



# FEMFAT 2021 News and More

ECS Simulation Conference 2021  
Gaier, ECS

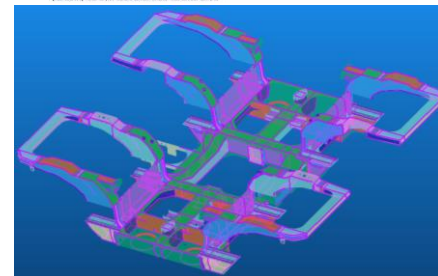
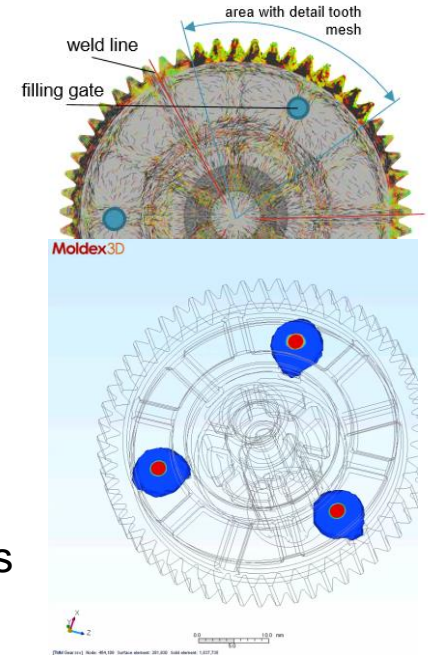
# History of FEMFAT Releases, New Versioning



- 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!



- 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  $\chi$ MCF 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
```



Parallelize Analysis

Number of Processes:  of allowable 16

License Limit: 51 Processes

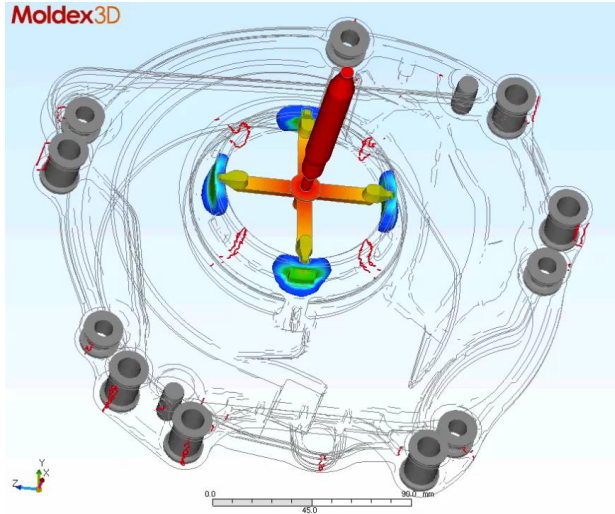
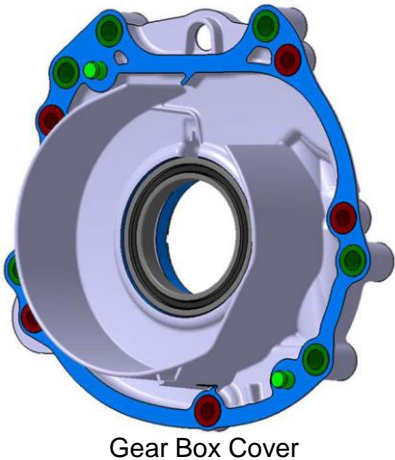
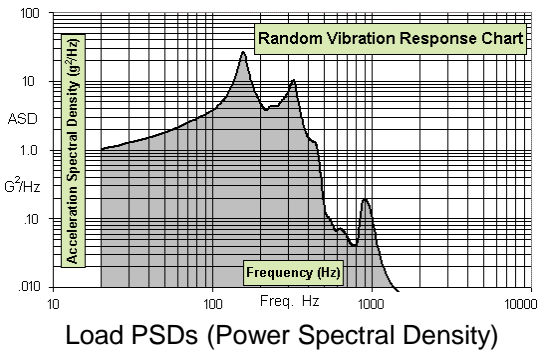
Use 1 additional License(s)

Use 90 Parallel Tokens

Remove temporary data after analysis

Deactivate node filter for model segmentation

# SPECTRAL: Assessment of Short Fiber Reinforced Plastics



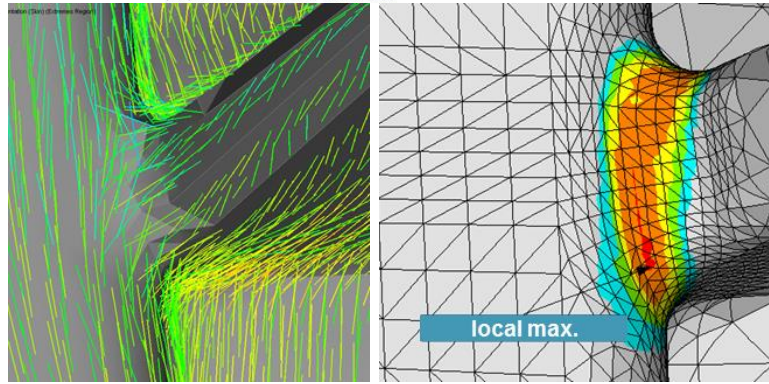
CAD

Filling Simulation (MoldFlow, Moldex3D)

Fiber Orientation & Material Mapping (Digimat-MAP)

Modal FE- Analysis Frequency Response (Nastran, ABAQUS)

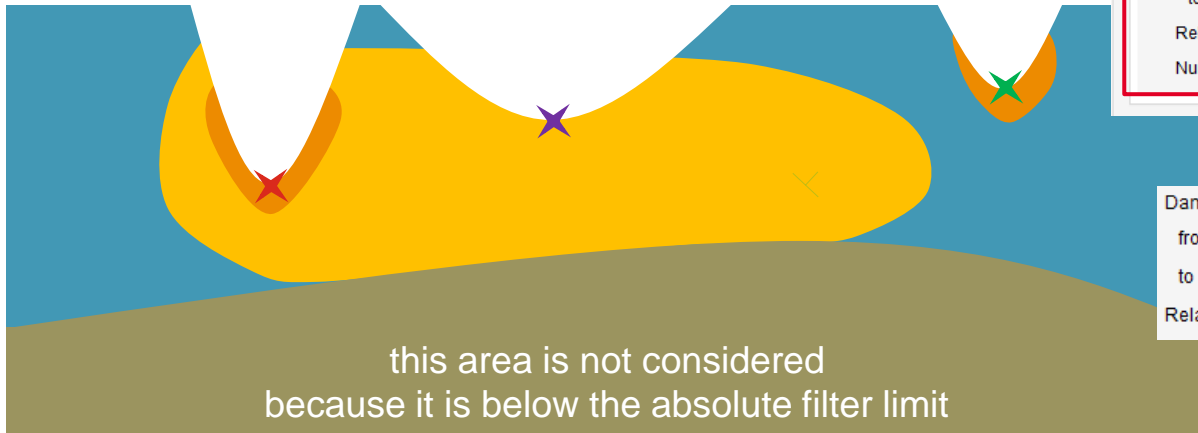
**Fatigue Analysis FEMFAT spectral**



# 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



Create/Modify Group Entries

Nodes Nodes Based on... Elements

Based on Damage Values / Safety Factors

from  to

only most critical node per SPOT nugget

Based on Isothermal Nodal Temperature [°C]

from  to

Based on Local Critical Areas

Damage / Safety Factor

from

to

Relative limit for size of local area  [%]

Number of relative extrema

Damage / Safety Factor

from

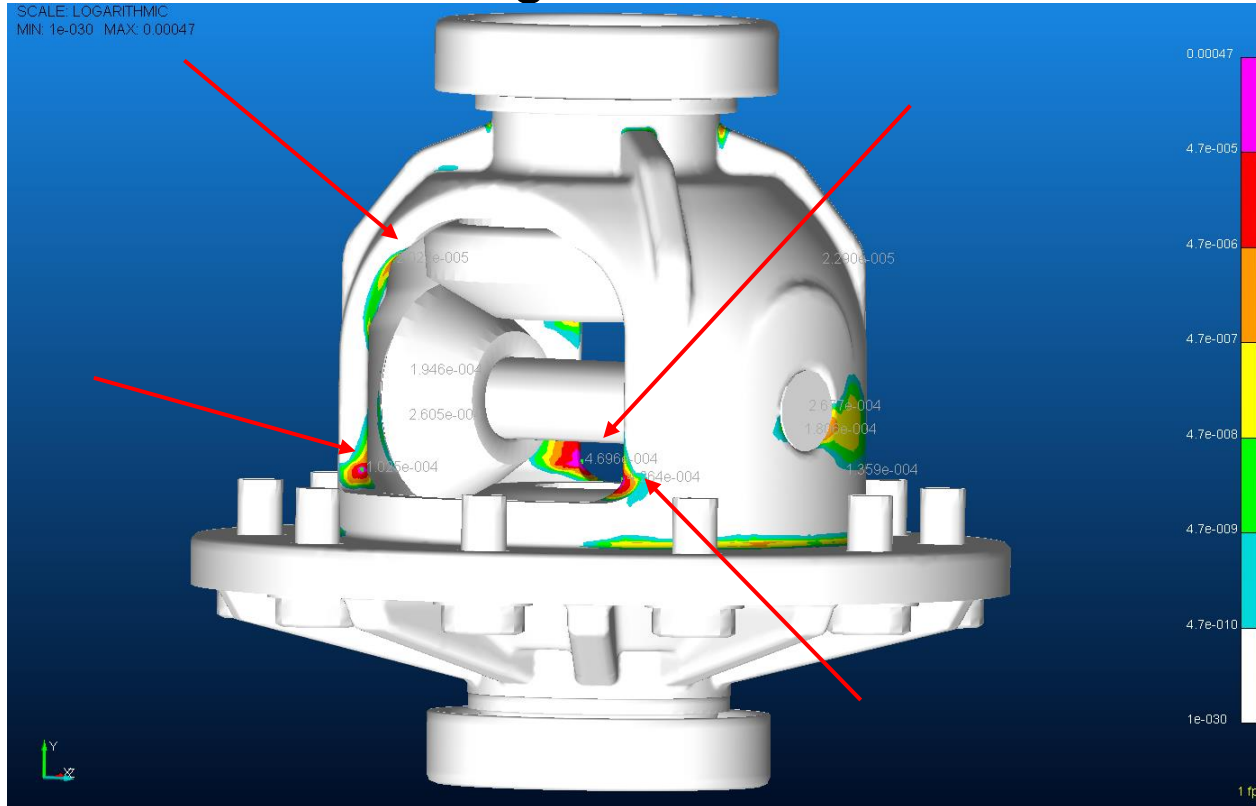
to

Relative



# 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

Report Items

General Input Data

- Header
- General Input Data
- Specimen Material Data
- Influence Factors

Structural Node Data

Damage Data/Safety Factors:  Top  Bottom

Max. Damage Component:  Top  Bottom

Stress Gradient:  Top  Bottom

Mean Stress Rearrangement:  Top  Bottom

- Surface Roughness
- Technological Size
- Tempering Condition
- Temperature
- Range of Dispersion (10% to 90%)

WELD Specific Output

- Local Direction Specific Information
- Notch Factor Specific Information

SPOT Specific Output

- Critical Results for All Sheets

Local Extrema in Areas

- Based on Local Critical Areas

Damage / Safety Factor

from:

to:

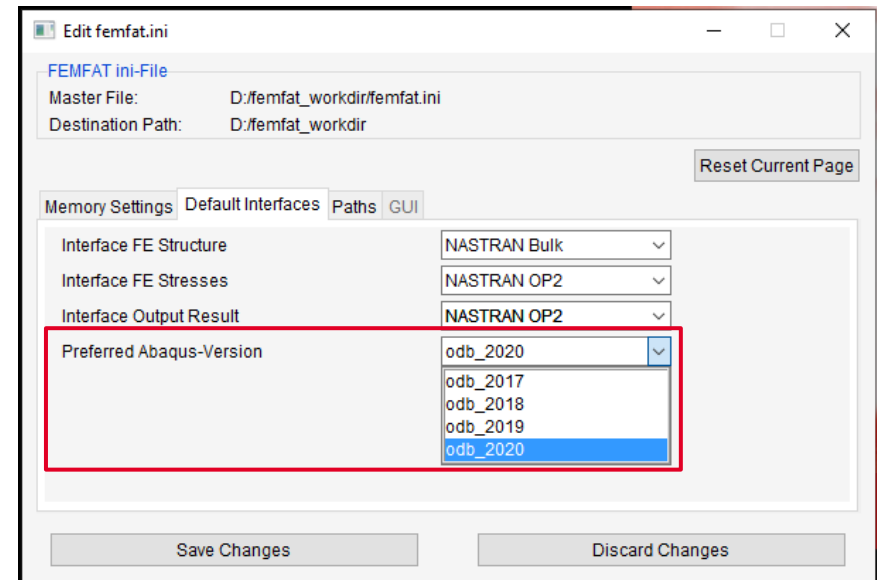
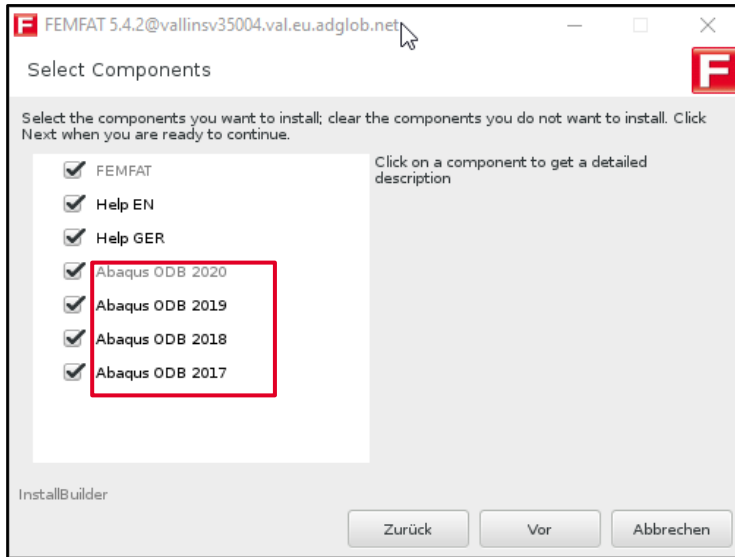
Relative limit for size of local area:  [%]

Number of relative extrema:



# 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



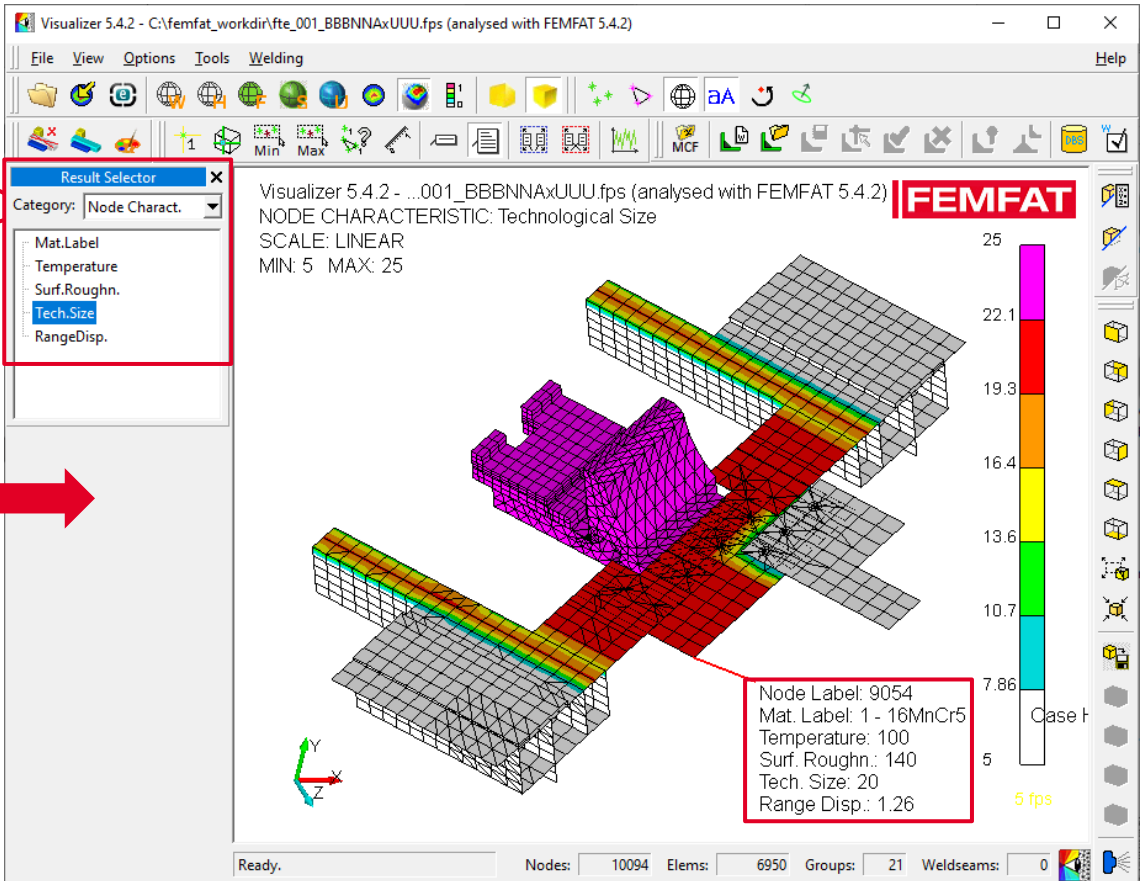
Desired properties can be selected.

