



Forward. For all.

FEMFAT 2025 News and More

Christian Gaier, Manager FEMFAT Development

2025-05-14



A Short History of FEMFAT Releases

- A really very long time ago:
FEMFAT 1.0 ~ 1990
- ... (time passing by) ...
- **FEMFAT 2023** released April 2023,
presented at ECS Simulation
Conference 2023
- **FEMFAT 2024a** and **2023a** released
May 2024
- **FEMFAT 2024.1** and **2024b** released
November 2024
- **FEMFAT 2025** will be released in
June 2025

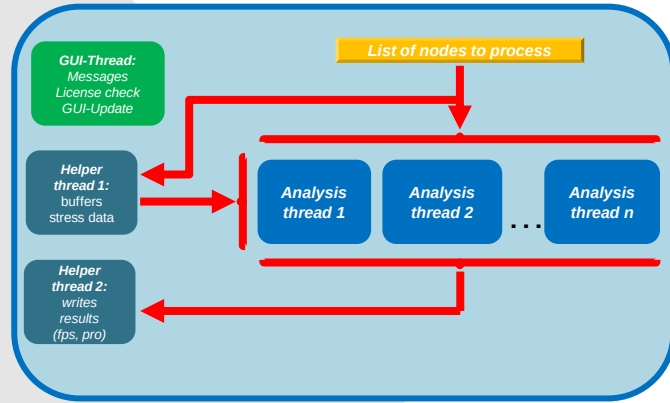




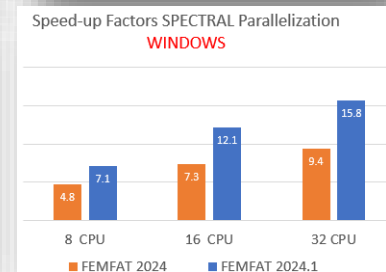
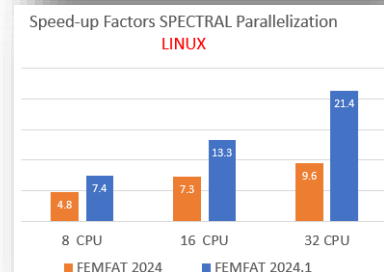
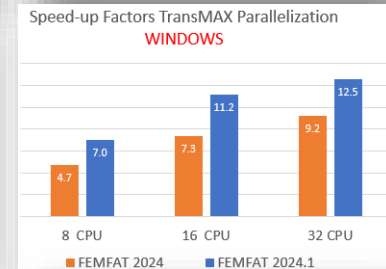
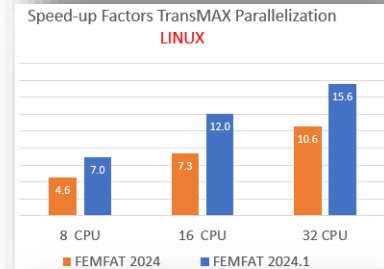
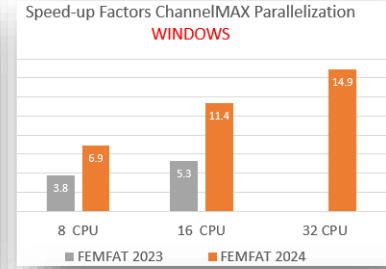
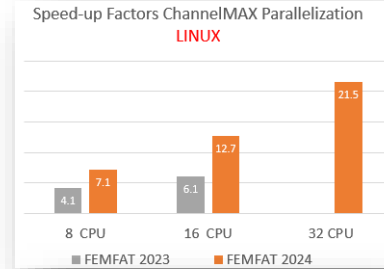
01

