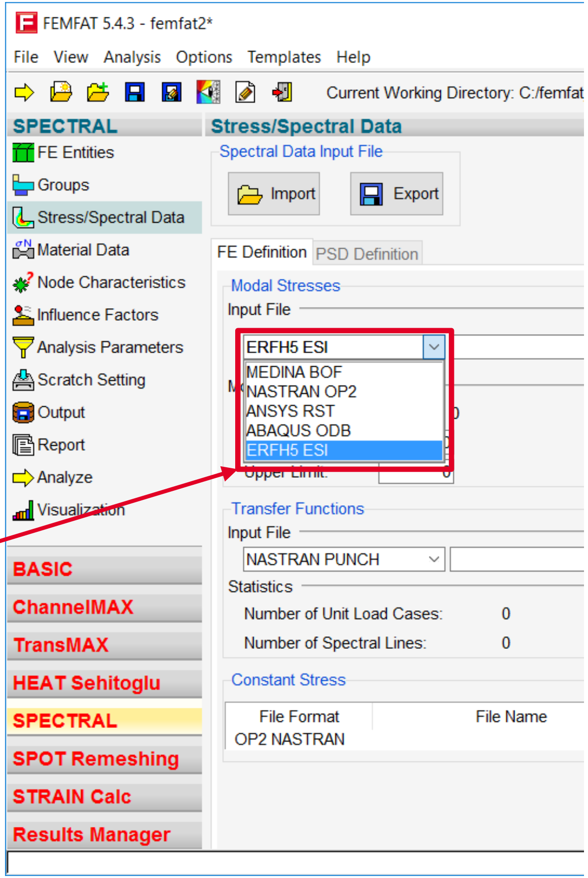
SPECTRAL: Output of Equivalent Stress PSDs as ASCII CSV File (*.psd)

- Output for all nodes in the DETAILED RESULTS group



Outlook FEMFAT 2021 (planned to be released in Q3 2021)

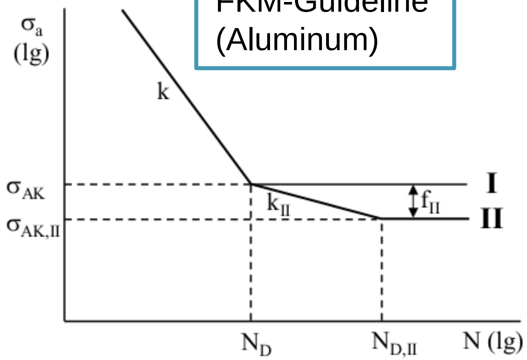
- Highlights
  - New multi-segment S-N curve model
  - Automatic material assignment
  - Automatic channel generation and assignment of load time data to channels
  - VISUALIZER: New graphic kernel and enhanced post processing possibilities
  - SPOT: New advanced self piercing rivet model
- Methods
  - WELD Automatic Stress Correction: Evaluation distance 0 for root assessment
  - Combination of influence factors according to the German FKM-Guideline
  - BREAK: Static safety factor analysis available in STRAIN calc
- Textual and graphical output
  - Display exact point coordinates in the S-N diagram and Haigh diagram
  - Critical load combination - separate output of times for maximum and minimum stress
  - WELD: Pro-file output of both ROOT and TOE results as table
  - Simultaneous output of SPOT detailed results and base material results to dma-file
  - SPOT: Output of the history of section forces as CSV file
  - Output of cycle limit in Haig-diagram
- Interfaces
  - ESI HDF5 stress interface for SPECTRAL
  - Accelerated ABAQUS interface (import of elemental/nodal forces only if needed)
- Miscellaneous
  - ALTAIR licensing for FEMFAT inside ANSYS



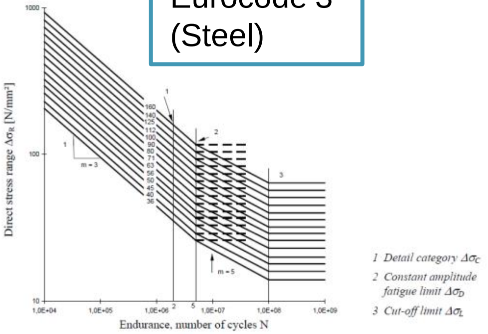
New Multi-Segment S-N Curve Model

- S-N curves from different standards:

FKM-Guideline (Aluminum)



Eurocode 3 (Steel)



Eurocode 9 (Aluminum)

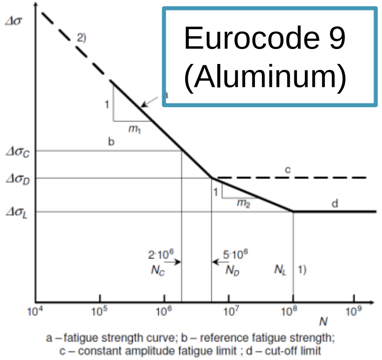
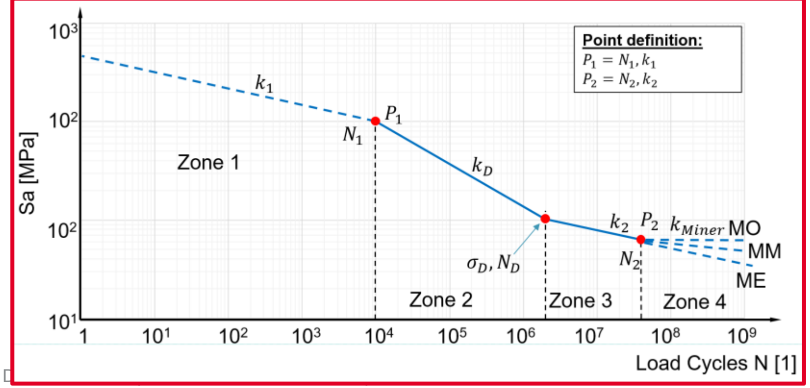


Figure 6.1 —Fatigue strength curve logΔσ-logN

- Support of S-N curve model with 4 segments in FEMFAT:
  - Zone 1: Low Cycle Fatigue
  - Zone 2: High Cycle Fatigue
  - Zone 3: Very High Cycle Fatigue
  - Zone 4: Miner Modification



New Multi-Segment S-N Curve Model

Material Data

- Type Dependent S-N Data
  - Type-Code of S-N Curve: Multi-Segment Model
  - Low Cycle Fatigue
    - Slope of S-N Curve: 20.00 (optional, will be autocompleted)
    - Cycle Limit: 10000 (optional, will be autocompleted)
    - Slope of S-N Curve for Shear Loading: 0.00 (optional, will be autocompleted)
    - Cycle Limit for Shear Loading: 0 (optional, will be autocompleted)
  - High Cycle Fatigue
    - Slope of S-N Curve: 11.00
    - Cycle Limit of Endurance: 2000000
    - Stress Limit of Endurance: 310.0 [N/mm2]
    - Cycles at Failure: 0 (not used for analysis)
    - Ultimate Strength: 0.0 [N/mm2] (not used for analysis)
    - Survival Probability: 90.0000 [%]
    - Slope of S-N Curve for Shear Loading: 0.00
    - Cycle Limit of Endurance for Shear Loading: 0
  - Very High Cycle Fatigue
    - Slope of S-N Curve: 30.00
    - Cycle Limit: 100000000
    - Slope of S-N Curve for Shear Loading: 0.00
    - Cycle Limit for Shear Loading: 0
  - MINER Modified
    - Parameter a: 2.00
    - Parameter b: -1.00