- FEMFAT Results
- Animated Results
- FEA Stresses
- Node Charact.

- Result Selector
- Category: Node Charact.
- Mat.Label
  - Temperature
  - Surf.Roughn.
  - Tech.Size
  - RangeDisp.

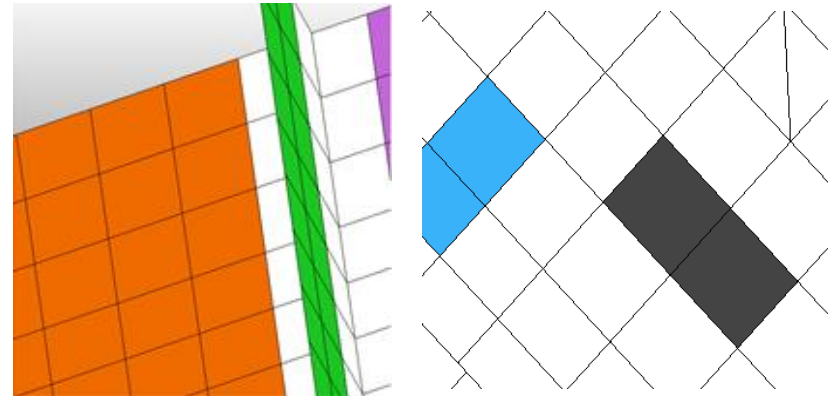
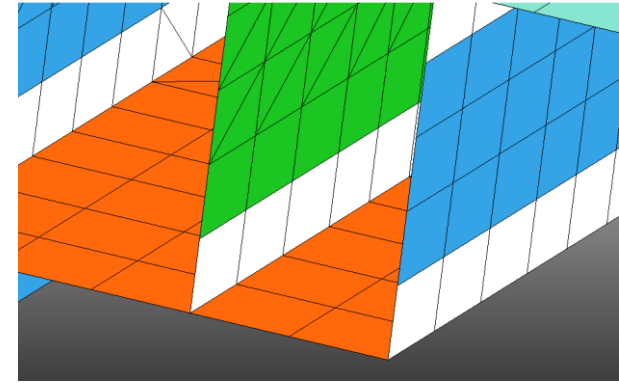
FPS Setting for Visualizer Display

- Material
- Temperature
- Surface Roughness
- Technological Size
- Range of Dispersion



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.

FPS Setting DMA-Column Setting Result Modification Module Specific

Number of Scratch Results: 21 Default  Deselect All

Main Results	Stress	General Factors1	General Factors2	Surface	Misc	Node Charact.
(7)	(5)	(4)	(0)	(0)	(0)	(5)

- Node Characteristics
  - Material
  - Temperatur
  - Surface Roughness
  - Technological Size
  - Range of Dispersion

FPS Setting DMA-Column Setting Result Modification Module Specific

Output Results Selected: Default Clear All

Col	Result	Group
1	Damage	Main Results
2	Inverse(Damage)	Main Results
3	Stress Amplitude	Stress
4	Mean Stress	Stress
5	arctan(Mean Stress/Amplitude Stress)	Stress

Main Results Stress General Factors2 Surface Misc. **Node Charact.**

Material:  Column

Temperatur:

Surface Roughness:

Technological Size:

Range of Dispersion:

Context menu (right mouse)

- Clear
- Clear All
- Column
- Replace with...
- Insert
  - Main Results
  - Stress
  - General Factors1
  - General Factors2
  - Surface
  - Misc.
  - Node Charact.

- Material
- Temperatur
- Surface Roughness
- Technological Size
- Range of Dispersion

# VISUALIZER: Add and Rearrange Multiple Subwindows



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

2.

- Add Subwindow
- Add all Subwindows
- Delete Subwindow
- Configure Subwindows
- Create a Group
- New Weldseam (S)
- Modify a Weldseam (M)
- Show/hide all Weldseams
- End of Weldseam Definition (E)
- Show History
- Select/Deselect Node (N)
- Deselect all Nodes
- Rotation Center (D)
- Select/Deselect Element
- Deselect all Elements
- Done

1.

3.

Visualizer 5.4.1 - D:\mryesamples\02\_06\_CHANNELMAX\_DIFFGEAR\FEMFAT21\_Max\_Diffgear\_testrun.fps (analysed with FEMFAT 5.4.1)

RESULT: Damage  
SCALE: LOGARITHMIC  
MIN: 1e-030 MAX: 0.00047

Node Label	Damage Mmod	1/Damage	Rel Str Grad	Log10 Damage	Log10 1/Dam	6th Root Dam	Mean Stress	Str. Ratio R	atan(Sm/Sa)	LocFatigLim
59395	2.022e-005	4.944e+004	0.34629	-4.694	4.694	0.1651	308.6	-0.6674	11.23	224.1
59900	2.605e-004	3.833	0.4423	-3.711	3.711	0.2407	533	-1.221	-5.687	270.8
59900	2.605e-004	3.833	0.3774	-3.584	3.584	0.2527	532	-1.222	-5.702	270.8
28752	2.677e-004	3.735	0.4987	-3.572	3.572	0.2539	570.1	-1.189	-4.166	281.3
249190	2.290e-005	4.368e+004	0.3213	-4.64	4.64	0.1685	301.9	-0.6239	13.04	216.8
87781	4.696e-004	2.129	0.432	-3.328	3.328	0.2788	495.3	-0.8893	3.354	243.8