FEMFAT Highlights

New Thread-based Parallelization for ChannelMAX, TransMAX and SPECTRAL

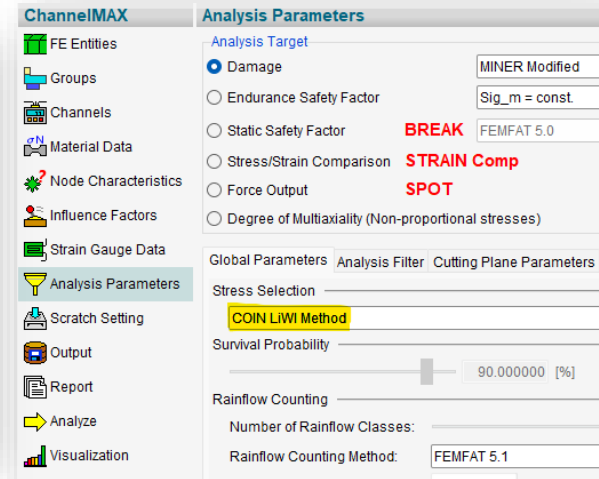
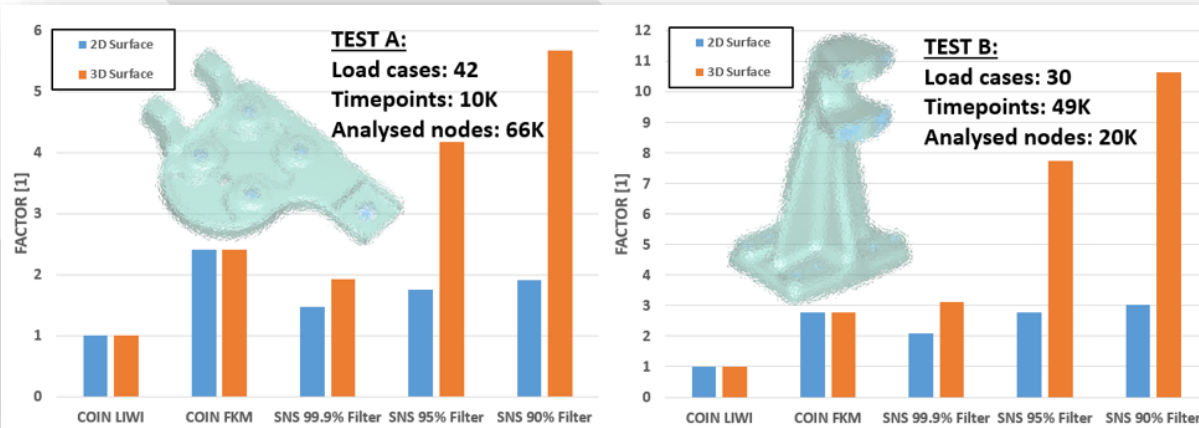


- Much faster than old parallelization based on group segmentation
- Much less memory needed → Works for 32 CPUs and even more



A Brand-New Multiaxial Fatigue Theory in MAX: COIN LiWI – Fast and Reliable

- Based on complex invariants (Ref.: Alexander Schmidt, Int. Journal of Fatigue 168, 2023, 107474)
- Equivalent stress history without non-physical discontinuities
- **No search of critical plane → very fast !!!**
- Implicit consideration of phase difference between bending / torsion load
- Also, equivalent stress according FKM-Guideline included (FKM LiWI)



Many New Material Classes in Material Generator

Covering a wide range of technical materials, no material tests needed!

The image displays four screenshots of the Material Generator software interface, illustrating the range of material classes available. Red arrows indicate the flow of selection across the different screenshots.

Screenshot 1 (Top Left): Shows the 'Define Material Parameters' dialog. The 'Material Group' is set to 'Iron'. The 'Material Class' dropdown is open, showing options: Aluminum, Magnesium, Other Metal Alloys, and Polymer. A red arrow points from this dropdown to the 'Material Class' dropdown in Screenshot 2. A red box labeled 'FEMFAT 2023' points to the 'Material Parameter' dropdown.

Screenshot 2 (Middle Left): Shows the 'Define Material Parameters' dialog. The 'Material Group' is set to 'Magnesium'. The 'Material Class' dropdown is open, showing options: Magnesium Wrought Alloys and Magnesium Casting Alloys. A red arrow points from this dropdown to the 'Material Class' dropdown in Screenshot 3.

Screenshot 3 (Bottom Left): Shows the 'Define Material Parameters' dialog. The 'Material Group' is set to 'Other Metal Alloys'. The 'Material Class' dropdown is open, showing options: Titanium Base Alloys and Titanium Casting Alloys. A red arrow points from this dropdown to the 'Material Class' dropdown in Screenshot 4.