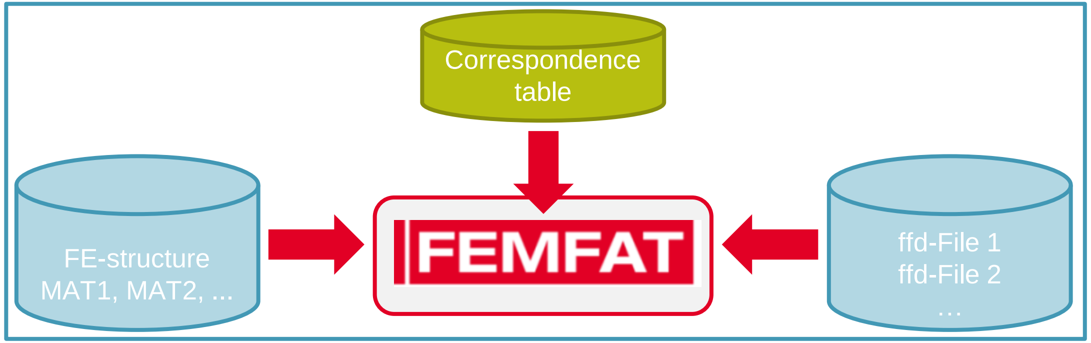
Influence Factors

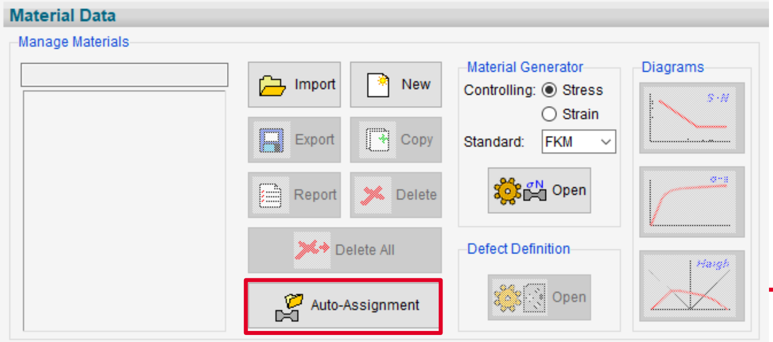
Modification Method for Multi-Segment S-N Curve

- [x] Modify Slope k1 (Low Cycle Fatigue)
- [ ] Modify Slope k2 (Very High Cycle Fatigue)

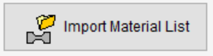
Visualization

Automatic Material Assignment





- Import material correspondence table *.fma
- Loading all ffd-Files automatically
- Automatic assignment of materials to nodes at elements with MID/PID according to the correspondence table
- Supported FE-interfaces: NASTRAN BDF & OP2, ANSYS CDB, MEDINA BIF, I-DEAS MS UNV
- Other interfaces: material import without assignment



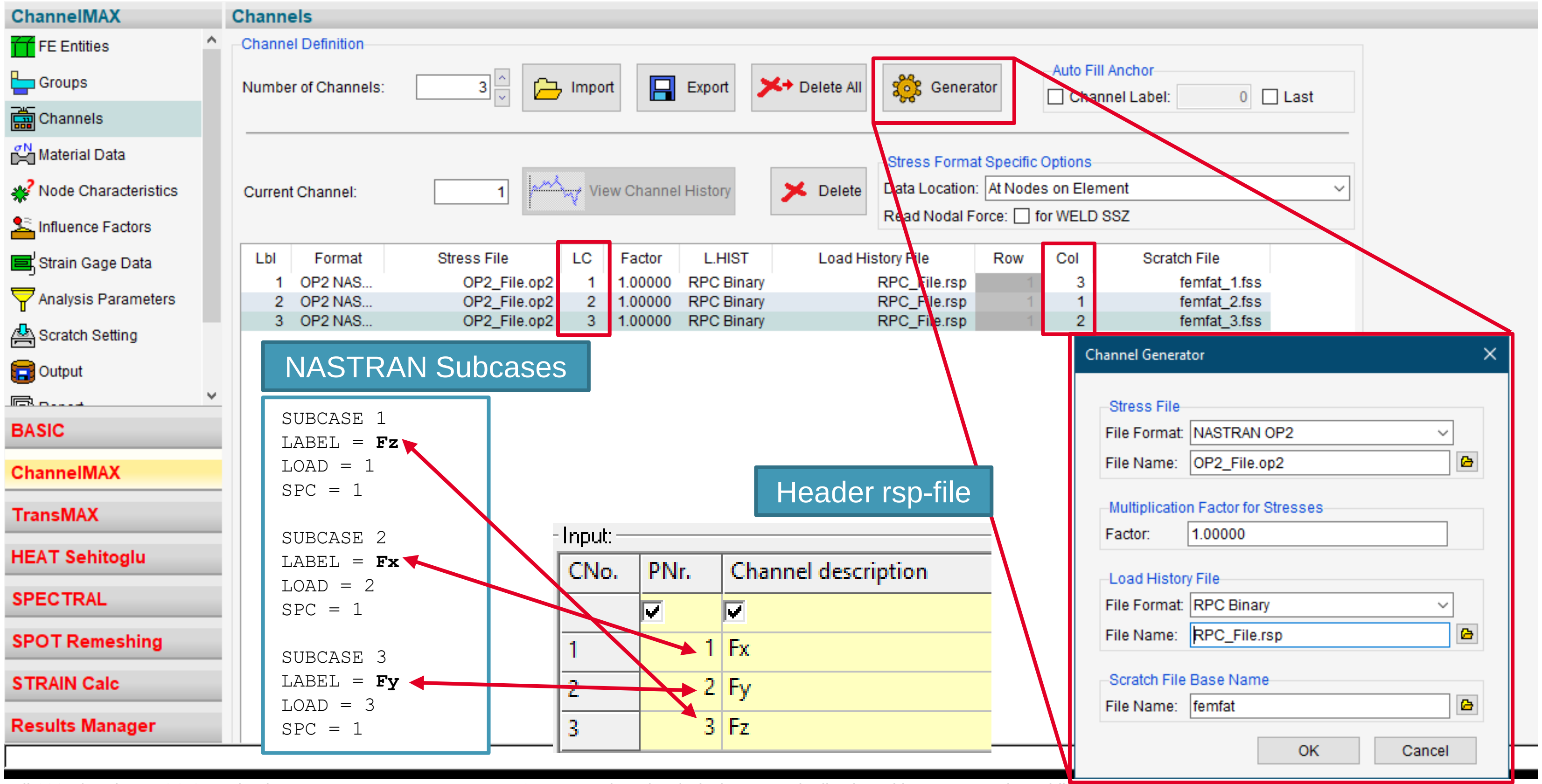
Material correspondence table *.fma-file

```
# Example

Material Database Path ($MatDBPath): C:\MaterialData
#      MID      |       PID      |        FEMFAT material file (.ffd)        |    Material description      | Comment
# ---------------------------------------------------------------------------------------------------------------
       1        |                |  EN-GJL-250_HEAT_FKM2002.ffd               | Grey Cast Iron               |
                |        2       |  EN-AC-21100_sand_casting_FKM2002.ffd      | Aluminum                     |
       2,3;4    |                |  EN-E295_FKM2002.ffd                       |                              |
       5-6      |        6       |  EN-MC-MgAl4RE2_casting_FKM2000.ffd        |                              |
```

Automatic Channel Generation and Assignment of Load Time Data to Channels

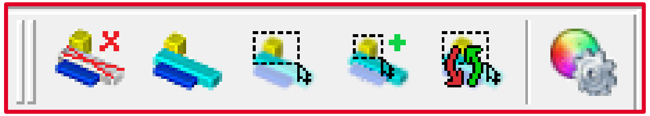
- NASTRAN Subcase label is used for automatic assignment of RPC load time history.