269370	4.264e-004	2.345	0.409	-3.37	3.37	0.2743	490.5	-0.9381	3.074	243
288202	1.806e-004	5.537	0.3875	-3.743	3.743	0.2378	513.6	-1.236	-6.018	271.9
261262	1.359e-004	7.359	0.3867	-3.867	3.867	0.2267	465.4	-1.126	-3.401	265.9
87412	1.025e-004	9.757	0.4298	-3.989	3.989	0.2163	446.9	-1.121	-3.263	264.9

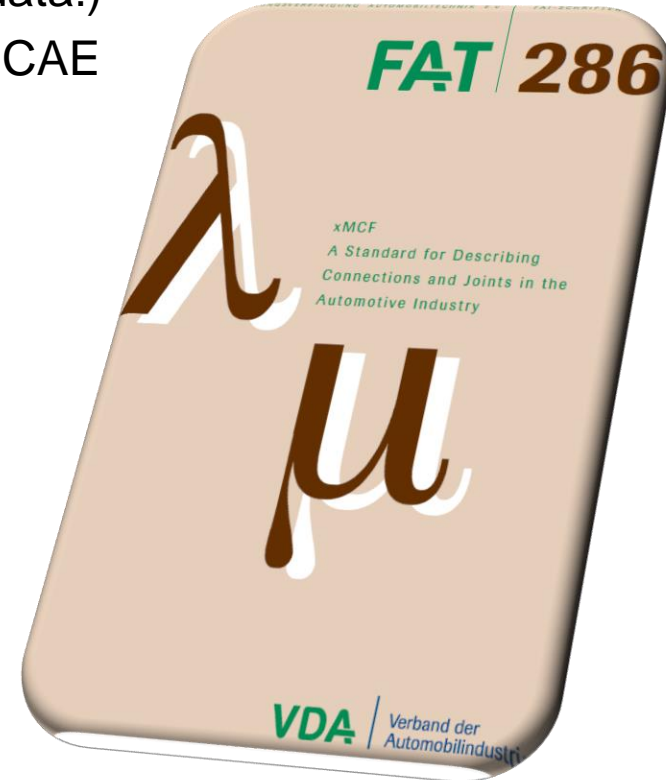
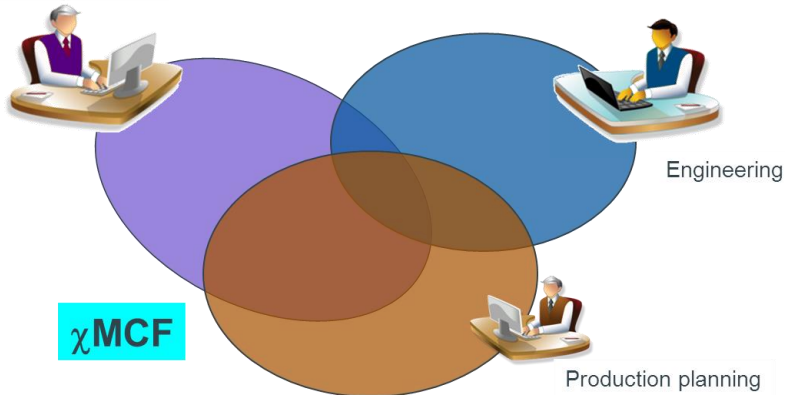
Ready.

Nodes | 307541 | Elements | 201883 | Groups | 67 | Weldseams | 0

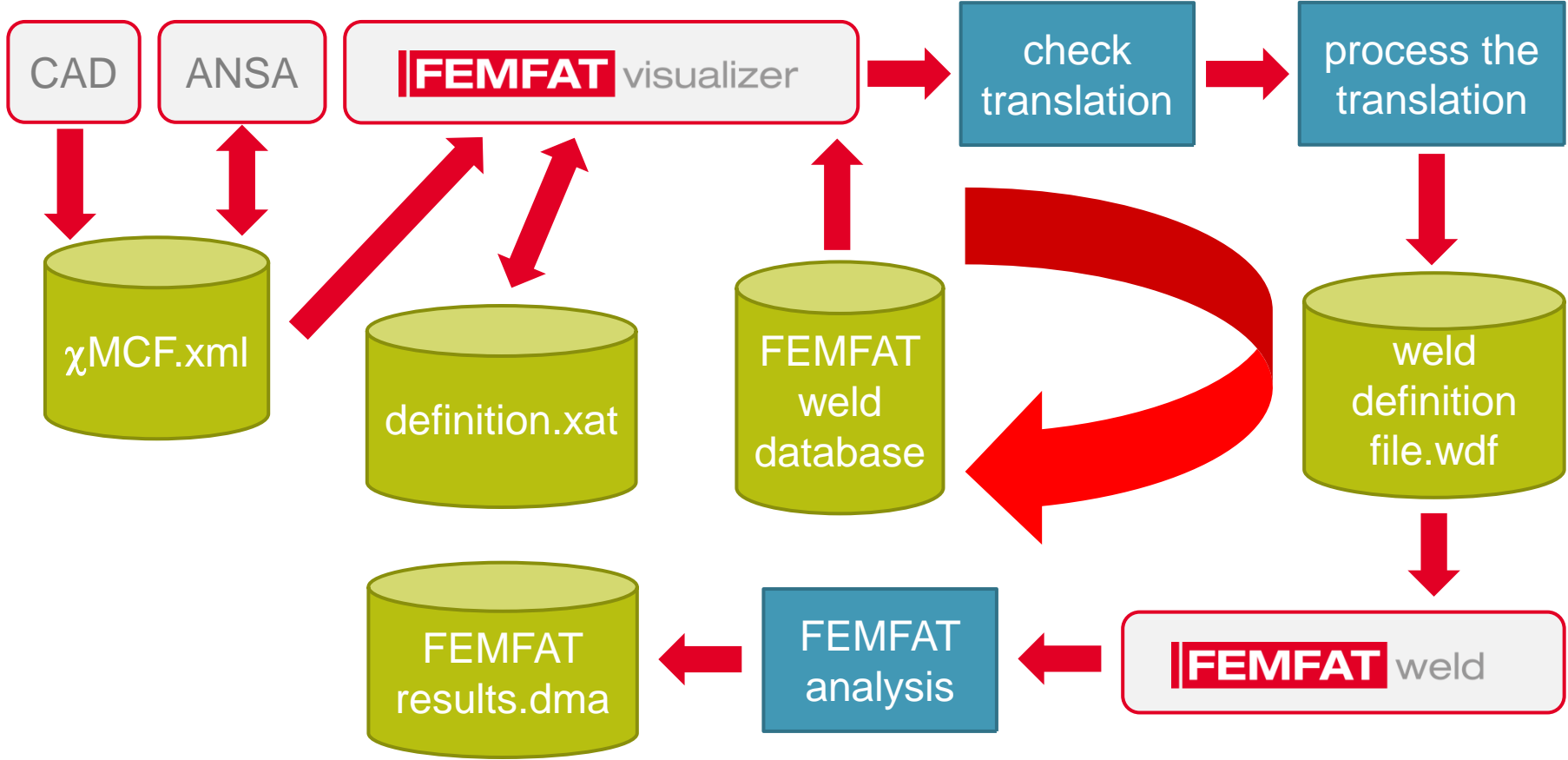
# The Idea behind $\chi$ MCF

- $\chi$ MCF 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

Design, Construction

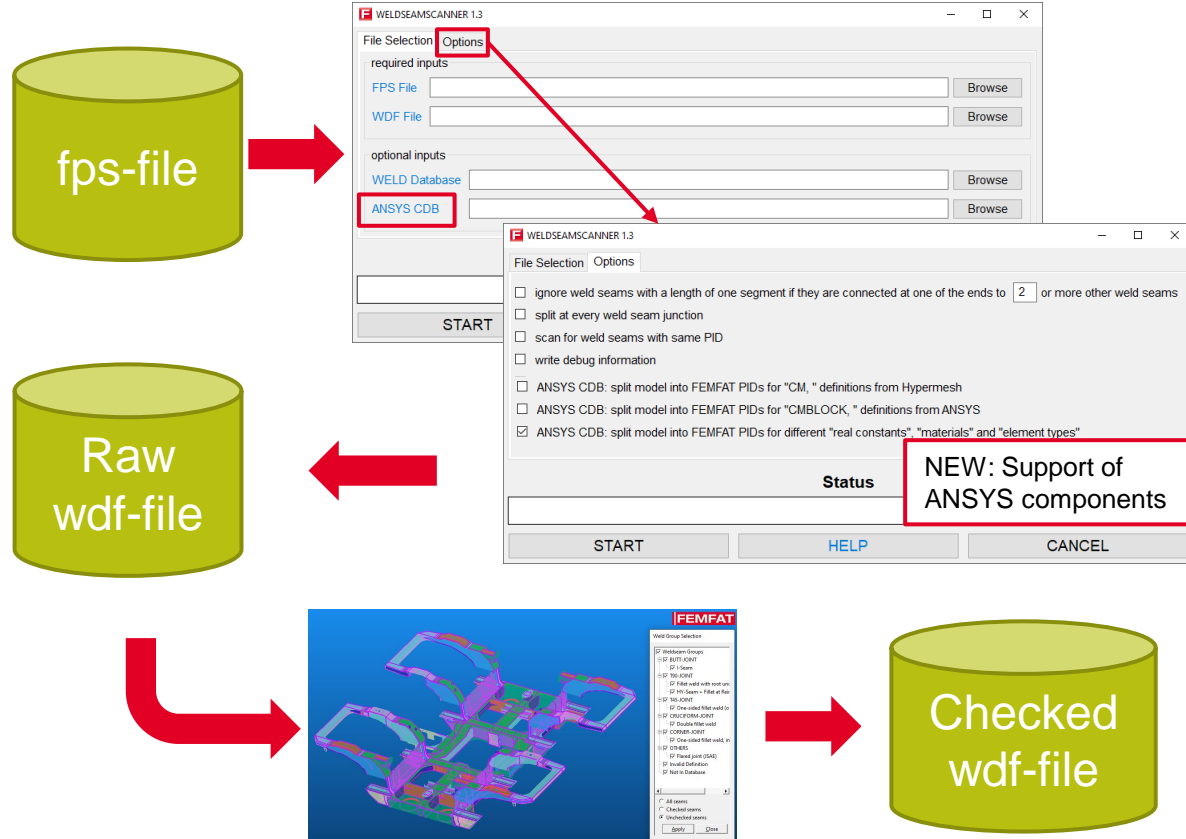


# Workflow of Weld Definition via $\chi$ MCF 3.0



# 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

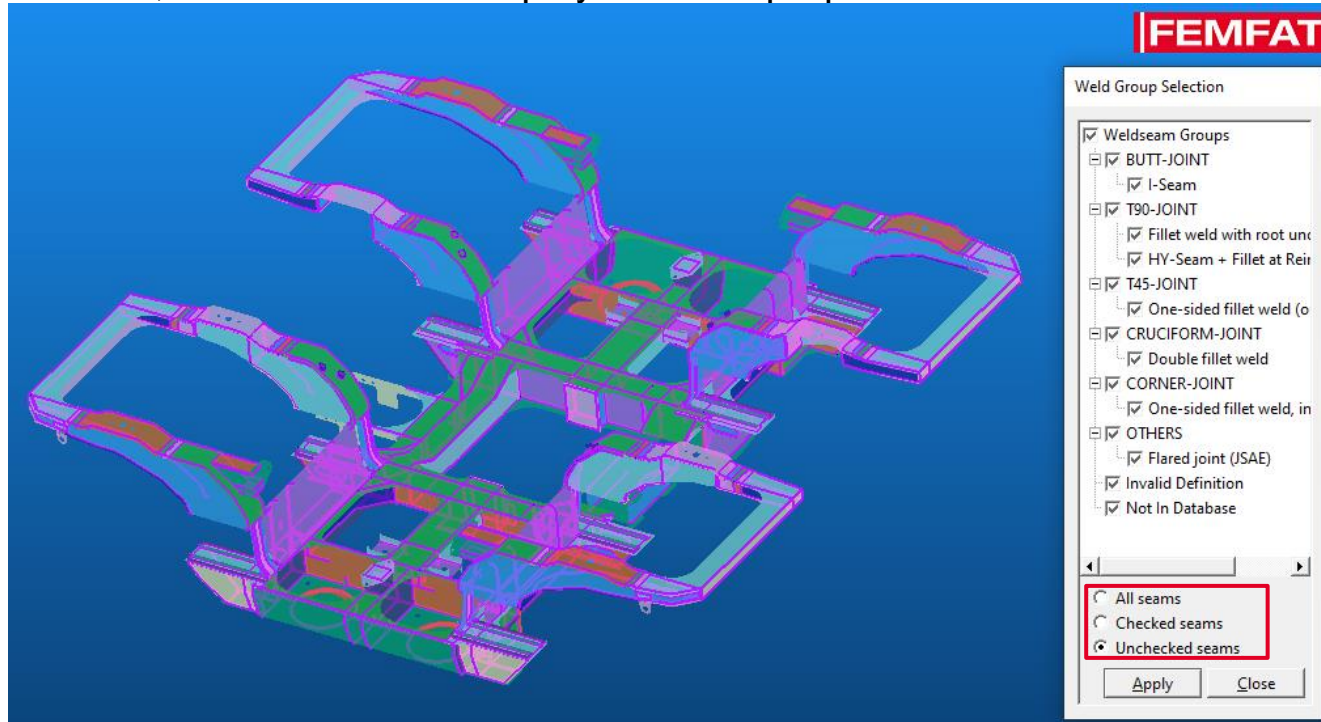




# New Functionalities for the Checking of Welding Seams



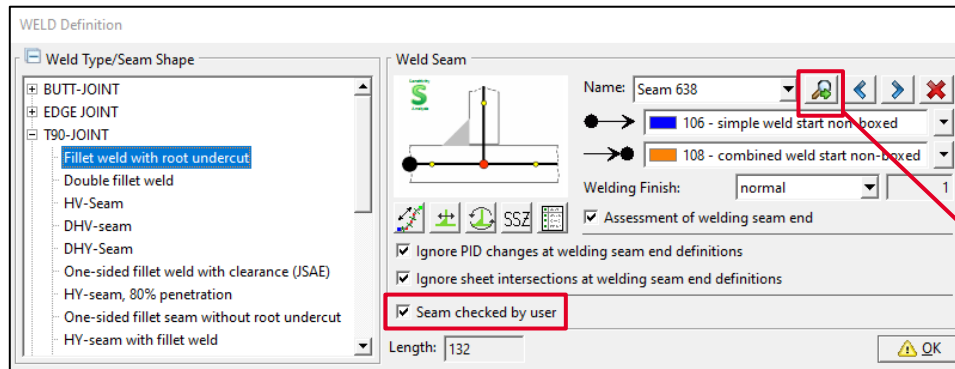
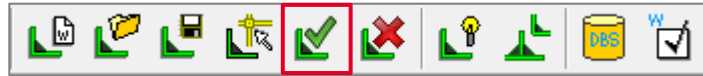
- Welding seams with missing information (e.g. welded side) that have been automatically generated (via xMCF, Weld Seam Scanner or automatic seam separation) are marked as “Unchecked” → Checking the weld definition is strongly recommended!
- In the VISUALIZER, such seams are displayed with a purple seam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



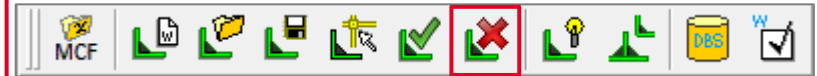
# New Functionalities for the Checking of Welding Seams



- 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.



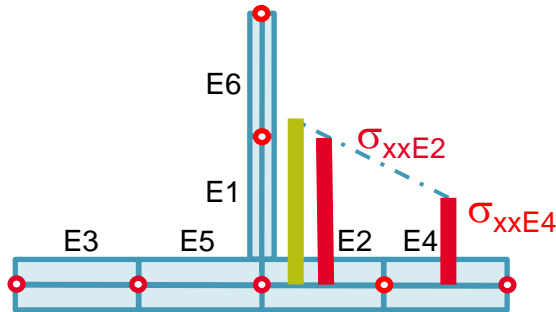
## Also new:



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



WELD Stress

Stress Averaged at Element

Automatic Stress Correction

Sheet Thickness Factor:

Stress Interpolation Parameters: A:  B:  C:

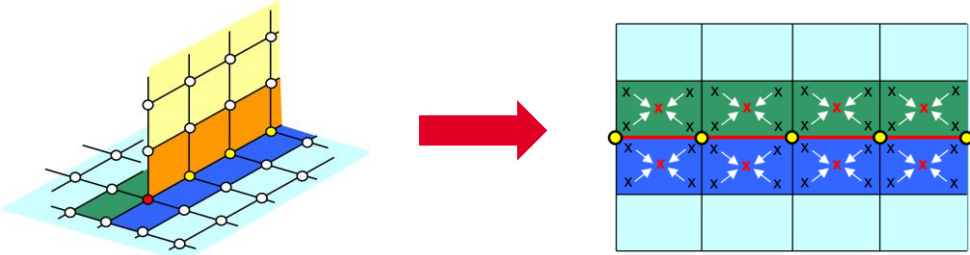
Stress Averaged along Seam

Element Nodal Stress (non Averaged)

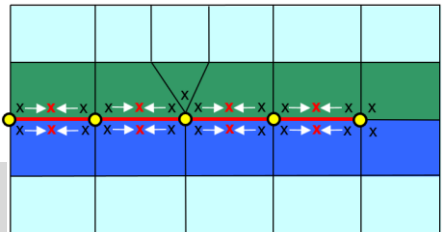
# WELD: Extended Stress Averaging Possibilities



1) By default, element nodal stresses will be averaged

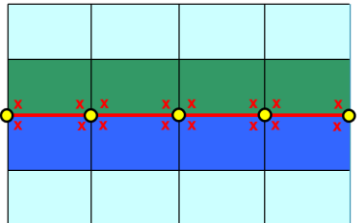


2) Averaging along the welding seam

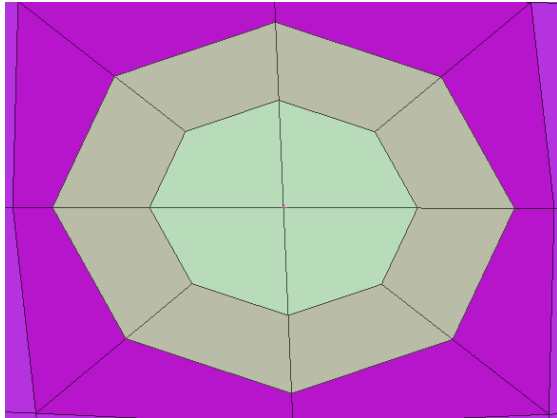


x ... Element nodal stresses (Cubic stresses)  
x ... Averaged stresses

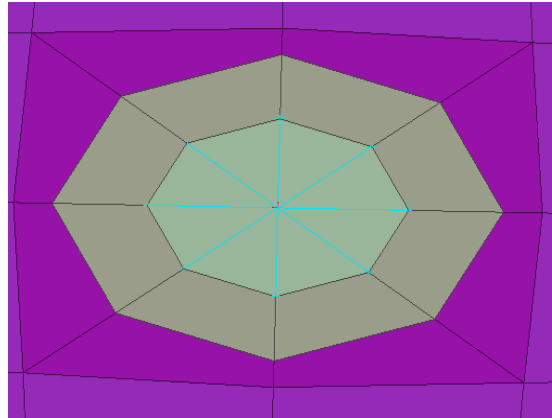
3) Usage of element nodal (cubic) stresses directly



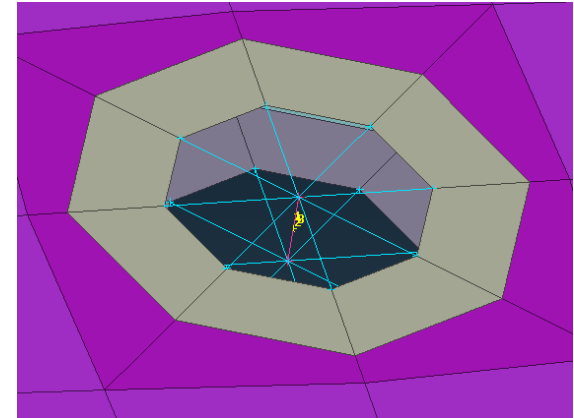
- Following modelling types will be supported in FEMFAT spot:
  - Added the RBE2 in the center of the nugget (modelling variant 1)
  - Removed inner shell elements and added RBE2 (modelling variant 2)



Standard model



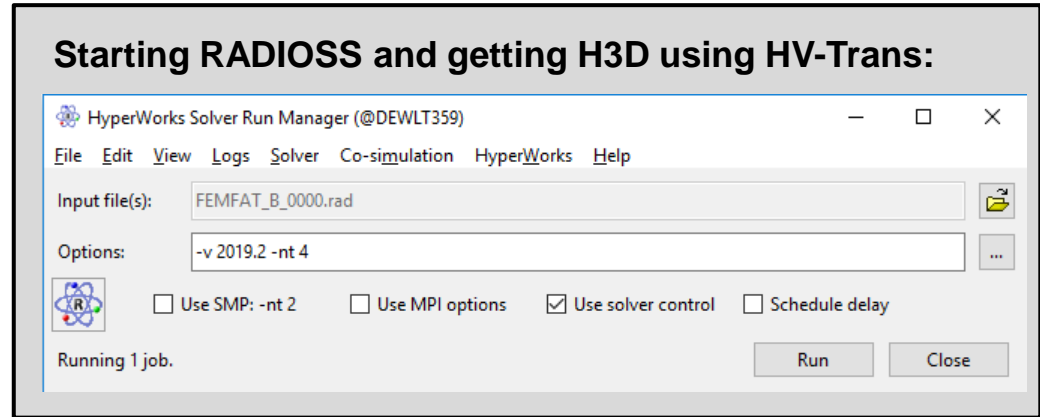
Standard model +  
RBE2 (variant 1)



Replaced inner shell  
with RBE2 (variant 2)

# Creation Options of H3D Files in Radioss

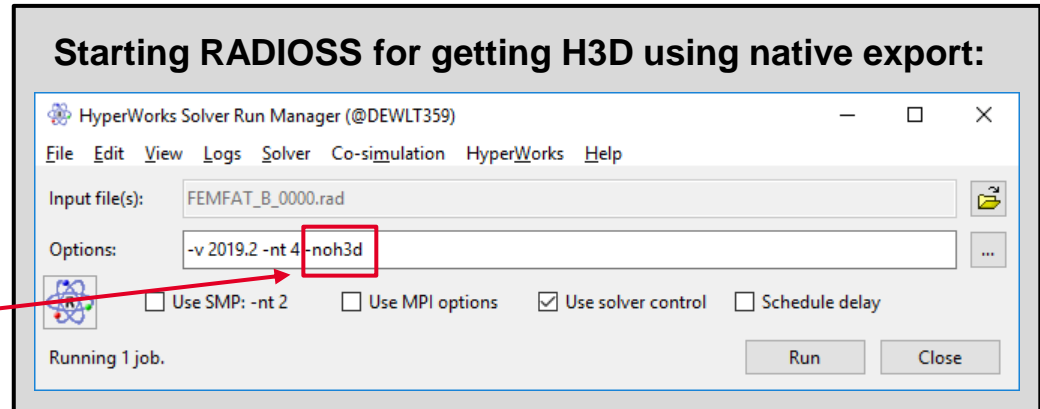
- Supported in FEMFAT:



- 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.