Screenshot 4 (Top Right): Shows the 'Define Material Parameters' dialog. The 'Material Group' is set to 'Iron'. The 'Material Class' dropdown is open, showing a list of material classes. A red box highlights the following options: General Structural Carbon Steels, Higher Strength Weldable Structural Carbon Steels, Highest Strength Steels - Conventional, Highest Strength Steels - ESR Process, Highest Strength Steels - VAR Process, Fine Grained Steels, Stainless Steel - Austenitic, Stainless Steel - Martensitic, Stainless Steel - Ferritic, Heat Treatable (Tempered) Steels, Nitriding Steels, Forging Steel - Quenched and Tempered, Forging Steel - Normalized, Case Hardening Steels, Micro-alloyed Steels, Grey Cast Irons, Compacted Graphite Irons CGI, Nodular Cast Irons, Ausferritic Ductile Iron with Nodular Graphite ADI, and Malleable Cast Irons.

Legend:

- ESR = Electro-slag remelting process
- VAR = Vacuum-arc remelting process

SPOT Force-Based Approach in SPECTRAL

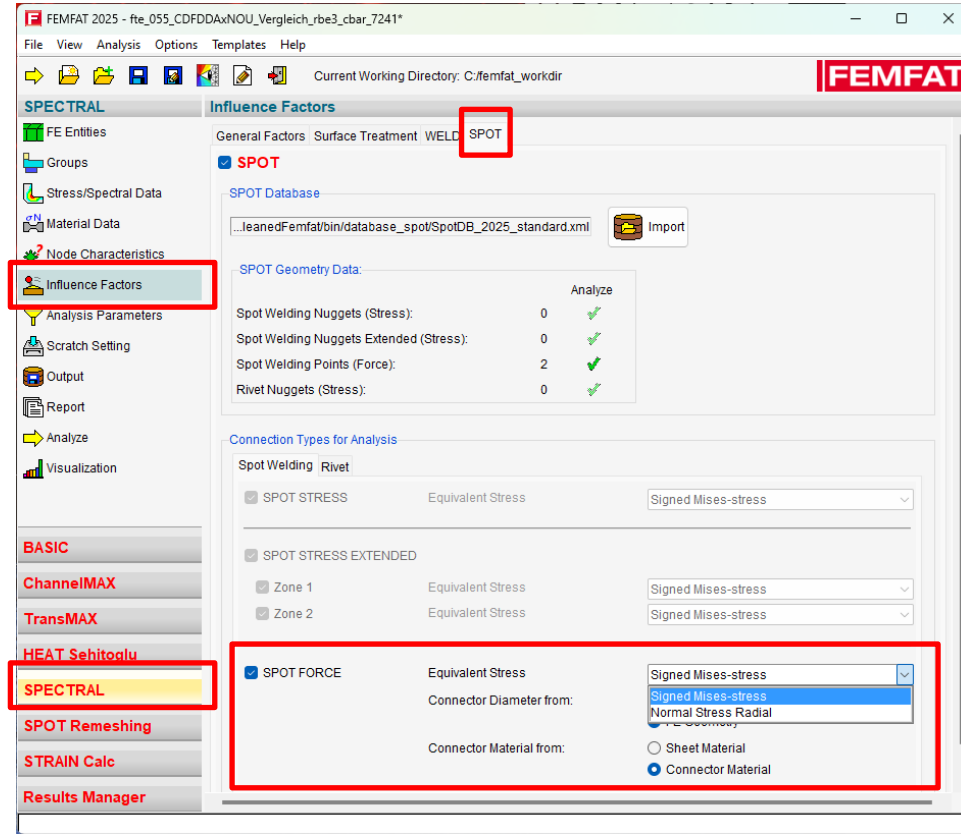
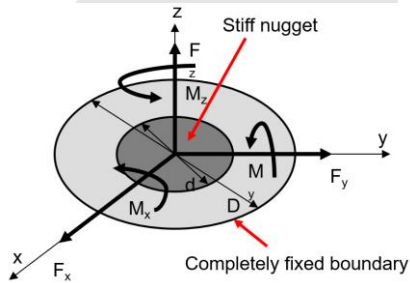
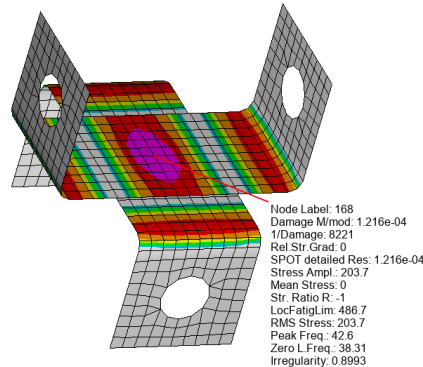
- A force-based SPOT assessment is possible in SPECTRAL analogous to BASIC and MAX
- Analytical SPOT joint model acc. to JSAE
- Available for all probability models