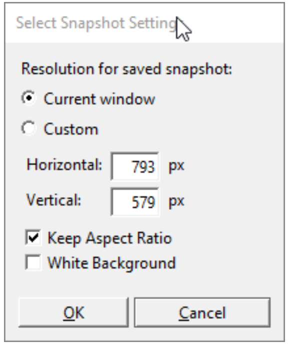


VISUALIZER: New Graphic Kernel and Enhanced Post Processing Possibilities

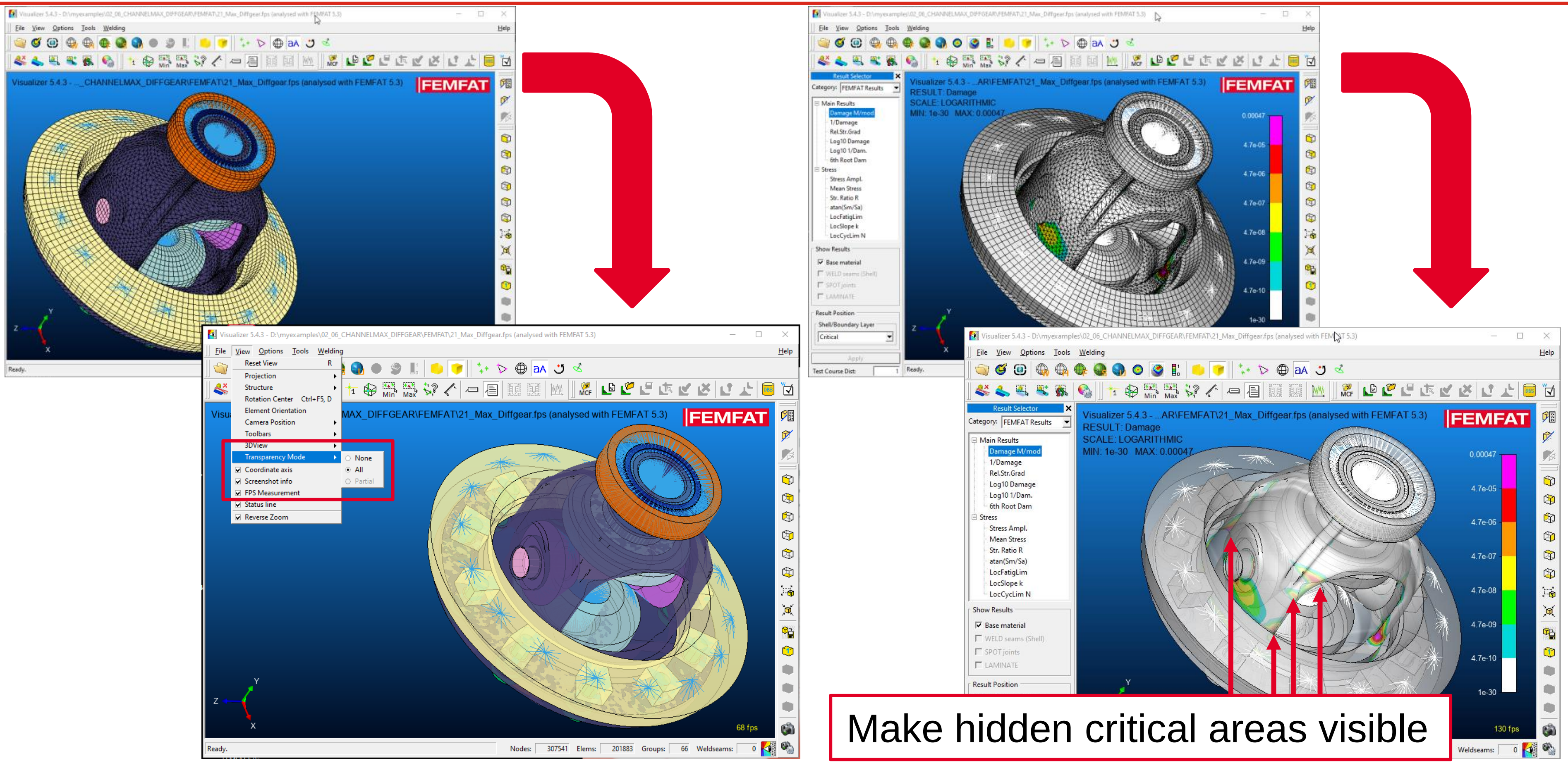
- New graphic kernel from ceetron
  - Increased performance.
  - Long term support of new graphic cards by usage of newer OpenGL versions.
  - Prework for future enhanced functionalities as e.g. result display on cutting planes.

- Enhancements
  - Custom resolution and advanced options for screenshots, e.g. 4K images
  - Transparent mode for hidden parts
  - Area select for elements
  - Add neighboring elements
  - Inversion of visible elements
  - Edge detection and feature lines