# Display of Load Spectra for “Detailed Results” Group



ChannelMAX Output

FEMFAT Permanent Scratch File

Export File Name: femfat18.fps

FPS Setting DMA-Column Setting Result Modification Module Specific

MAX SPOT STRAIN

DETAILED RESULTS Files

Maximal Stresses:  FPS  CSV femfat.cms

Unit Stresses:  FPS  CSV femfat.cus

Equivalent Stresses:  FPS  CSV  RPC Binary femfat.esh

Equivalent Stress of  FPS

**Load Spectra:  FPS**

Partial Damage:  FPS  CSV  RPC Binary femfat.pdh

Total Damage:  FPS  CSV  RPC Binary femfat.tdh

Critical Load Case

Output File: femfat\_CritLC.dat

Load Case Number from...: File NASTRAN Bulk

Load Case Definition:  with Max Amplitude + Max/Min Value  at Time Point 1

Write

ChannelMAX Visualization

FE Entities

Groups

Channels

Material Data

Node Characteristics

Influence Factors

Strain Gage Data

Analysis Parameters

Scratch Setting

Output

Report

Analyze

Visualization

BASIC

ChannelMAX

TransMAX

HEAT Sehitoglu

SPECTRAL

SPOT Remeshing

STRAIN Calc

Results Manager

Visualization

Analysis Result

Charts

S-N

Hair

Rainflow

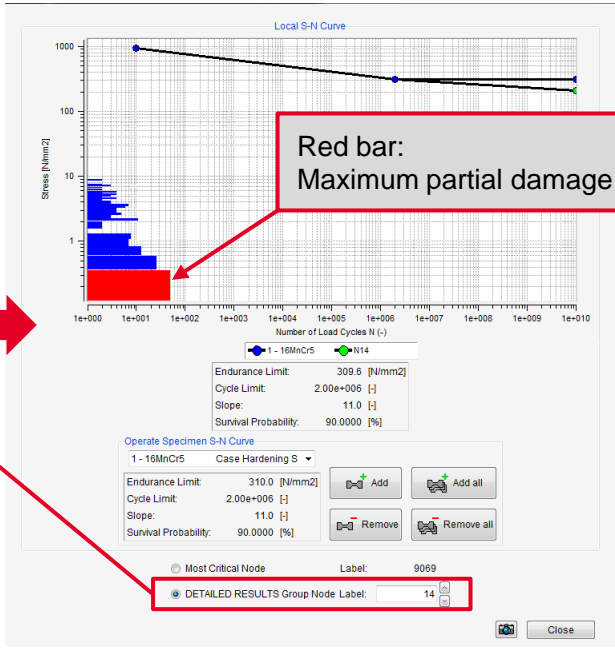
History Charts

Strain

Damage

Strain Gage Chart

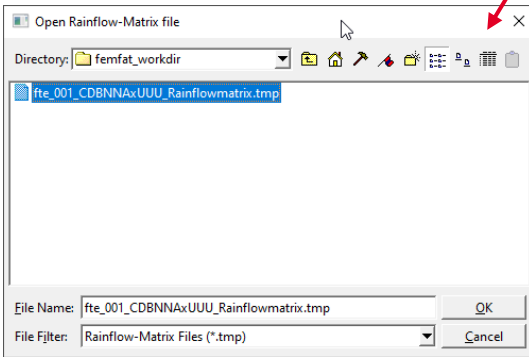
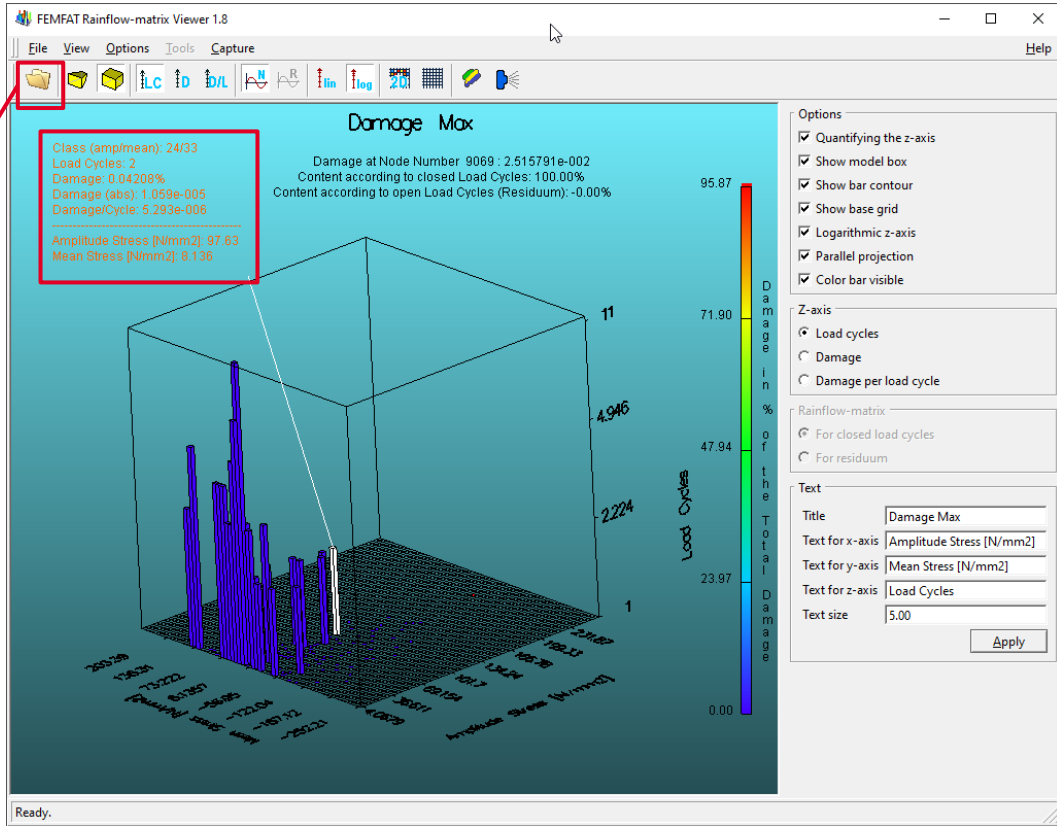
FEMFAT Visualizer



# 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



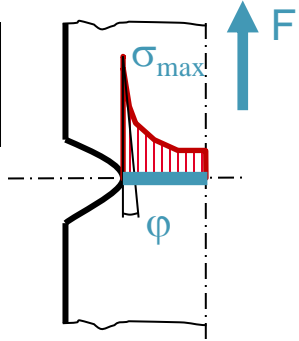


# New Features of FEMFAT 5.4.2 released February 2021

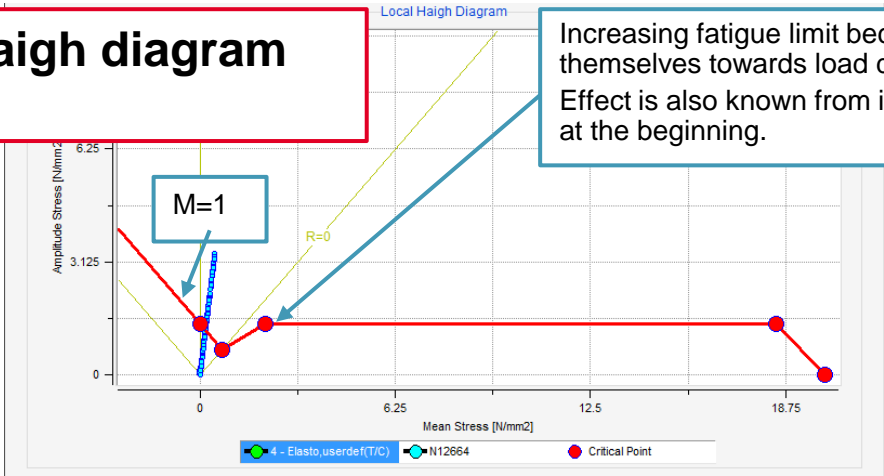


- **New stress gradient analysis method**
- Results Manager: multiple fps-file selection
- Results Manager: Enhanced 'Formula' combination method
- SPECTRAL: Output of equivalent stress PSDs as ASCII CSV file
- Dynamic memory allocation for load histories
- Import of averaged-at-nodes stresses (and strains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

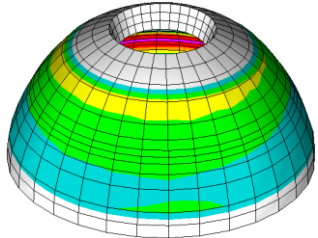
$$\chi' = \frac{1}{\sigma_{max}} \left( \frac{d\sigma}{dx} \right)$$



## User defined Haigh diagram for elastomers

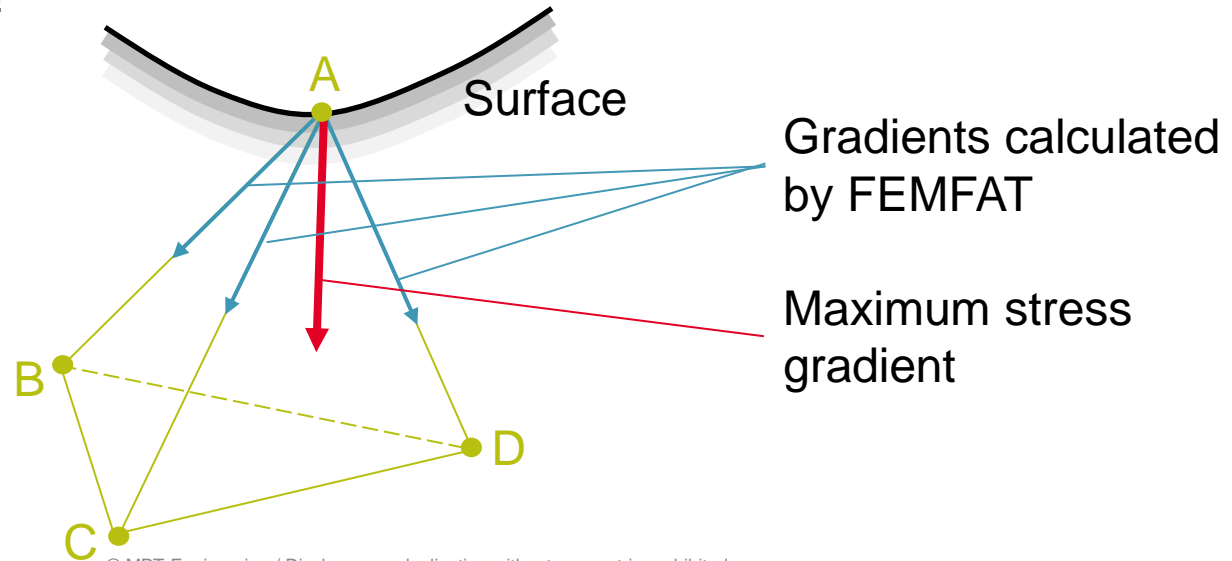


Increasing fatigue limit because of strain crystallization: Chains orient themselves towards load direction and block cracks.  
Effect is also known from inflating balloons, which needs more power at the beginning.



- **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!



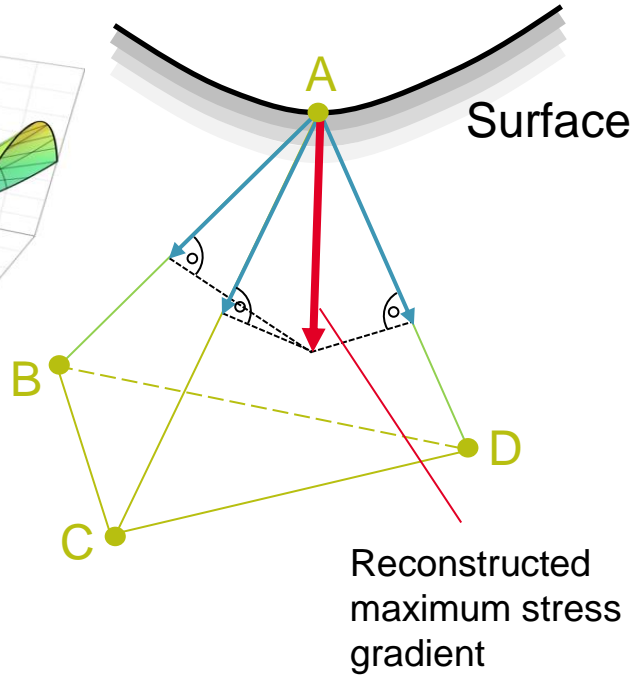
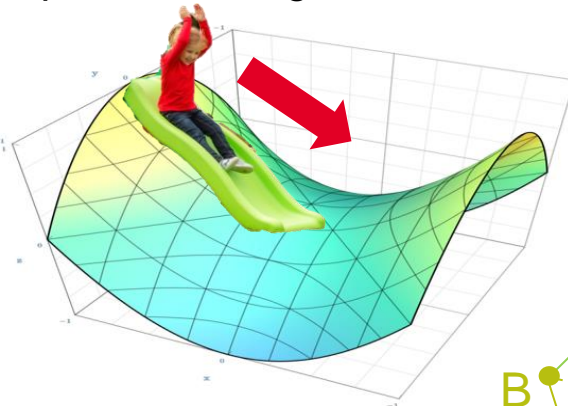
# New Stress Gradient Analysis Method



- **Our solution**

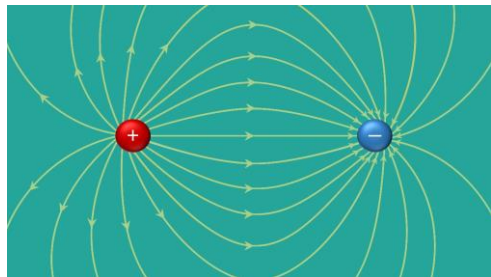
- A stress gradient is a vector pointing in the direction of the largest stress decrease, which can be reconstructed by its components along the Finite Element edges:

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



- Analogon: Static electric field:

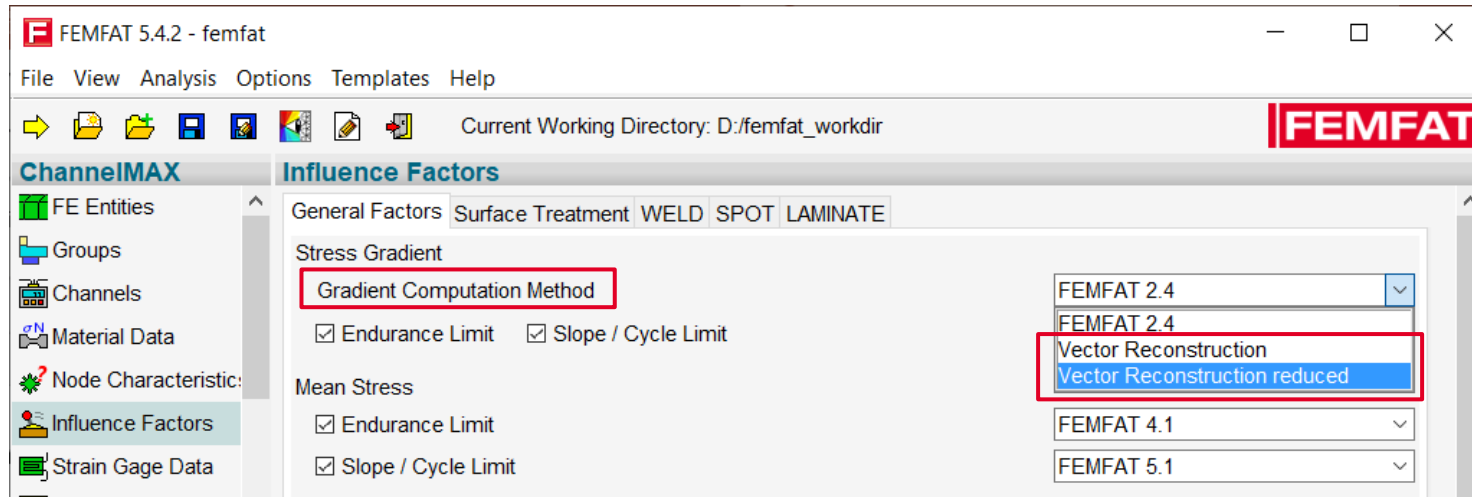
$$\vec{E} = -\vec{\nabla}\varphi$$



# New Stress Gradient Analysis Method

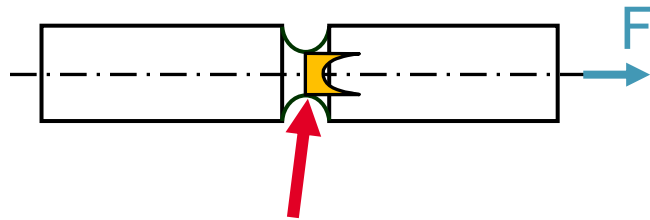


- Additional improvements
  - Usage of superimposed stresses in ChannelMAX
    - Ensures the invariance of the stress gradient with respect to the analysis type (static mounting or inertia relief)
  - Equivalent method to TransMAX. Same results are obtained for same loading.