Supported interfaces:

- Nastran OP2
- Nastran H5
- Abaqus ODB
- Medina BOF

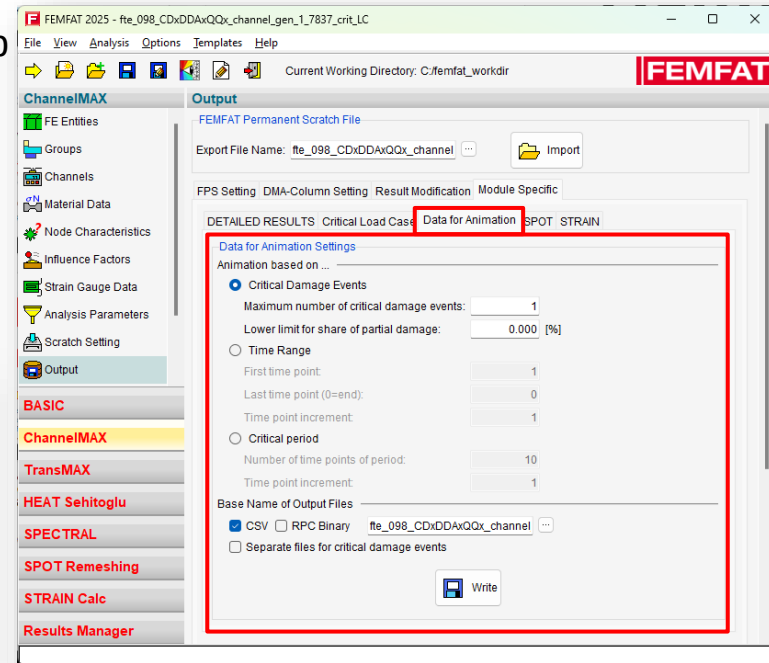
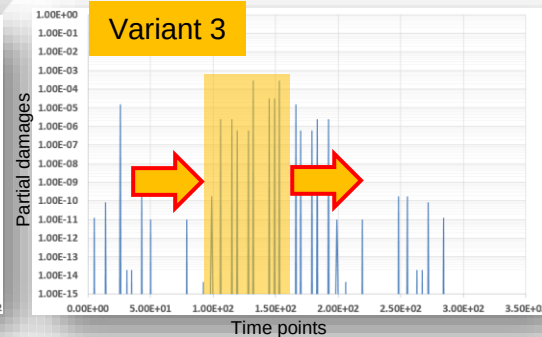
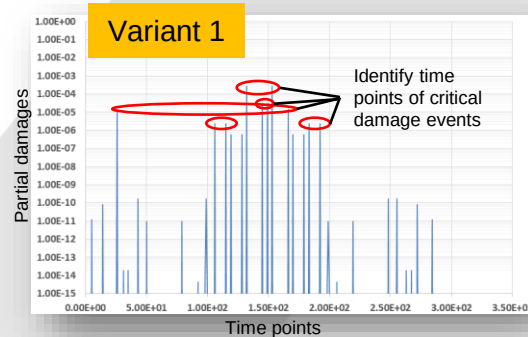
Equivalent stresses:

- Signed Mises-Stress
- Radial stress



Generation of Channel Histories of Critical Damage Events for Deformation Animation

- Three variants are provided:
 1. Searching for critical damage events
 2. User defined time range
 3. Searching for a critical period
- For variant 1 & 3 output for all nodes of Detailed Results group
- Output of critical channel histories as:
 - CSV file
 - RPC binary file
- Load superposition and animation possible in:
 - ANSA/META
 - Animator
 - HyperView (with Python commands)