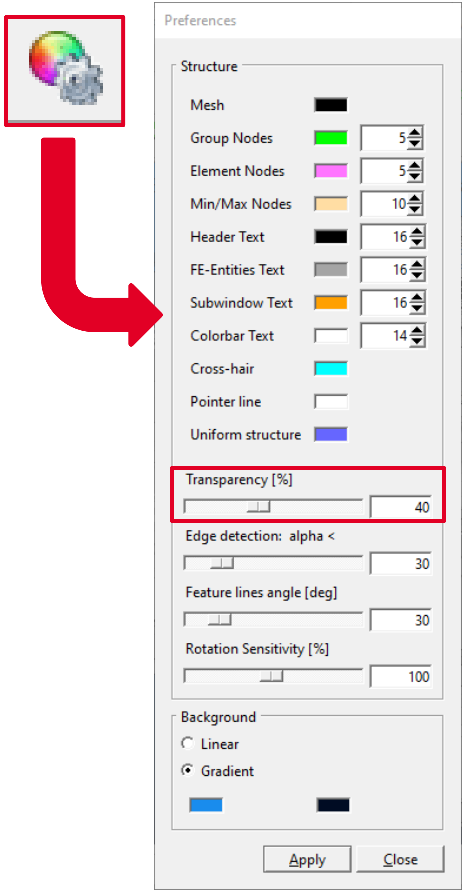


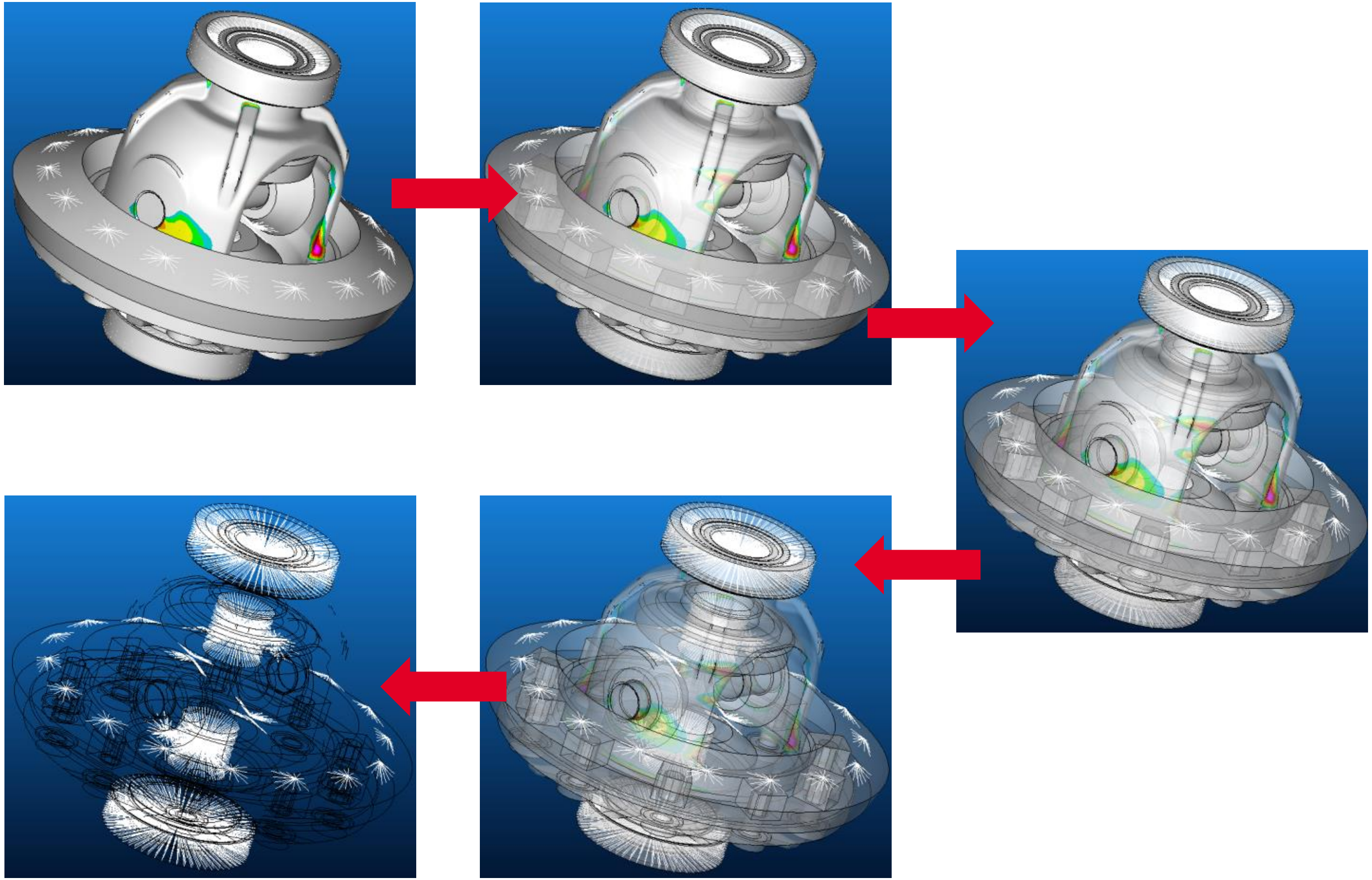


VISUALIZER: Transparent Mode for Hidden Parts

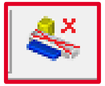


VISUALIZER: Transparent Mode for Hidden Parts

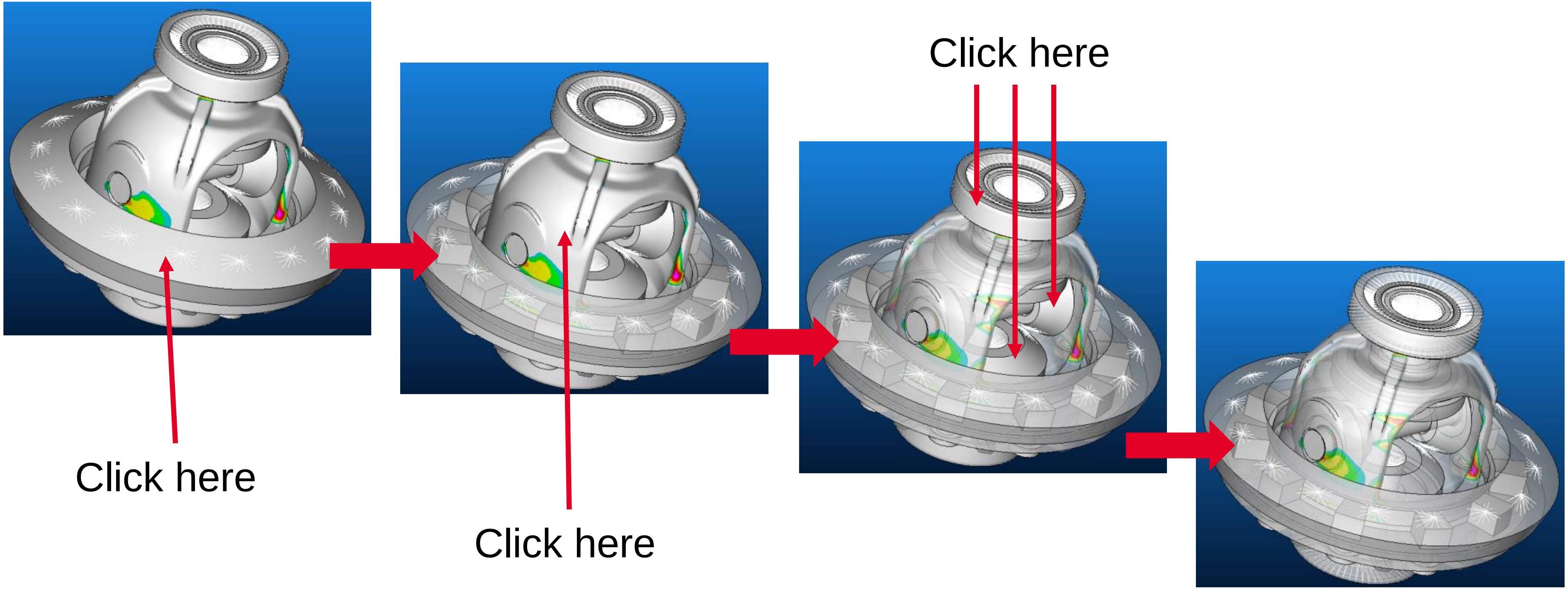




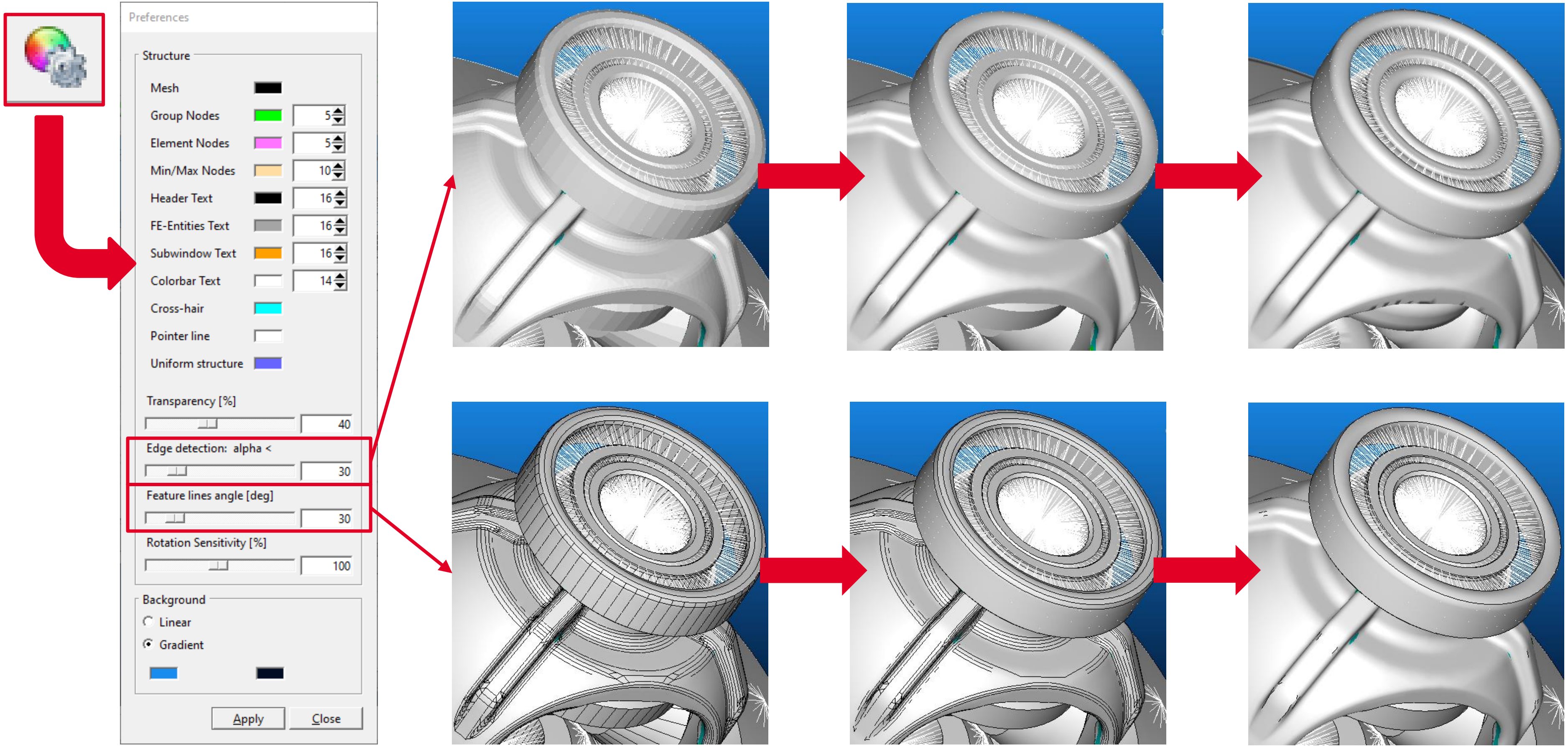
VISUALIZER: Transparent Mode for Hidden Parts