  - **Vector Reconstruction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

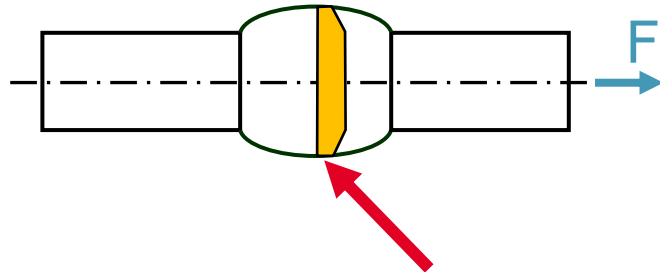


- Further improvements

- 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

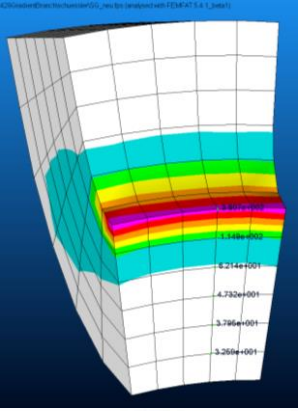
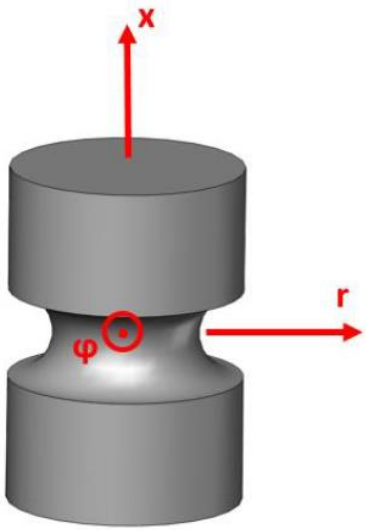


# New Stress Gradient Analysis Method



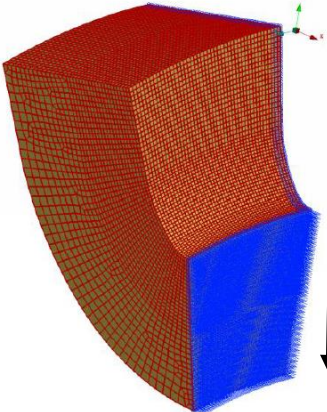
- Stress gradient progress at notch root into the depth

Specimen



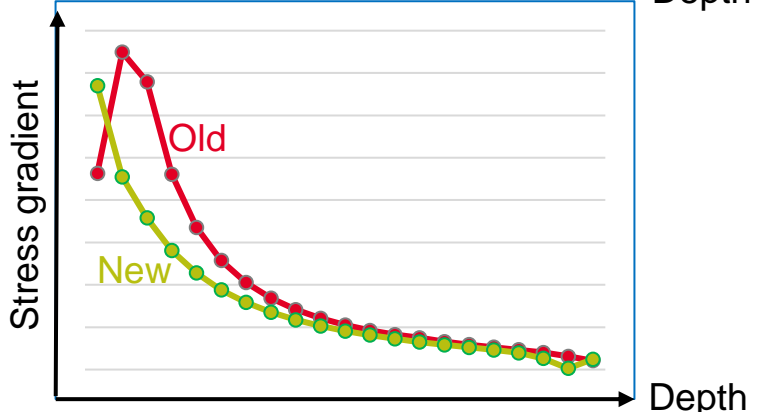
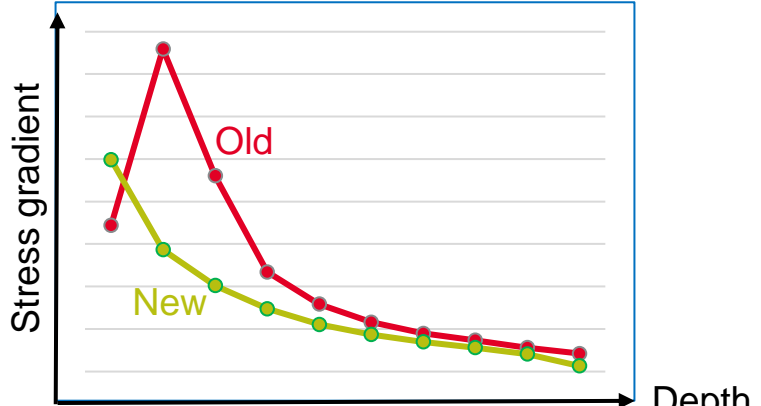
Rough mesh

Depth



Fine mesh

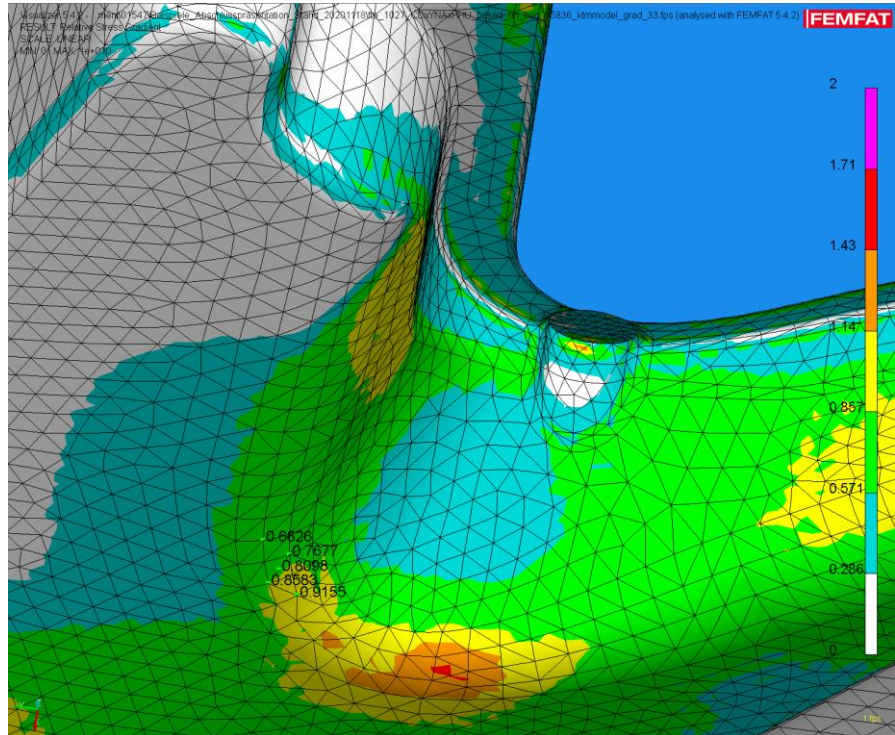
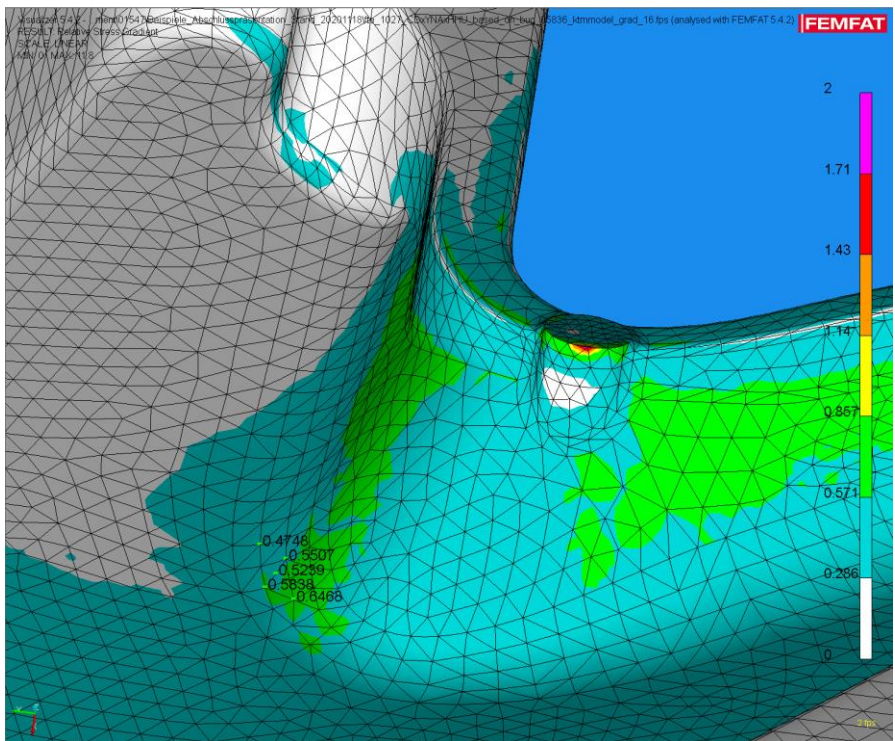
Depth



# New Stress Gradient Analysis Method, Example



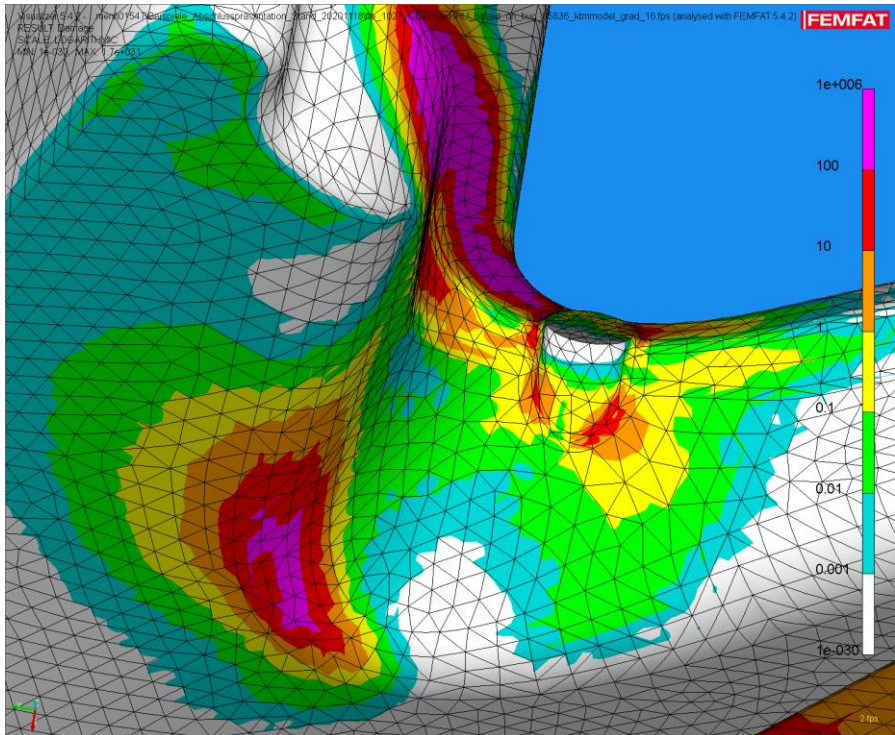
- Stress gradient result, old method
- Vector reconstruction



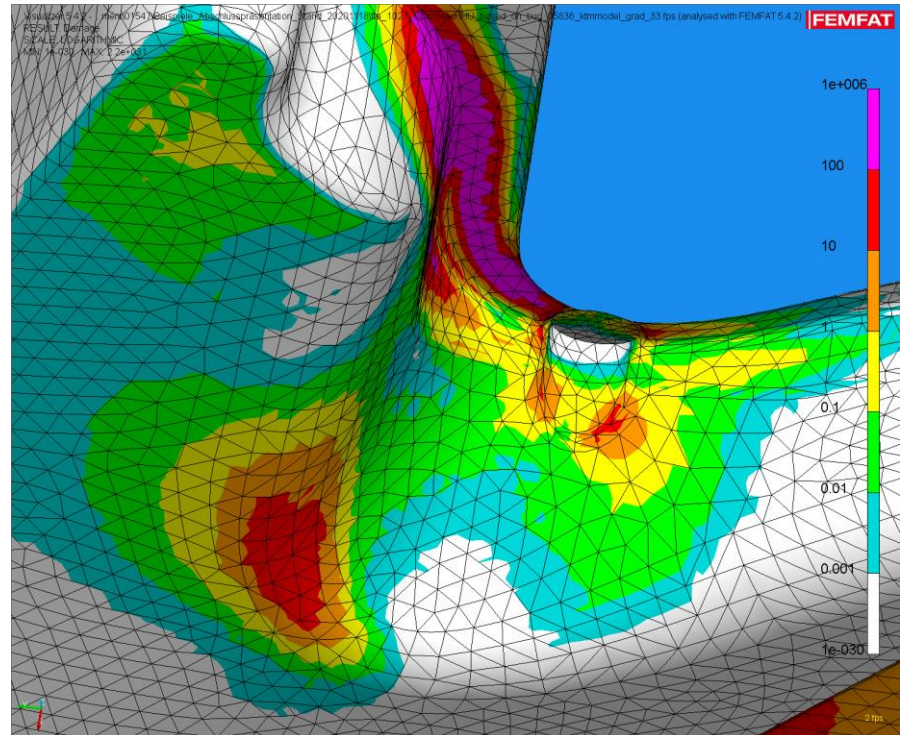
# New Stress Gradient Analysis Method, Example



- Damage result, old method



- Damage distribution, vector reconstruction

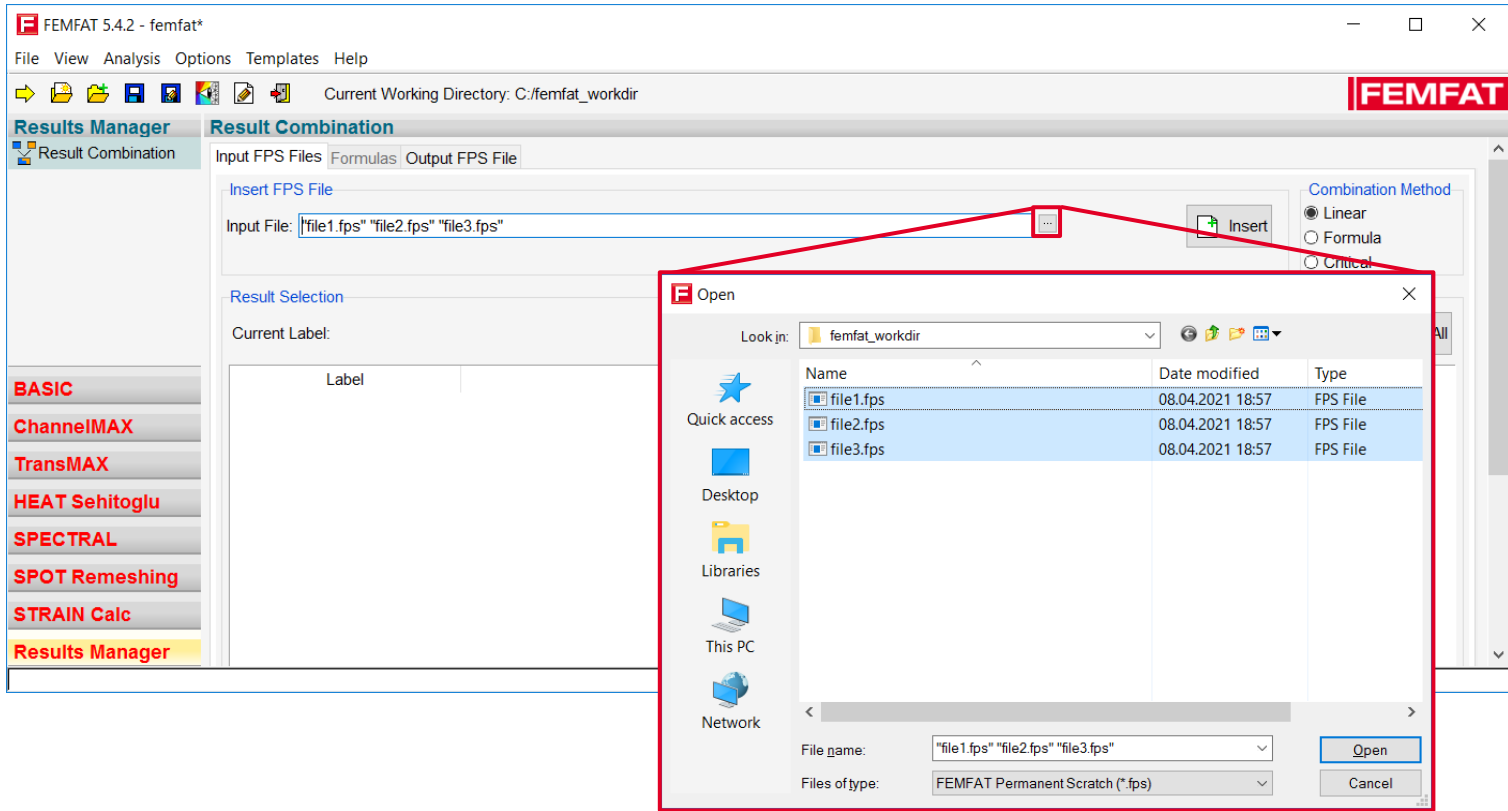




# Results Manager: Select Multiple fps-Files




- 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. 

## Result Combination

Input FPS Files Formulas Output FPS File

### Formula Definition

Name	Formula	Critical Result
User def 1	[File_1:Damage_M]mod*3	Max
User def 2	[File_2:Stress_Ampl.]	Max
User def 3		Max
User def 4		Max
User def 5		Max
User def 6		
User def 7		

Templates Help

- Elastomere\_Analysis\_Settings
- GL\_2010
- WELD\_Sensitivity\_Damage\_all
- WELD\_Sensitivity\_Damage\_gap
- WELD\_Sensitivity\_Damage\_inclination\_angle
- WELD\_Sensitivity\_Damage\_penetration\_degree
- WELD\_Sensitivity\_Damage\_seam\_thickness

### Critical Position from

Formula Output Result

- Formula Output Result
- All Input Files
- All Files in Formula
- File\_1
- File\_2
- Formula Output Result

Formula Output Result: The critical layer will be determined from the formula result after combination.

All Input Files: The critical layer will be determined from the most critical result of all input files before combination.

All Files in Formula: The critical layer will be determined from the most critical result of all files used in the formula before combination.

Selected File: The critical layer will be determined from the most critical result of the selected file before combination.