02

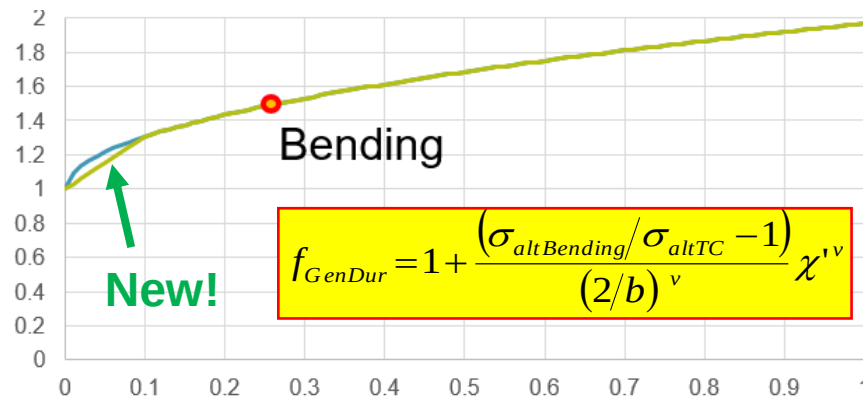
Methodological Improvements

Improved Stress Gradient Influence Method

- Method **Eichlseder** is FEMFAT standard
- **Advantage:** Exact prediction of bending fatigue
- **Disadvantage:** Infinite slope of the notch support curve at gradient = 0 makes difficulties for fitting to tests results because very sensitive!
- **New:** Linear function for $0 < \text{gradient} < 0.1$
- **New default method** in FEMFAT 2025

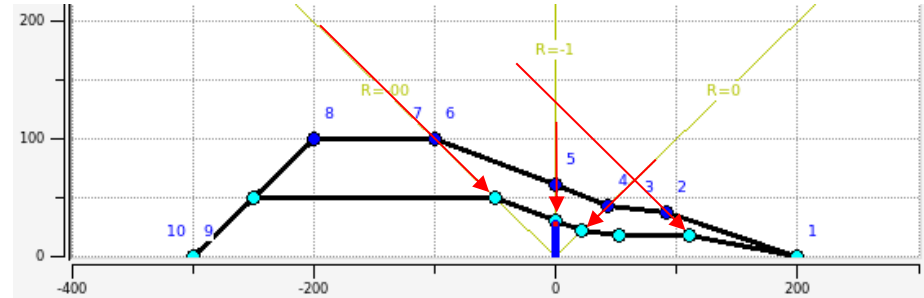
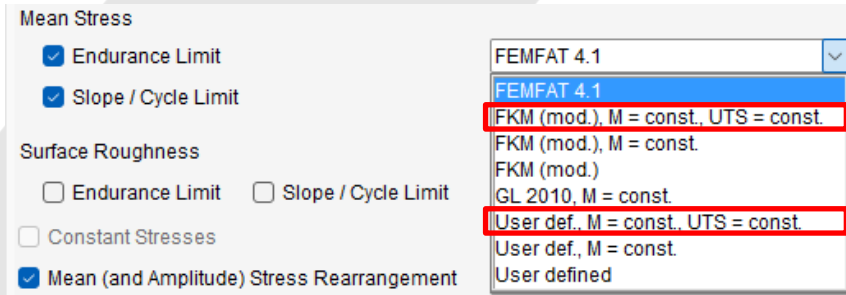


Attention: Results will change compared to previous versions at nodes with gradient < 0.1 !



Mean Stress Influence According to FKM-Guideline

- Haigh-diagram construction according to FKM-Guideline between point 2 and 8 → for the material Haigh-diagram as well as for the local component Haigh-diagram at nodes.
- Influences (Gradient, temperature, etc.) do not change mean stress sensitivity.
- In contrast to FKM, where Haigh-diagram is limited by YS multiplied with the plastic notch factor, the FEMFAT Haigh-diagram is limited by UTS multiplied with the “Modified Haigh Diagram” influence factor (point 1 and 10) → Less conservative results.



Separated Influence on Slope / Cycle Limit for Surface Roughness and Forge Influence

- When surface roughness and forge influence should only be considered for the endurance limit, but not for the slope and cycle limit (e.g., for the fatigue assessment according to the FKM-Guideline), it can be deactivated separately.

Influence Factors

General Factors Surface Treatment WELD SPOT

Stress Gradient

Gradient Computation Method

☒ Endurance Limit ☒ Slope / Cycle Limit

Mean Stress

☒ Endurance Limit

☒ Slope / Cycle Limit

Surface Roughness

☐ Endurance Limit ☐ Slope / Cycle Limit

Influence Factors

General Factors **Surface Treatment** WELD SPOT

☐ Shot Peen

☐ Cold Rolling

☐ Carburize

☐ Nitride

☐ Inductive Hardening

☐ Flame Hardening

Forge

☐ Endurance Limit ☐ Slope / Cycle Limit

☐ General Surface Treatment Factor

Slope of the local component S-N curve

$$k_C = \left(\frac{k_M - IFK2}{(f_{GR,sf})^{IFK3} - 1 + \frac{1}{(f_{1,af})^{IFK3}} + IFK2} \right) \cdot \frac{1}{f_{m,sf}}$$

$$f_{1,af} = 1 - \text{sign}(1 - f_{SR,af}) \sqrt{(1 - f_{SR,af})^2 + (1 - f_{DF,af})^2}$$