+ Shift: Make elements of a PID transparent

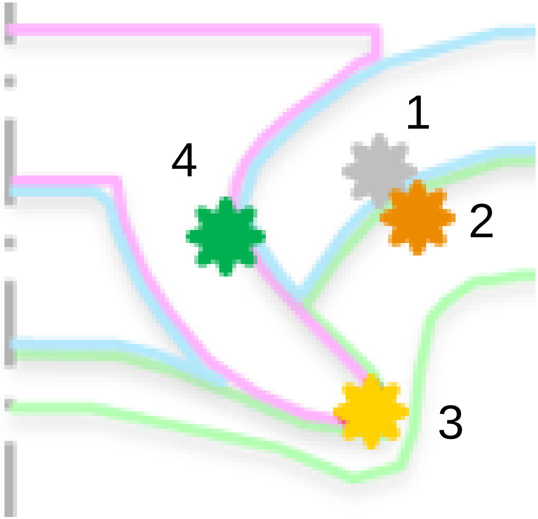


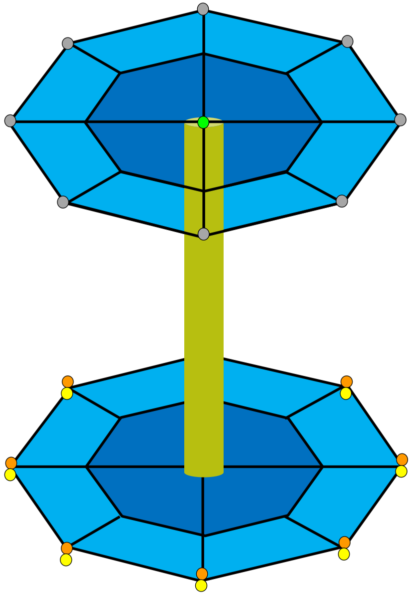
VISUALIZER: Edge Detection and Feature Lines



SPOT: New advanced Self Piercing Rivet Model

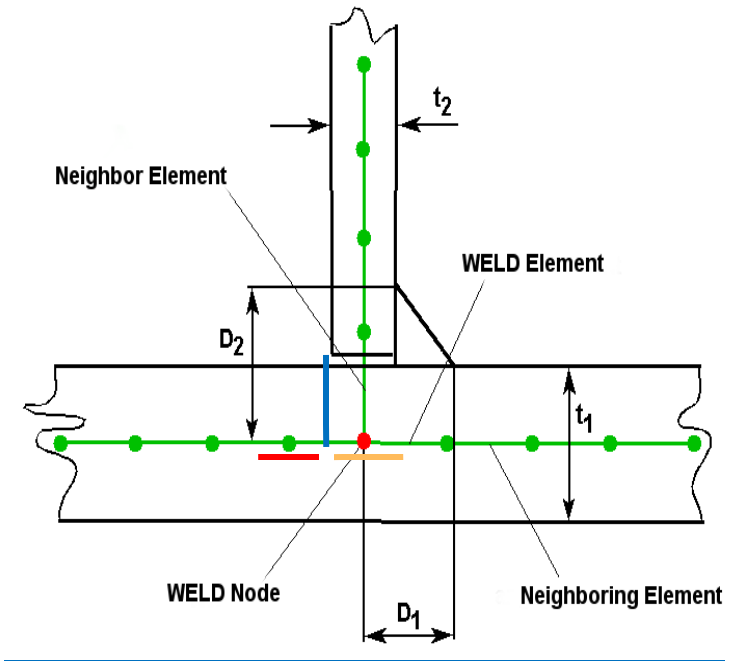
- Assessment of sheet failure and rivet failure in one analysis run.
- Assessment of stresses from nugget model and forces/moments from connecting beam element
- Stress based assessment of stamped sheet (failure position 1)
- Stress based assessment of die sheet (failure positions 2 and 3)
- Force based assessment of rivet (failure position 4)
- New <rivet_advanced> entry in SPOT database

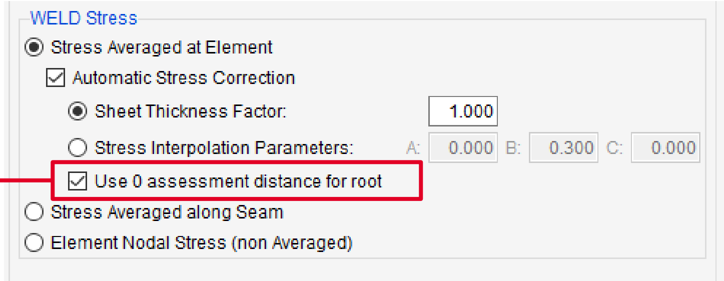




WELD: Reduced assessment distance for root

- Assessment distance for automatic stress correction can be chosen different for root and toe:





Toe: $D = f_{ele}\,(0{,}5\ t_{neighbor} + t_{min})$

Root: $D = 0$

Combination of Influence Factors According to the German FKM-Guideline

Extract from the FKM-Guideline

The design factors of rod-shaped (1D) non-welded components for normal stress and for shear stress are [*1]