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



- Output for all nodes in the **DETAILED RESULTS** group

**SPECTRAL** Output

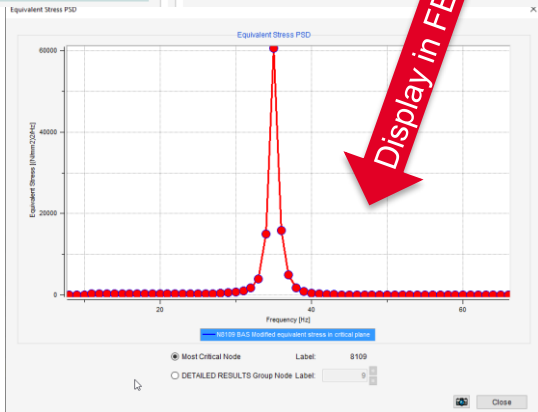
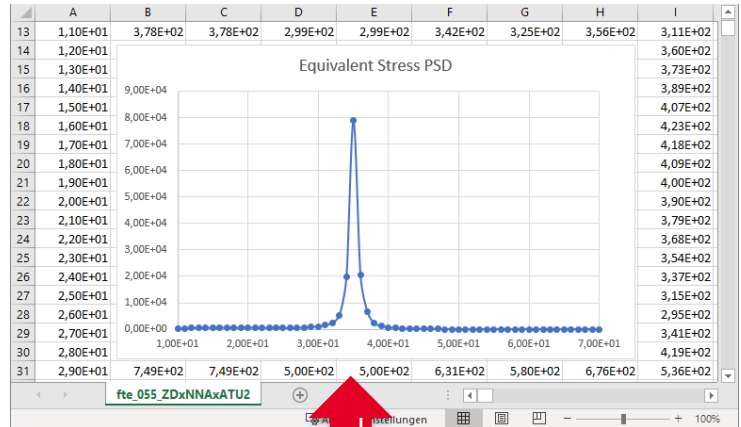
FEMFAT Permanent Scratch File

Export File Name:  Import

FPS Setting DMA-Column Setting Result Modification Module Specific

**SPECTRAL** SPOT

Equivalent Stress PSDs:  FPS  CSV



# 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**

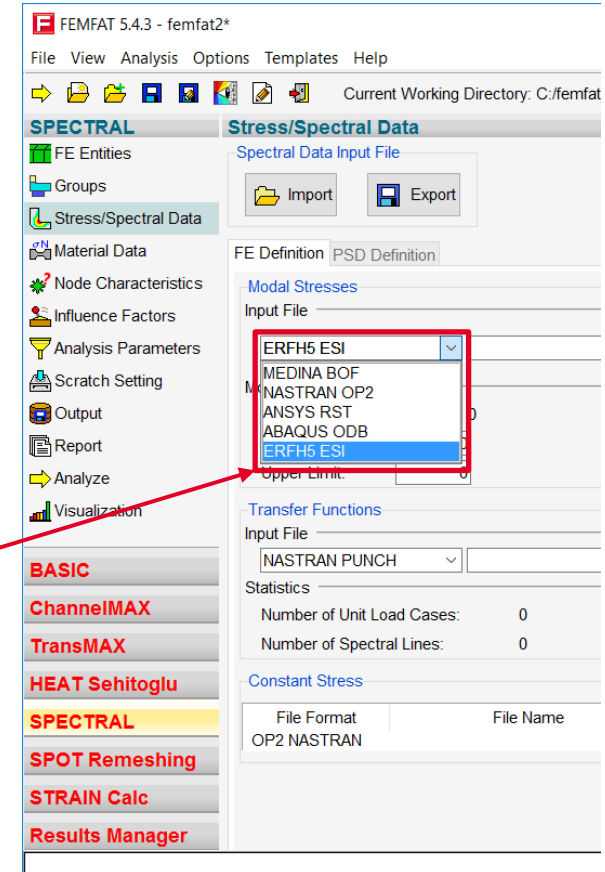
- Display exact point coordinates in the S-N diagram and Haigh diagram
- Critical load combination - separate output of times for maximum and minimum stress
- WELD: Pro-file output of both ROOT and TOE results as table
- Simultaneous output of SPOT detailed results and base material results to dma-file
- 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**

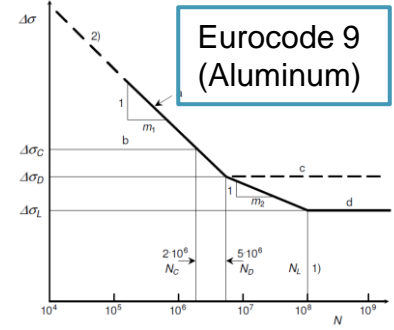
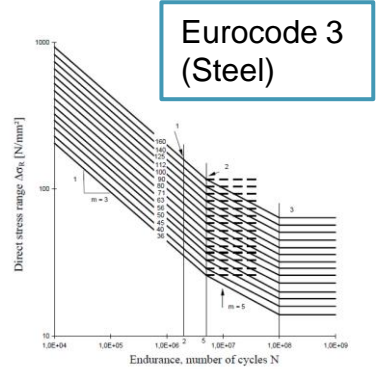
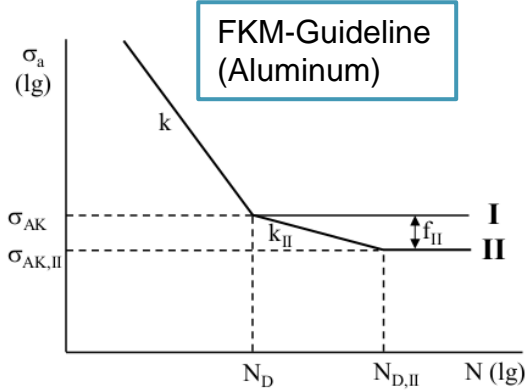
- ALTAIR licensing for FEMFAT inside ANSYS



# New Multi-Segment S-N Curve Model



- S-N curves from different standards:

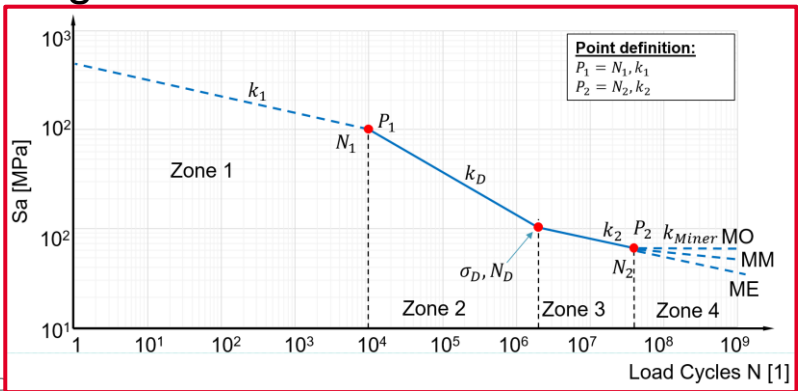


- 1 Detail category  $\Delta\sigma_c$
- 2 Constant amplitude fatigue limit  $\Delta\sigma_D$
- 3 Cut-off limit  $\Delta\sigma_L$

Figure 6.1 — Fatigue strength curve  $\log\Delta\sigma\text{-}\log N$

- 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:  (optional, will be autocompleted)

Cycle Limit:  (optional, will be autocompleted)

Slope of S-N Curve for Shear Loading:  (optional, will be autocompleted)

Cycle Limit for Shear Loading:  (optional, will be autocompleted)

**High Cycle Fatigue**

Slope of S-N Curve:

Cycle Limit of Endurance:

Stress Limit of Endurance:  [N/mm2]

Cycles at Failure:  (not used for analysis)

Ultimate Strength:  [N/mm2] (not used for analysis)

Survival Probability:  [%]

Slope of S-N Curve for Shear Loading:

Cycle Limit of Endurance for Shear Loading:

**Very High Cycle Fatigue**

Slope of S-N Curve:

Cycle Limit:

Slope of S-N Curve for Shear Loading:

Cycle Limit for Shear Loading:

**MINER Modified**

Parameter a:

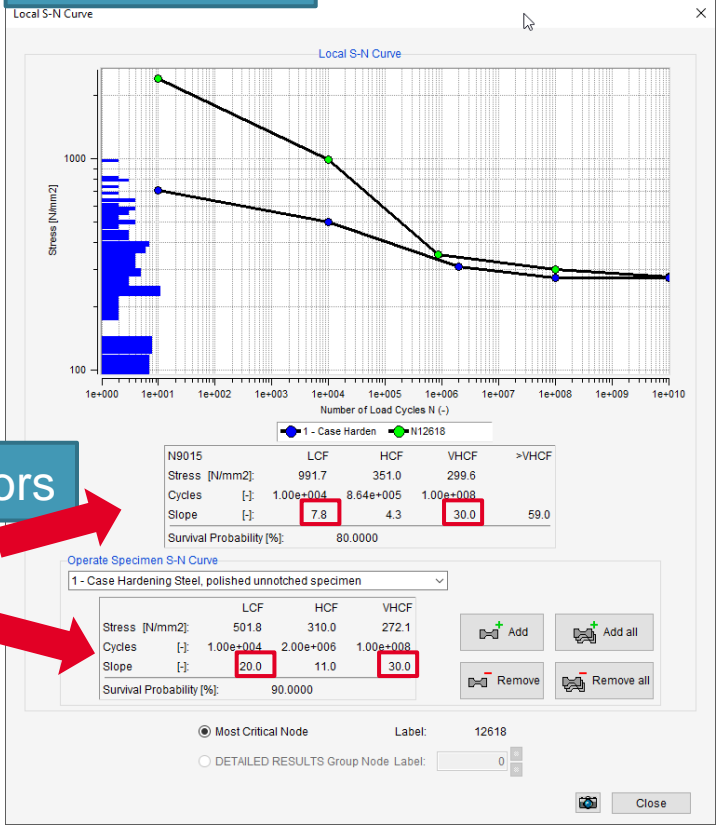
Parameter b:

## Influence Factors

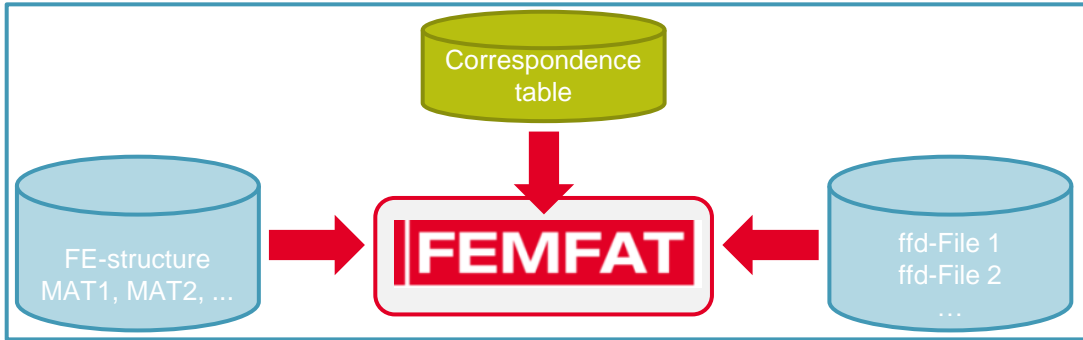
Modification Method for Multi-Segment S-N Curve

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

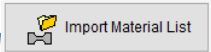
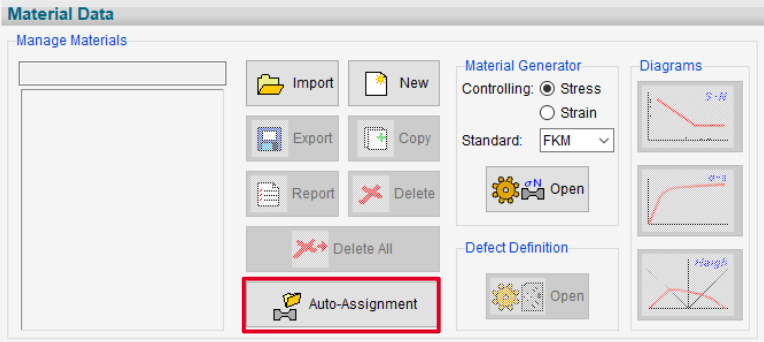
## Visualization



# 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	
	2,3;4	2	EN-AC-21100_sand_casting_FKM2002.ffd	Aluminum	
	5-6	6	EN-E295_FKM2002.ffd		
			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.

The screenshot displays the ChannelIMAX software interface. On the left is a navigation tree with categories like BASIC, ChannelIMAX, TransMAX, HEAT, SPECTRAL, SPOT, STRAIN, and Results Manager. The main window is titled 'Channels' and shows 'Channel Definition' settings. A table lists three channels with columns for Lbl, Format, Stress File, LC, Factor, L.HIST, Load History File, Row, Col, and Scratch File. A 'Generator' button is highlighted with a red box. Below the table, a 'Channel Generator' dialog box is open, showing settings for Stress File, Multiplication Factor, Load History File, and Scratch File Base Name. Red arrows point from the dialog box to the 'NASTRAN Subcases' and 'Header rsp-file' sections.