Set $f_{SR,af} = 1$ (surface roughness)

Set $f_{DF,af} = 1$ (degree of forging)

Source: Synthetic S-N Curves 1983, 1999

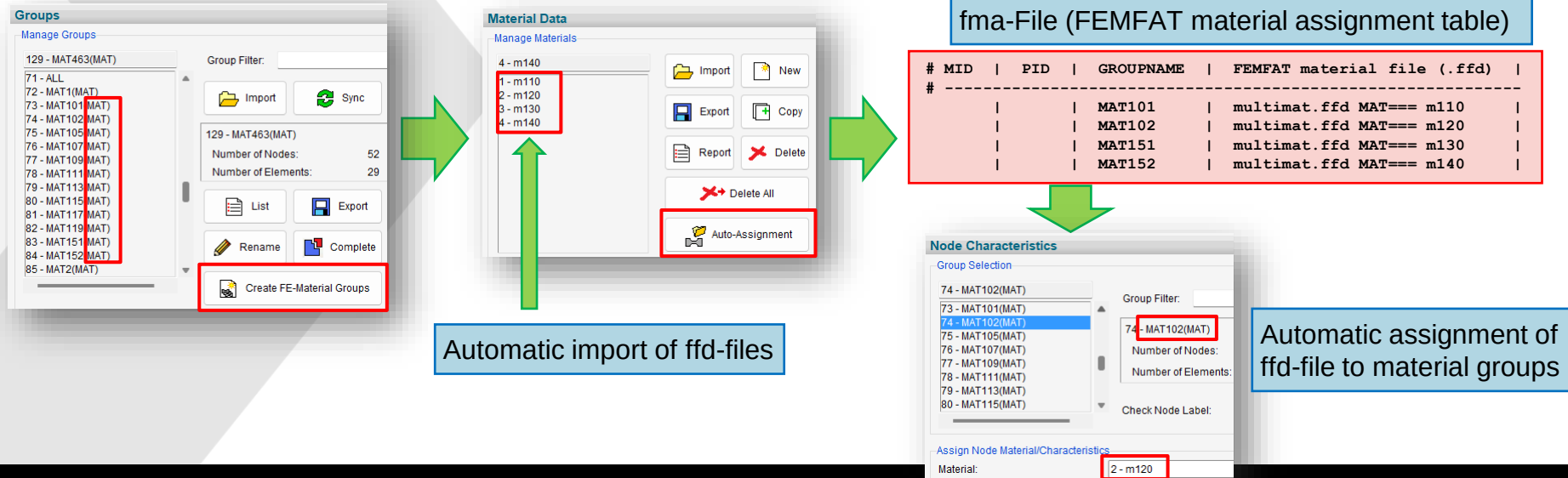


03

Automations and Best Practice

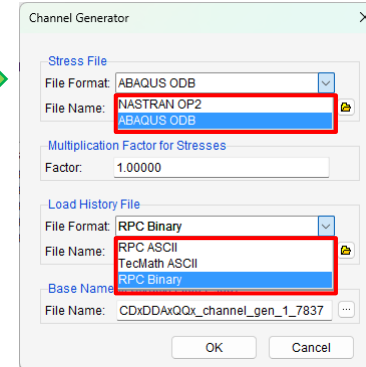
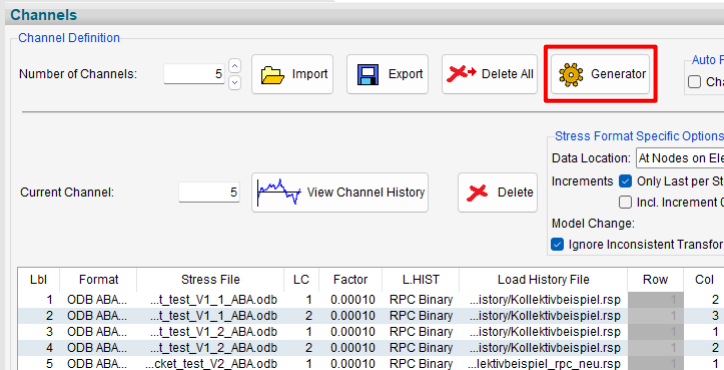
Automatic Material Assignment for ABAQUS

- Based on MIDs / PIDs ffd-files can be assigned to nodes automatically for every interface (Nastran, Ansys, etc.) except ABAQUS
- In ABAQUS materials are identified by names. Therefore, FEMFAT 2025 provides the new possibility to generate material groups, to which ffd-files can be assigned automatically!