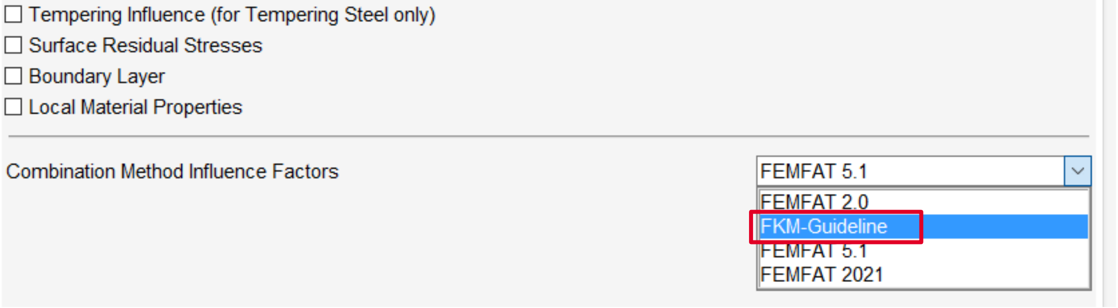
$$K_{WK,\sigma} = \frac{1}{n_\sigma}\cdot\left(1+\frac{1}{\tilde{K}_f}\cdot\left(\frac{1}{K_{R,\sigma}}-1\right)\right)\cdot\frac{1}{K_V\cdot K_S\cdot K_{NL,E}}, \qquad (4.3.1)$$

$$K_{WK,\tau} = \frac{1}{n_\tau}\cdot\left(1+\frac{1}{\tilde{K}_f}\cdot\left(\frac{1}{K_{R,\tau}}-1\right)\right)\cdot\frac{1}{K_V\cdot K_S}.$$

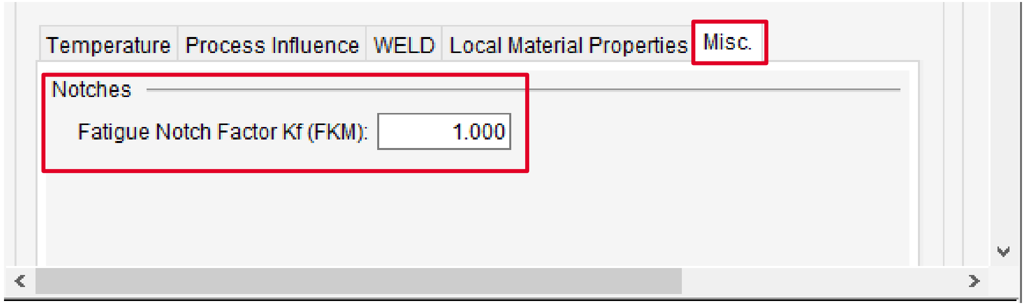
n_σ, ...	K_t-K_f ratio, Chapter 4.3.2,
$\tilde{K}_f$	constant, Table 4.3.1, if no better estimate is available,
$K_{R,\sigma}$, ...	roughness factor, Chapter 4.3.3,
K_V	surface treatment factor, Chapter 4.3.4,
K_S	coating factor, Chapter 4.3.4,
$K_{NL,E}$	constant for GG, Chapter 4.3.5.

Aim of the formulas: Reduced roughness sensitivity in notches!