**Channel Definition Table:**

Lbl	Format	Stress File	LC	Factor	L.HIST	Load History File	Row	Col	Scratch File
1	OP2 NAS...	OP2_File.op2	1	1.00000	RPC Binary	RPC_File.rsp	1	3	femfat_1.fss
2	OP2 NAS...	OP2_File.op2	2	1.00000	RPC Binary	RPC_File.rsp	1	1	femfat_2.fss
3	OP2 NAS...	OP2_File.op2	3	1.00000	RPC Binary	RPC_File.rsp	1	2	femfat_3.fss

**Channel Generator Dialog Box:**

- Stress File: File Format: NASTRAN OP2, File Name: OP2\_File.op2
- Multiplication Factor for Stresses: Factor: 1.00000
- Load History File: File Format: RPC Binary, File Name: RPC\_File.rsp
- Scratch File Base Name: File Name: femfat

**NASTRAN Subcases:**

```

SUBCASE 1
LABEL = Fz
LOAD = 1
SPC = 1

SUBCASE 2
LABEL = Fx
LOAD = 2
SPC = 1

SUBCASE 3
LABEL = Fy
LOAD = 3
SPC = 1
    
```


**Header rsp-file:**

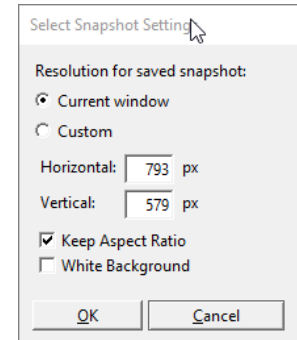
Input:	CNo.	PNr.	Channel description
		<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
	1		Fx
	2		Fy
	3		Fz



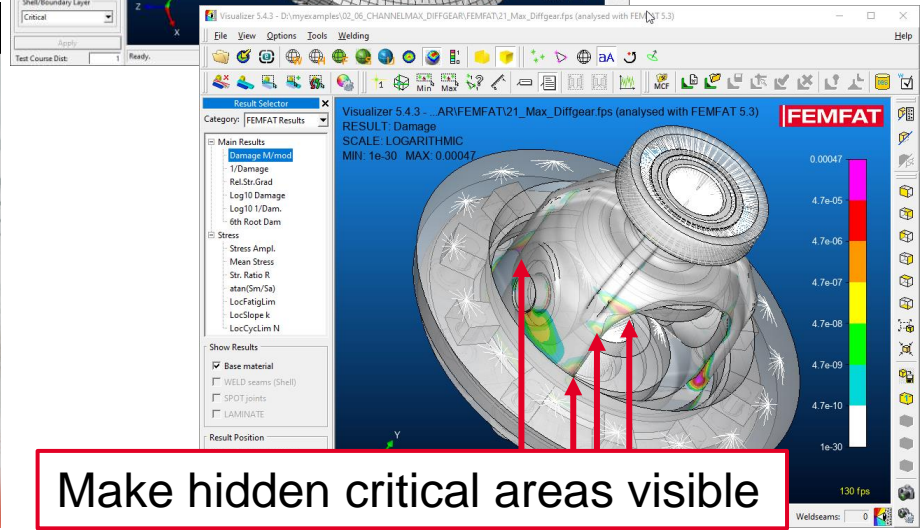
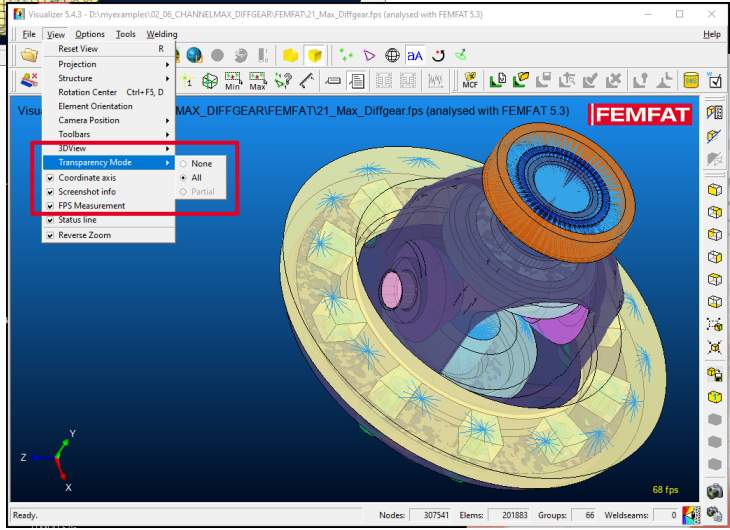
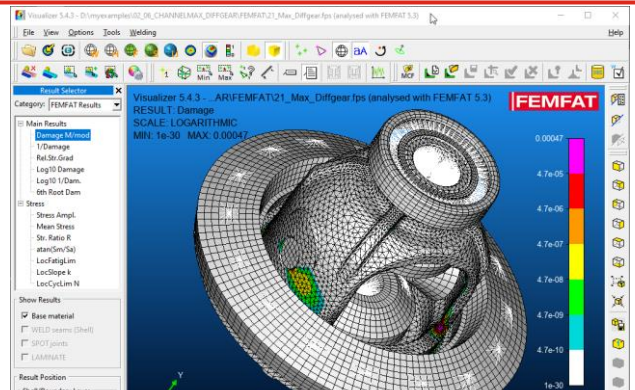
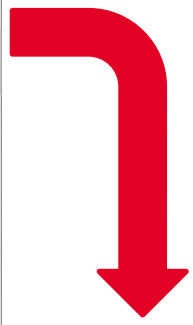
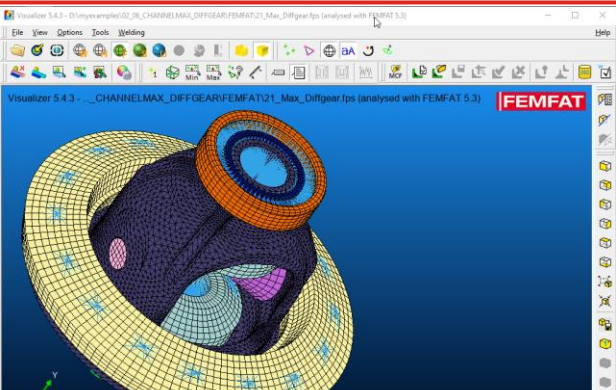
# VISUALIZER: New Graphic Kernel and Enhanced Post Processing Possibilities



- New graphic kernel from  **ceetron**  
understanding by visualization
  - 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

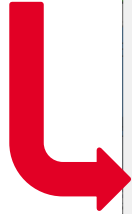


# VISUALIZER: Transparent Mode for Hidden Parts



Make hidden critical areas visible

# VISUALIZER: Transparent Mode for Hidden Parts



Preferences

Structure

- Mesh
- Group Nodes  5
- Element Nodes  5
- Min/Max Nodes  10
- Header Text  16
- FE-Entities Text  16
- Subwindow Text  16
- Colorbar Text  14
- Cross-hair
- Pointer line
- Uniform structure

Transparency [%]

Edge detection: alpha <

Feature lines angle [deg]

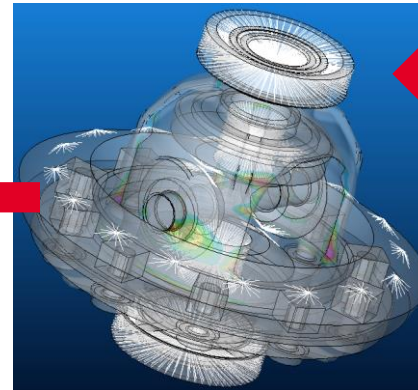
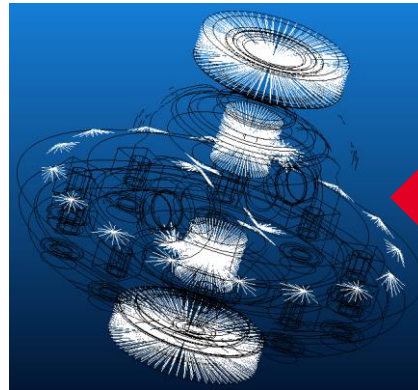
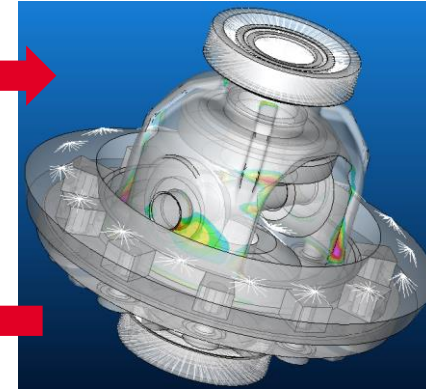
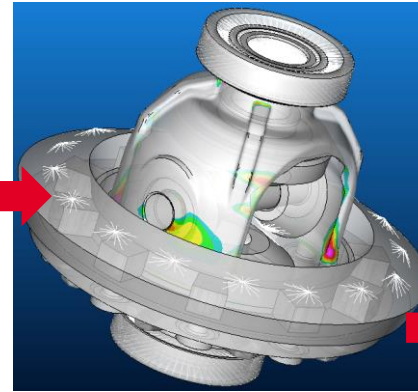
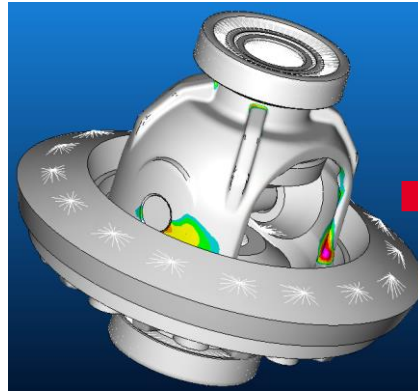
Rotation Sensitivity [%]

Background

Linear

Gradient

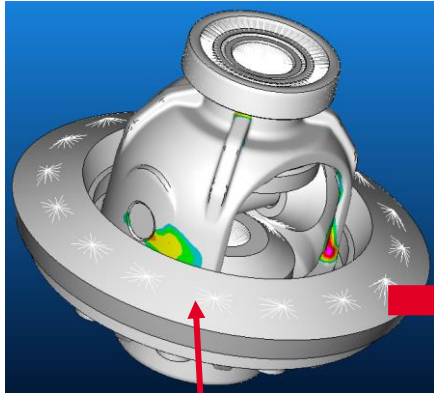
Apply Close



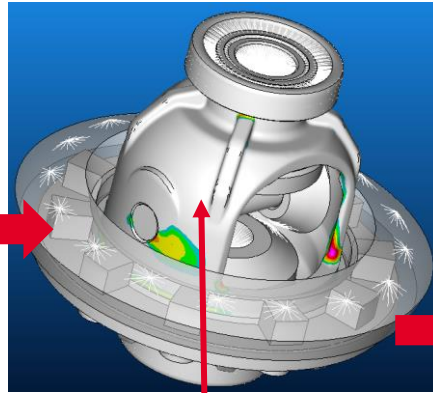
# VISUALIZER: Transparent Mode for Hidden Parts



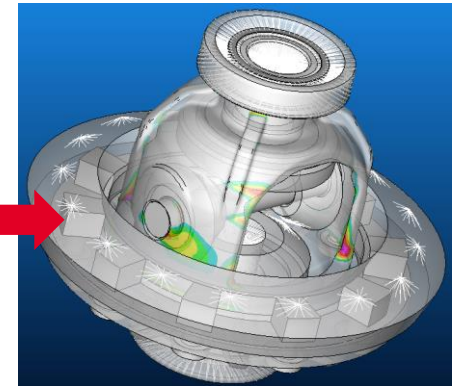
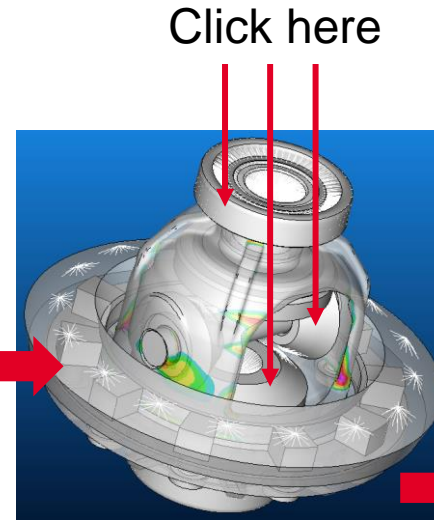
+ Shift: Make elements of a PID transparent



Click here



Click here



# VISUALIZER: Edge Detection and Feature Lines



Preferences

Structure

- Mesh
- Group Nodes  5
- Element Nodes  5
- Min/Max Nodes  10
- Header Text  16
- FE-Entities Text  16
- Subwindow Text  16
- Colorbar Text  14
- Cross-hair
- Pointer line
- Uniform structure

Transparency [%]

40

Edge detection: alpha <  30

Feature lines angle [deg]  30

Rotation Sensitivity [%]

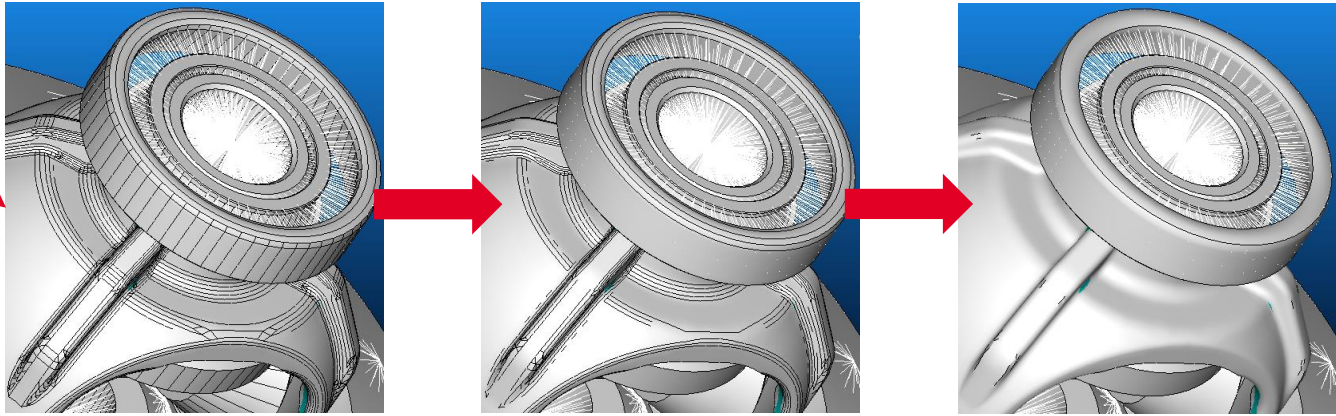
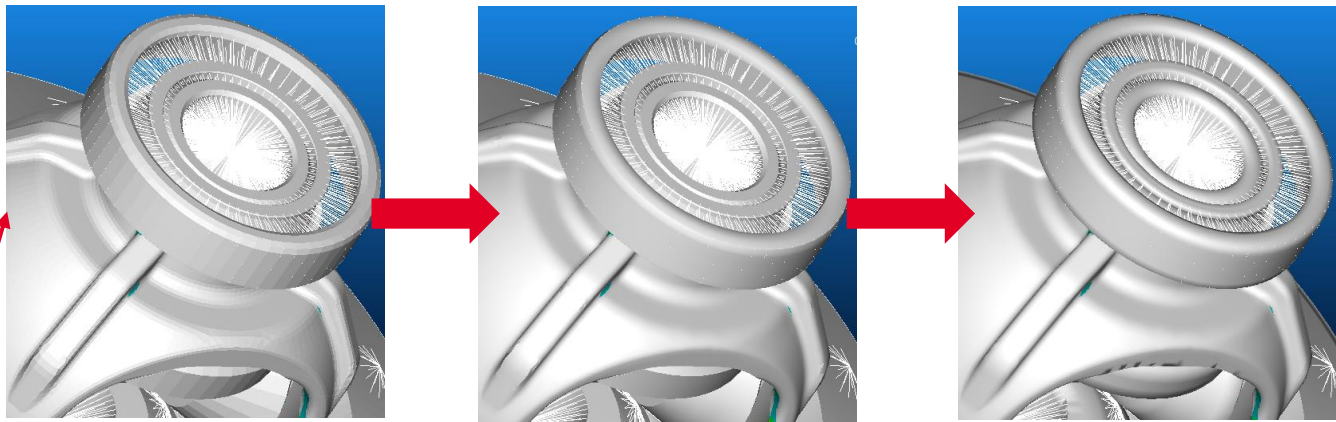
100

Background

Linear

Gradient

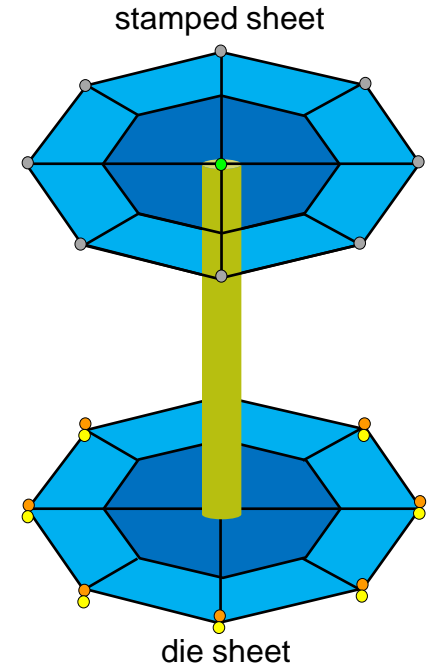
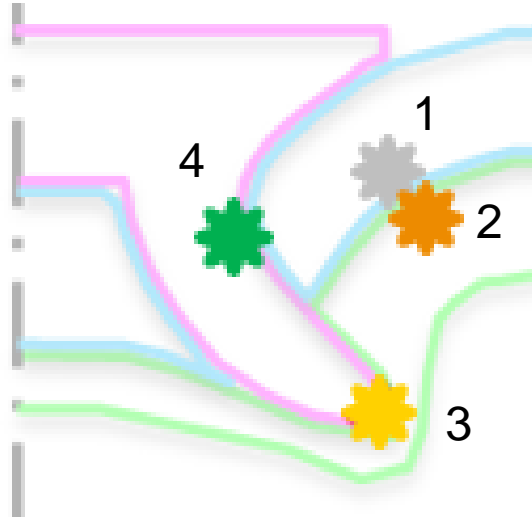
Apply Close



# SPOT: New advanced Self Piercing Rivet Model

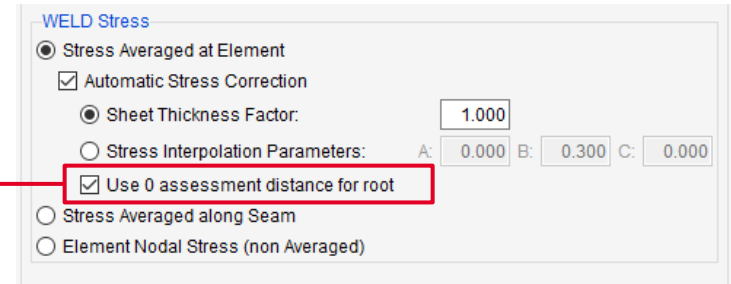
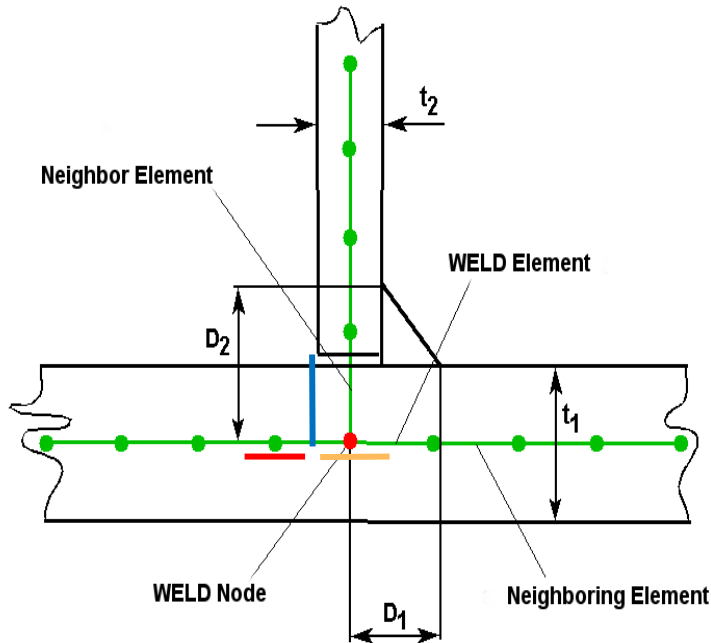


- Assessment of sheet failure and rivet failure in one analysis run.