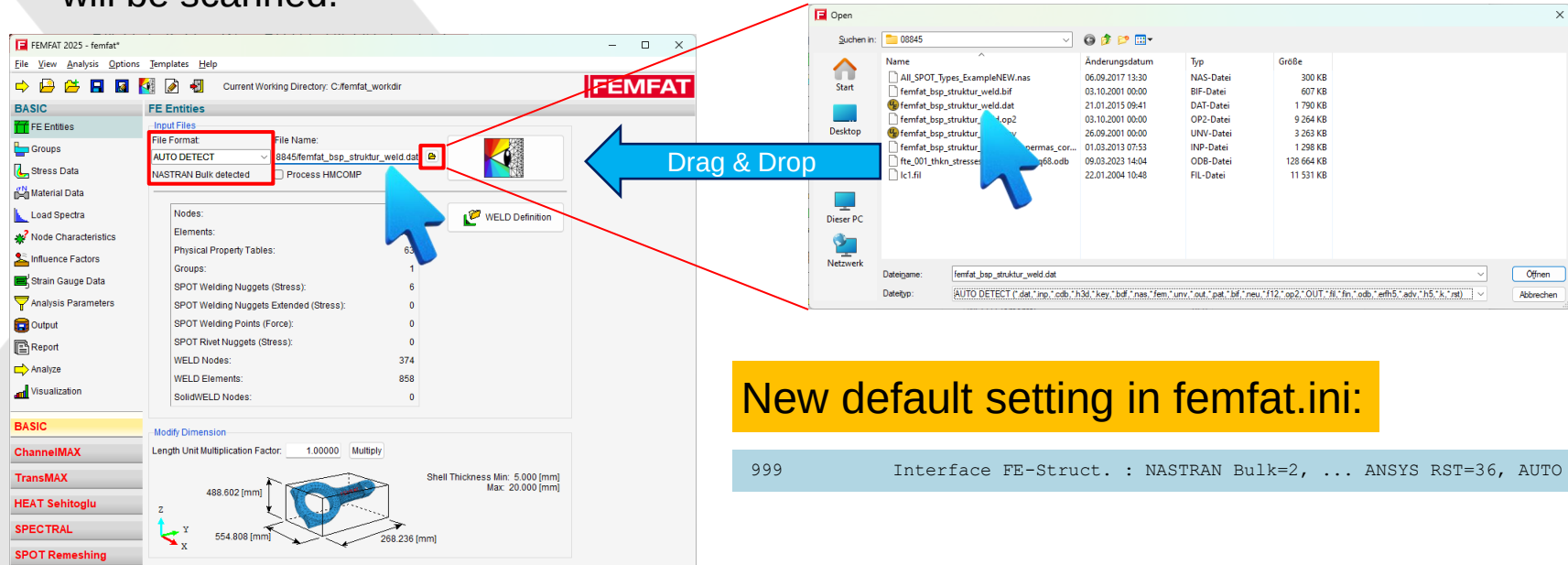
Automatic Channel Generator for ChannelMAX

- Automatic identification of unit stresses and related load histories by comparison of step names or load case names with channel names.
- Automatic channel generation
- Works for **Nastran OP2** and newly for **Abaqus ODB!**
- Works for **RPC Binary** and newly for **RPC ASCII** and **TecMath ASCII!**



Autodetect for FE Interfaces and Drag & Drop

- Detection is done by extension.
- If extension does not uniquely define the file format (i.e. Nastran.dat and Permas.dat), file will be scanned.



New default setting in femfat.ini:

999

Interface FE-Struct. : NASTRAN Bulk=2, ... ANSYS RST=36, AUTO DETECT=999

Identification of Local Extrema (Damage, etc.)