Influence Factors

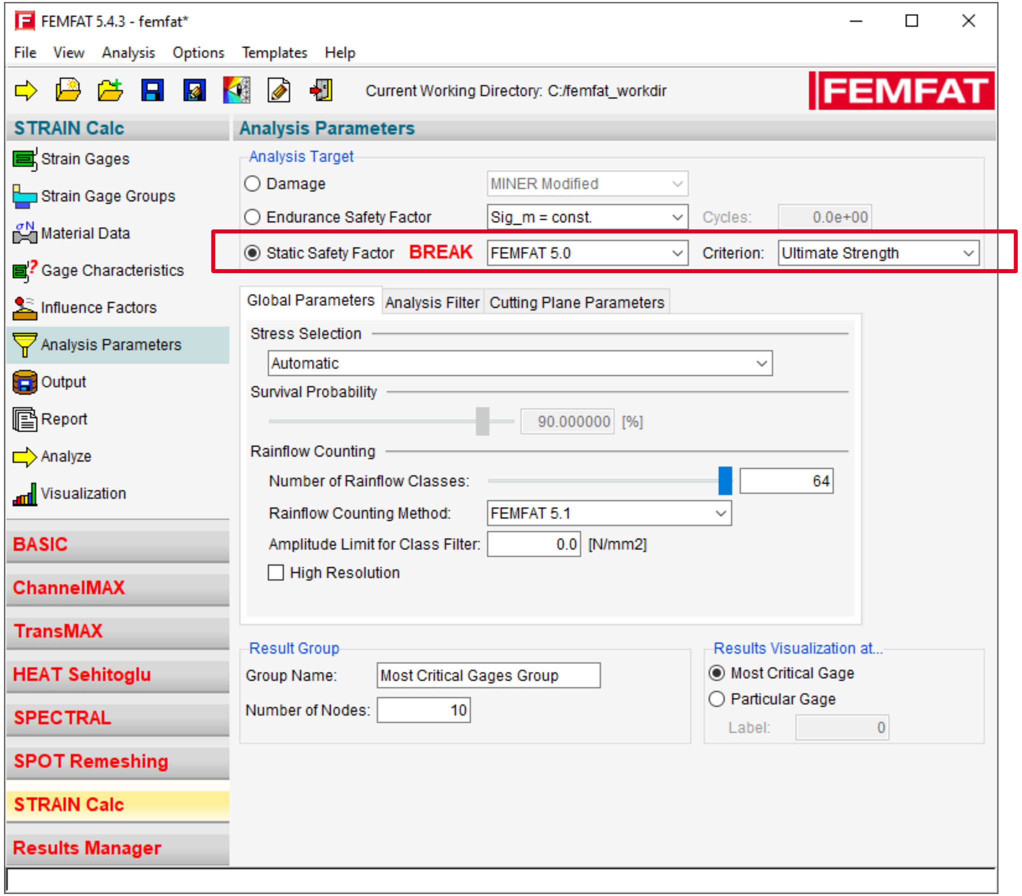


Node Characteristics



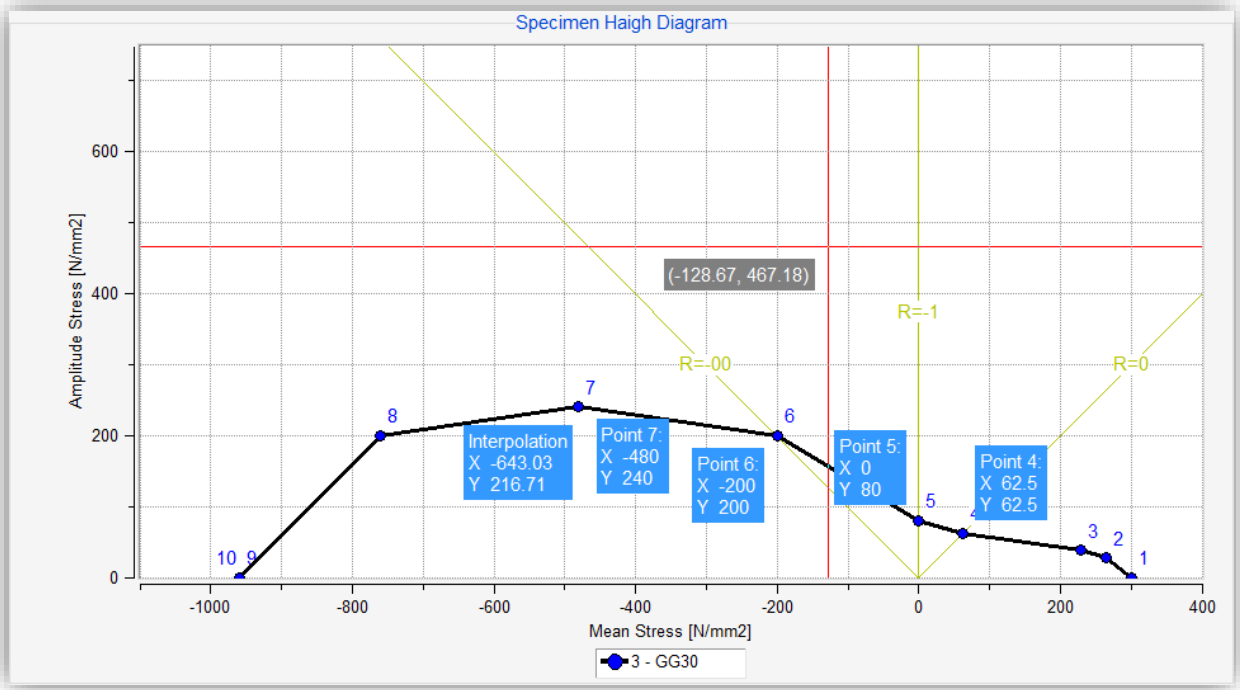
BREAK for STRAIN Calc

- Static safety factor analysis based on measured strains

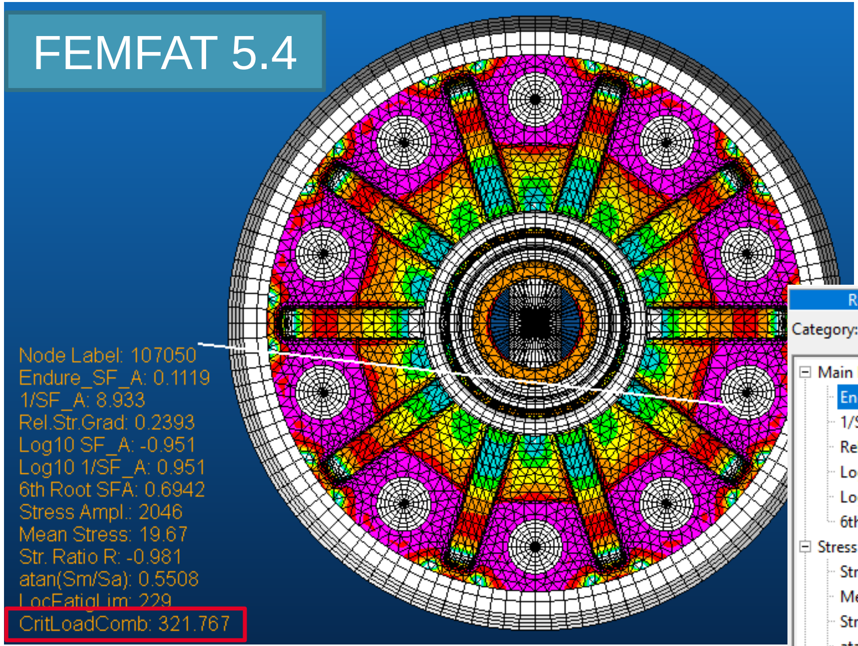


Display Exact Point Coordinates in the S-N and Haigh Diagram

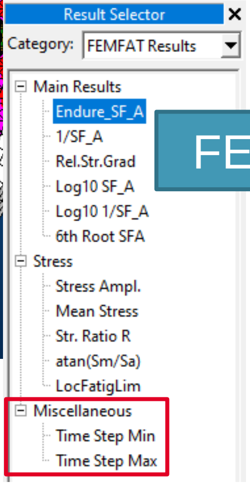
- Display Exact Point Coordinates in the S-N and Haigh Diagram by catching points
- Marker can be fixed
- Position of marker can be changed

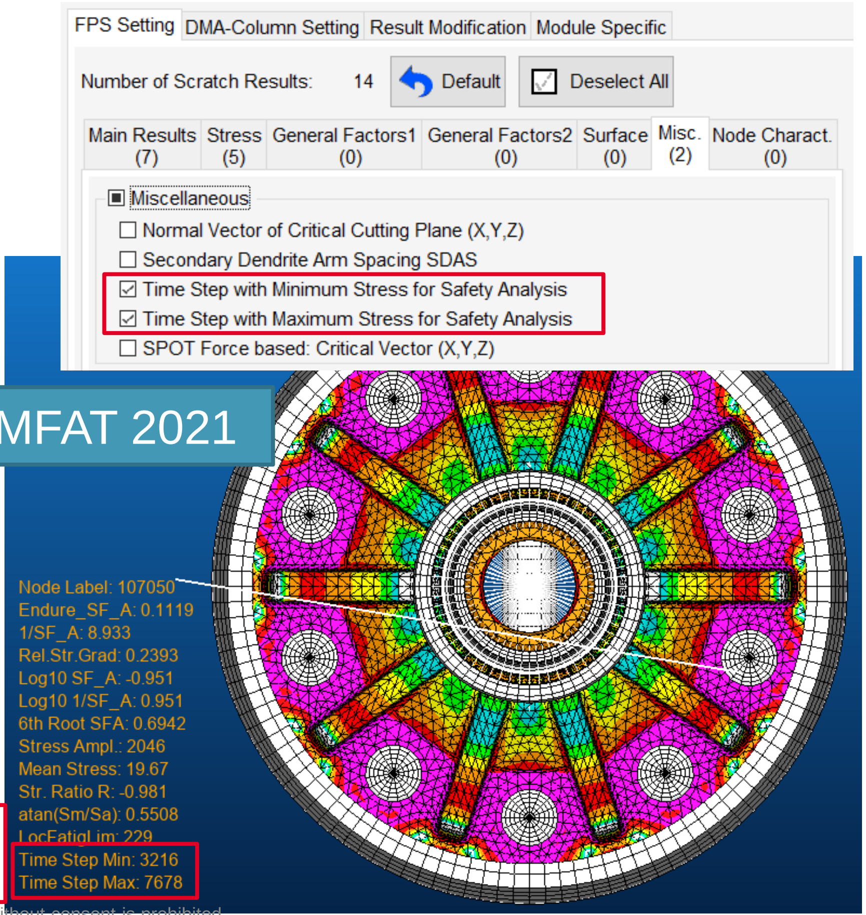


Critical Load Combination - Separate Output of Times for Maximum and Minimum Stress