- Assessment of stresses from nugget model and forces/moments from connecting beam element
- Stress based assessment of stamped sheet (failure position 1)
- Stress based assessment of die sheet (failure positions 2 and 3)
- Force based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

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

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

- $n_\sigma, \dots$   $K_t$ - $K_f$  ratio, Chapter 4.3.2,  
 $\tilde{K}_f$  constant, Table 4.3.1,  
if no better estimate is available,  
 $K_{R,\sigma}, \dots$  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

- Tempering Influence (for Tempering Steel only)
- Surface Residual Stresses
- Boundary Layer
- Local Material Properties

Combination Method Influence Factors

- FEMFAT 5.1
- FEMFAT 2.0
- FKM-Guideline**
- FEMFAT 5.1
- FEMFAT 2021

## Node Characteristics

Temperature Process Influence WELD Local Material Properties Misc.

Notches

Fatigue Notch Factor Kf (FKM):



# BREAK for STRAIN Calc

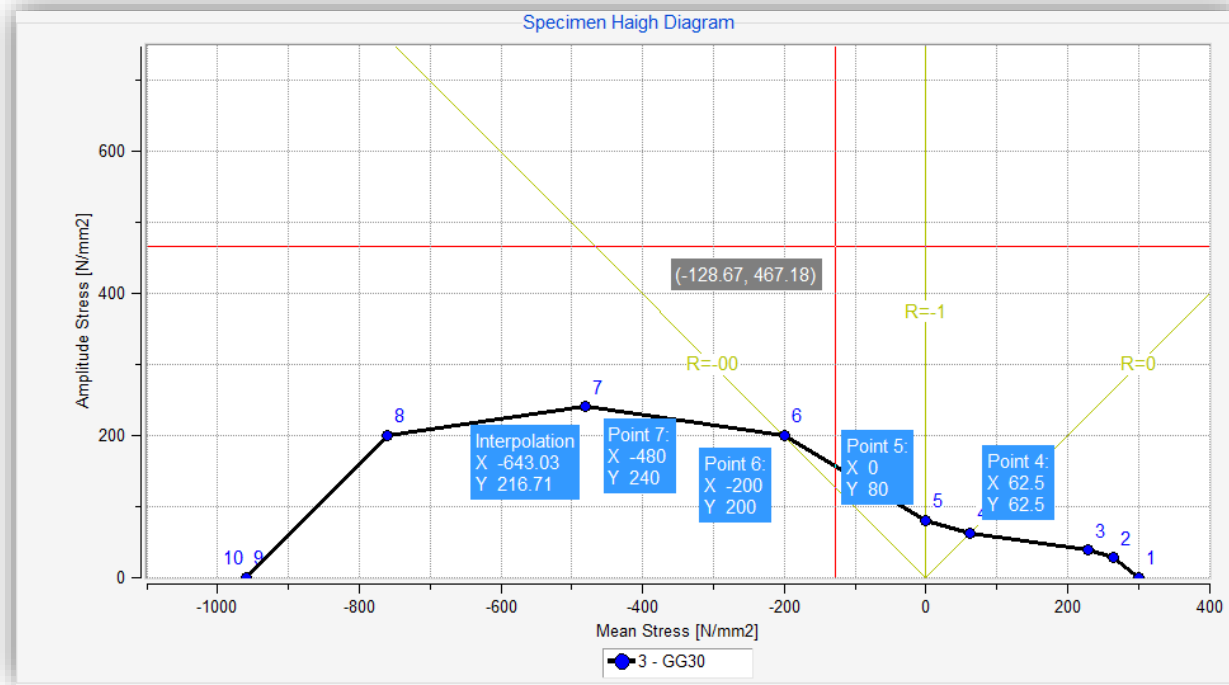


- Static safety factor analysis based on measured strains

The screenshot shows the FEMFAT 5.4.3 software interface. The 'Analysis Parameters' tab is active, and the 'Static Safety Factor' option is selected, with the word 'BREAK' in red text next to it. The analysis target is set to 'FEMFAT 5.0' and the criterion is 'Ultimate Strength'. Other parameters include 'MINER Modified' for damage, 'Sig\_m = const.' for endurance safety factor, and '90.000000 [%]' for survival probability. The 'Rainflow Counting' section is also visible, with 'Number of Rainflow Classes' set to 64 and 'Rainflow Counting Method' set to 'FEMFAT 5.1'. The 'Result Group' section shows 'Most Critical Gages Group' and 'Number of Nodes' set to 10. The 'Results Visualization at...' section has 'Most Critical Gage' selected.

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

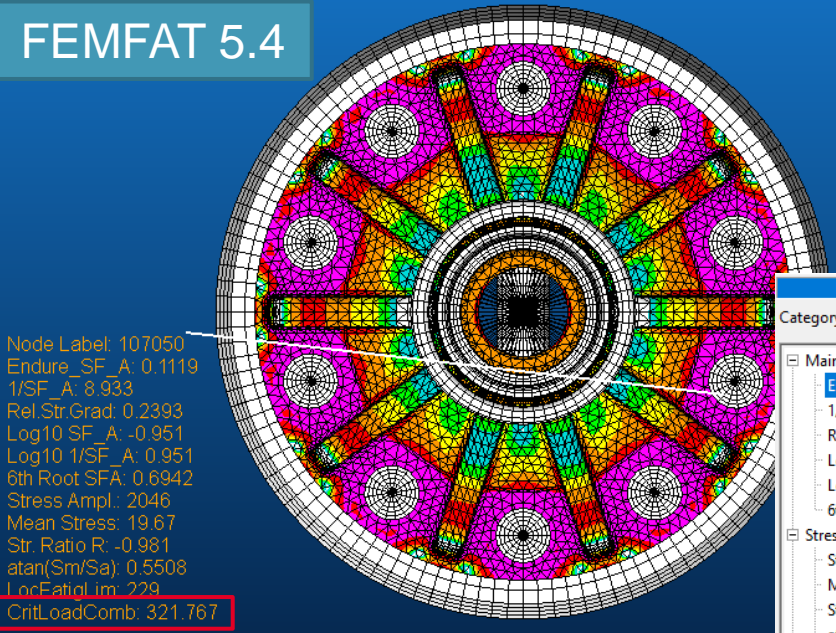
- Display Exact Point Coordinates in the S-N and Haigh Diagram by catching points
- Marker can be fixed
- Position of marker can be changed



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



## FEMFAT 5.4



Node Label: 107050  
 Endure\_SF\_A: 0.1119  
 1/SF\_A: 8.933  
 Rel.Str.Grad: 0.2393  
 Log10 SF\_A: -0.951  
 Log10 1/SF\_A: 0.951  
 6th Root SFA: 0.6942  
 Stress Ampl.: 2046  
 Mean Stress: 19.67  
 Str. Ratio R: -0.981  
 atan(Sm/Sa): 0.5508  
 LocFatigLim: 229  
**CritLoadComb: 321.767**

Only the leading 3 digits are displayed

Result Selector

Category: FEMFAT Results

- Main Results
  - Endure\_SF\_A
  - 1/SF\_A
  - Rel.Str.Grad
  - Log10 SF\_A
  - Log10 1/SF\_A
  - 6th Root SFA
- Stress
  - Stress Ampl.
  - Mean Stress
  - Str. Ratio R
  - atan(Sm/Sa)
  - LocFatigLim
- Miscellaneous
  - Time Step Min
  - Time Step Max

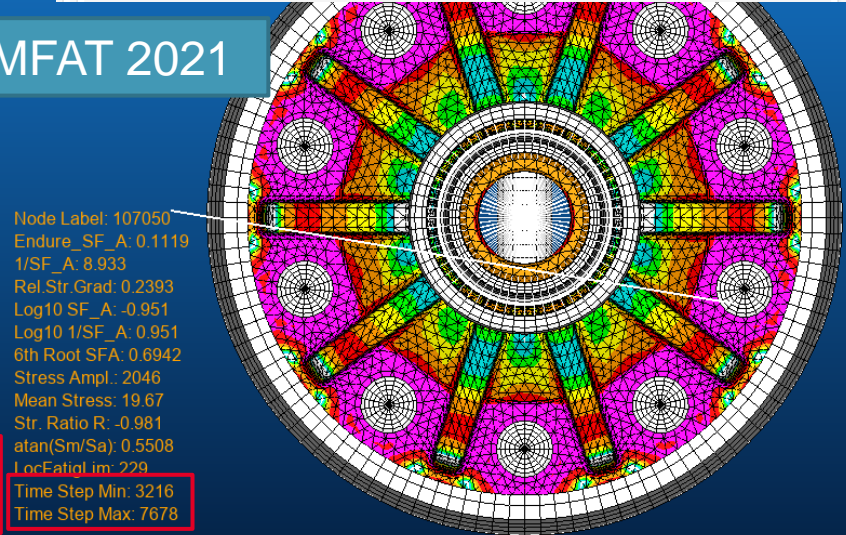
Separate output of time for max. and min. stress

FPS Setting DMA-Column Setting Result Modification Module Specific

Number of Scratch Results: 14 Default  Deselect All

Main Results (7)	Stress (5)	General Factors1 (0)	General Factors2 (0)	Surface (0)	Misc. (2)	Node Charact. (0)
<input checked="" type="checkbox"/> Miscellaneous <ul style="list-style-type: none"> <li><input type="checkbox"/> Normal Vector of Critical Cutting Plane (X,Y,Z)</li> <li><input type="checkbox"/> Secondary Dendrite Arm Spacing SDAS</li> <li><input checked="" type="checkbox"/> Time Step with Minimum Stress for Safety Analysis</li> <li><input checked="" type="checkbox"/> Time Step with Maximum Stress for Safety Analysis</li> <li><input type="checkbox"/> SPOT Force based: Critical Vector (X,Y,Z)</li> </ul>						

## FEMFAT 2021



Node Label: 107050  
 Endure\_SF\_A: 0.1119  
 1/SF\_A: 8.933  
 Rel.Str.Grad: 0.2393  
 Log10 SF\_A: -0.951  
 Log10 1/SF\_A: 0.951  
 6th Root SFA: 0.6942  
 Stress Ampl.: 2046  
 Mean Stress: 19.67  
 Str. Ratio R: -0.981  
 atan(Sm/Sa): 0.5508  
 LocFatigLim: 229  
**Time Step Min: 3216**  
**Time Step Max: 7678**

# 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

=====

The screenshot shows a software settings window with a sidebar on the left and a main panel on the right. The sidebar contains icons for Node Characteristics, Influence Factors, Strain Gage Data, Analysis Parameters, Output, Report, Analyze, and Visualization. The 'Report' option is selected. The main panel is titled 'Report Items' and contains several sections of checkboxes:

- General Input Data:**
  - Header
  - General Input Data
  - Specimen Material Data
  - Influence Factors
- Structural Node Data:**
  - Damage Data/Safety Factors:**
    - Top
    - Bottom
  - Max Damage Component:**
    - Top
    - Bottom
  - Stress Gradient:**
    - Top
    - Bottom
  - Mean Stress Rearrangement:**
    - Top
    - Bottom
  - Surface Roughness
  - Technological Size
  - Tempering Condition
  - Temperature
  - Range of Dispersion (10% to 90%)
  - Fatigue Notch Factor Kf (FKM)
- WELD Specific Output:**
  - Local Direction Specific Information
  - Notch Factor Specific Information
  - WELD Result Table
  - SSZ/MSZ-Method (SSZ-DetailedOutput.txt)
- SPOT Specific Output:**

# 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.

The screenshot shows the 'Result Modification' tab in the software. It features a table of selected output results and a list of checkboxes for including specific results in the output.

Col	Result	Group
1	Damage incl. SPOT - Detailed Result	Main Results
2	Inverse(Damage)	Main Results
3	Stress Amplitude	Stress
4	Mean Stress	Stress
5	arctan(Mean Stress/Amplitude Stress)	Stress
6	SPOT - Critical Nugget Result	Main Results

Below the table, there are tabs for 'Main Results', 'Stress', 'General Factors1', 'General Factors2', 'Surface', 'Misc.', and 'Node Charact.'. Under the 'Main Results' tab, a list of results is shown with checkboxes and column selection boxes:

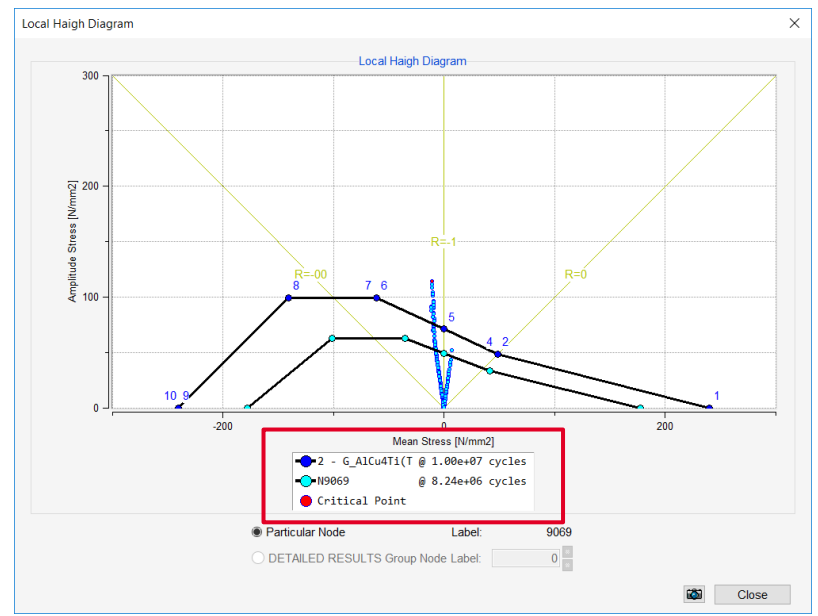
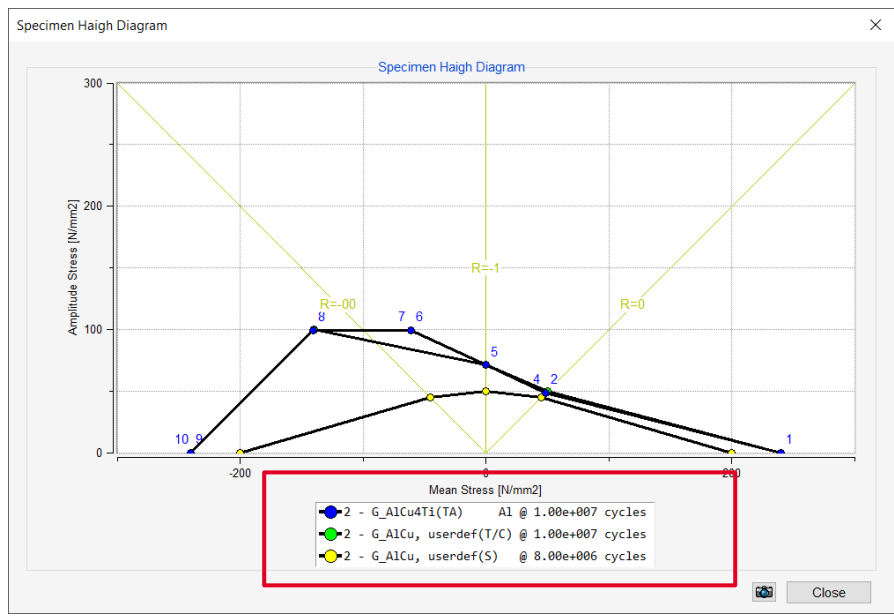
- Damage:  Include SPOT - Detailed Result
- Safety Factor B:
- Inverse(Damage):
- Inverse(Safety Factor B):
- Relative Stress Gradient:
- SPOT - Critical Nugget Result:
- Log10(Damage):
- Log10(Inverse Damage):
- 6th Root(Damage):

# 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.**