Only the leading 3 digits are displayed

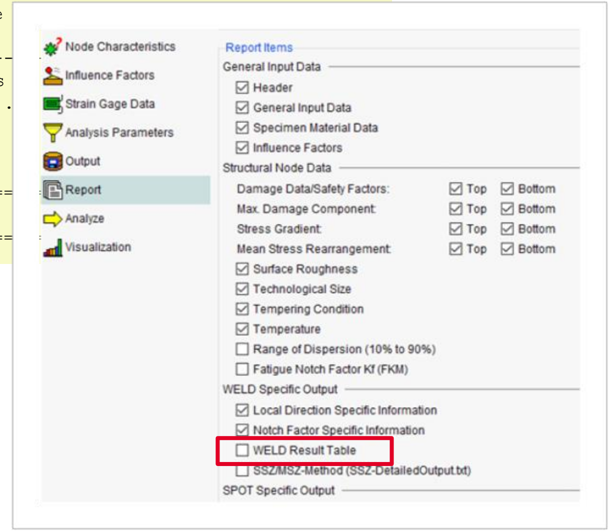




Separate output of time for max. and min. stress

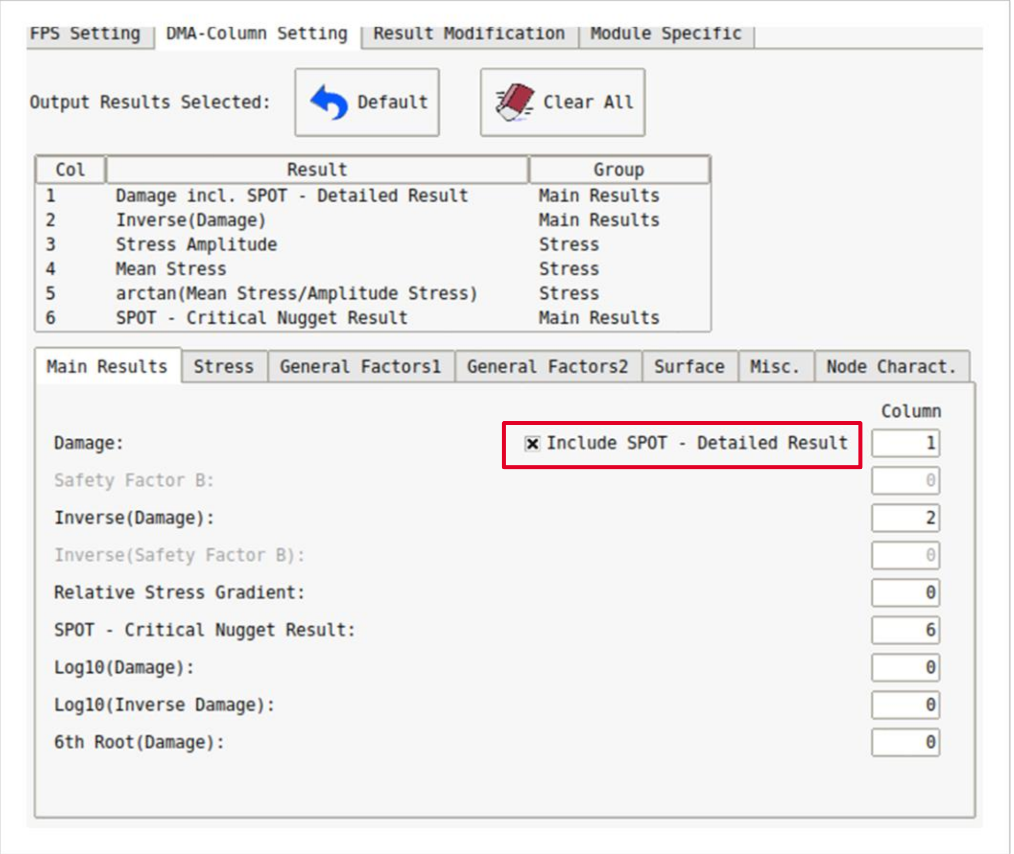
WELD: Pro-File Output of Both ROOT and TOE Results as Table

```
===========================================================================================================================================
 WELD Result Table:
===========================================================================================================================================
      Node       Damage   Critical Element    Type          Position         Notch Factor           Attribute
         5   3.170E-06                  14     222   Start /Toe /Bot                 1.24                C106
-------------------------------------------------------------------------------------------------------------------------------------------
                 Damage            Element    Type          Position   Notch Stress Ampl.   Notch Mean Stress   Fatigue Limit   Crit. Stress
...........................................................................................................................................
              5.428E-09                 13     220   Middle/Toe /Top               170.02              166.50          156.18      Sig_equiv
              7.140E-09                 13     220   Middle/Root/Top                30.83              173.17           37.00   Sig_parallel
              3.170E-06                 14     222   Start /Toe /Bot               229.52              233.10          177.77     Sig_normal
              4.710E-09                 14     222   End   /Root/Bot                81.98              180.70          192.15     Sig_normal
              8.296E-09                 15     224   Middle/Toe /Bot               104.90               29.51           62.08    Tau (Shear)
              1.107E-09                 15     224   Middle/Root/Bot                88.40              167.96           32.30    Tau (Shear)
===========================================================================================================================================
      Node       Damage   Critical Element    Type          Position         Notch Factor           Attribute
         …              …
--------------------------------------------------------------------------------------------------------------------
                 Damage            Element    Type          Position   Notch Stress Ampl.   Notch Mean Stress
....................................................................................................................
                                …
                                …
                                …
====================================================================================================================
 End - WELD Result Table
====================================================================================================================
```



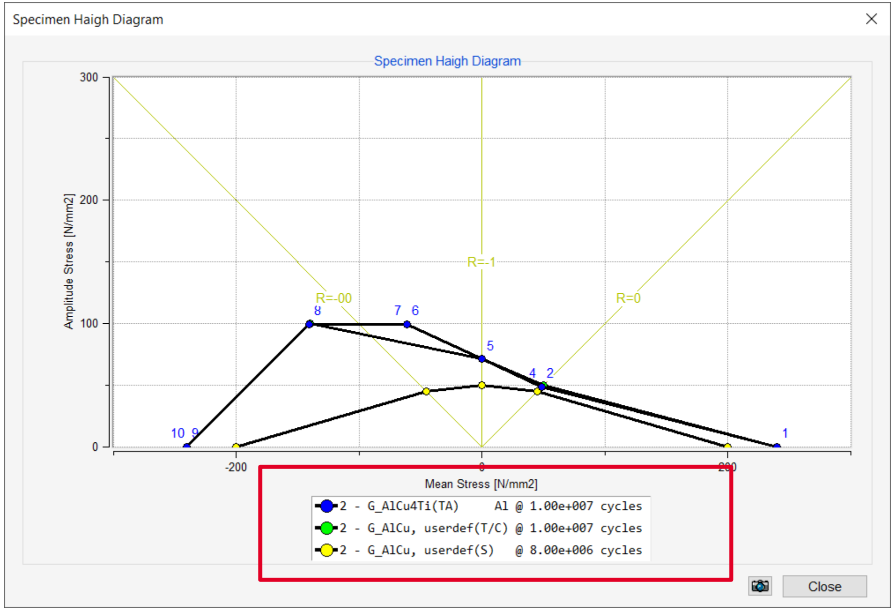
SPOT: Simultaneous Output of SPOT Detailed Results and Base Material Results to dma-File

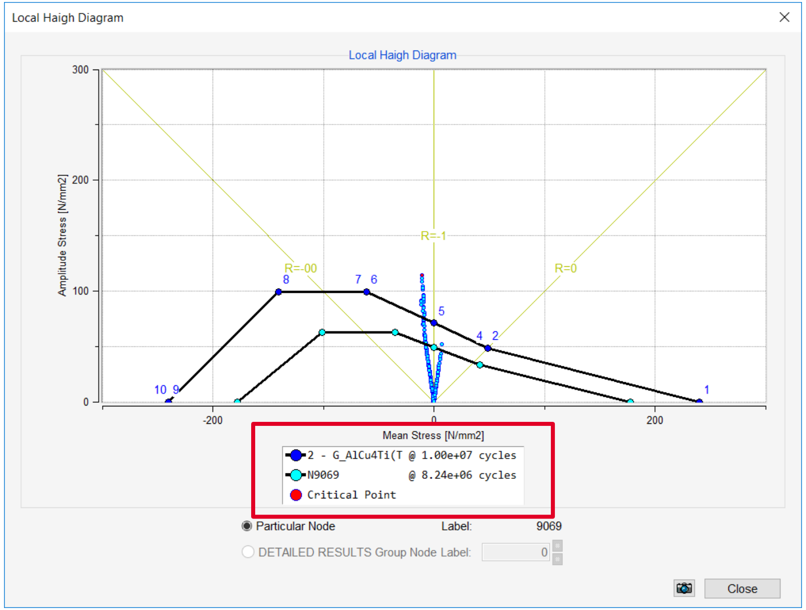
- In FEMFAT 5.4 only the critical SPOT result could be output together with the base material result.
- Now also the SPOT detailed result can be output together with the base material result.



Output of Cycle Limit in Haig-Diagram

- Example specimen Haigh-diagram
  - Standard-Haigh-diagram valid for 10 million cycles
  - User defined tension/compression Haigh-diagram valid for 10 million cycles
  - User defined shear Haigh-diagram valid for valid for 8 million cycles
- Example local Haigh-diagram
  - valid for 8.24 million cycles (analysed value)





DRIVING EXCELLENCE.
INSPIRING INNOVATION.