For Base Material, WELD and SPOT nodes, in FEMFAT and VISUALIZER

Create/Modify Group Entries

Nodes Nodes Based on... Elements

☐ Based on Damage Values / Safety Factors

from 0.000e+00 to 0.000e+00

☐ only most critical node per SPOT nugget

☐ Based on Isothermal Nodal Temperature [°C]

from 20.00 to 20.00

☒ Based on Local Critical Areas

Damage / Safety Factor

from Minimum to Maximum

to Maximum

Relative limit for size of local area 50.0000 [%]

Number of relative extrema 10

Considered node types

☒ Base Material

☒ SPOT

☒ WELD

DETAILED RESULTS

New

Add

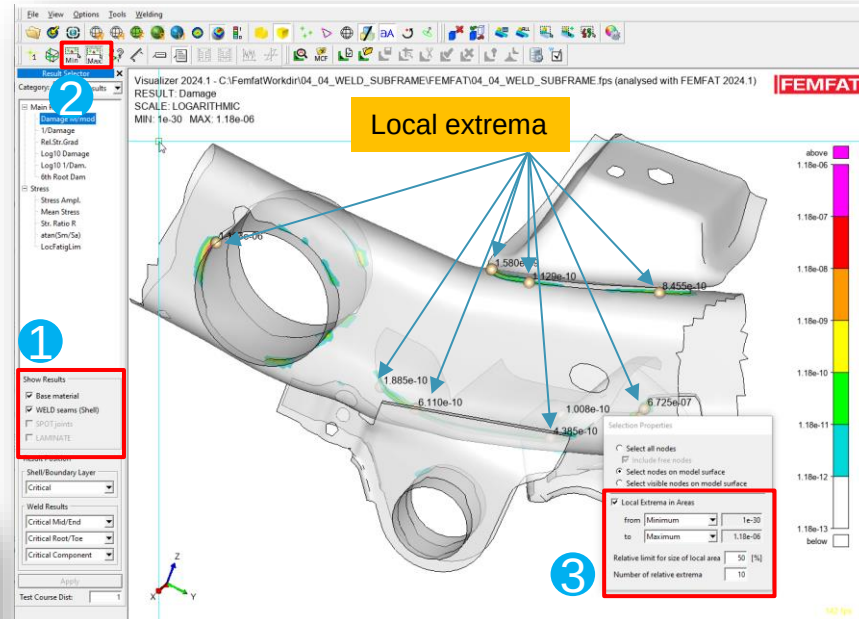
Remove

Report file

Critical Locations:

Location	NodeLab	Damage	Node Type	Result Pos.	Rel.Str.Grad	Stress Ampl.	Mean Stress	LocFatigLim
1	1835	2.826e+04	Base	top / surf	0.000	1213.692	0.000	310.000
2	1851	9.154e+02	Base	top / surf	0.000	888.562	0.000	310.000
3	1845	2.542e+01	Base	top / surf	0.000	641.487	0.000	310.000
4	12663	6.357e+00	SPOT	bottom	0.000	1566.713	0.000	418.776
5	12587	1.501e+00	SPOT	top	0.000	1152.507	0.000	418.776
6	12632	1.404e+00	SPOT	bottom	0.000	1136.195	0.000	418.776
7	10077	2.152e-03	WELD	EUEBOT	0.000	193.380	0.000	225.000
8	9108	2.100e-03	WELD	EWUBOT	0.000	192.820	0.000	225.000
9	9132	1.403e-03	WELD	EWUBOT	0.000	184.176	0.000	225.000
10	9130	1.141e-03	WELD	EUEBOT	0.000	179.992	0.000	225.000

End - Critical Locations

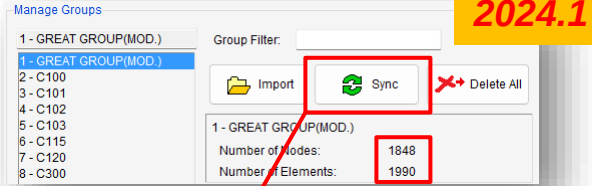
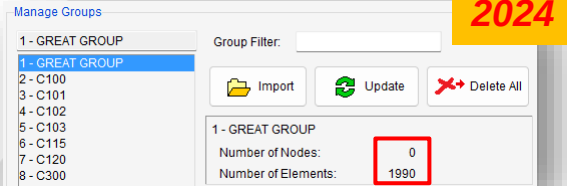
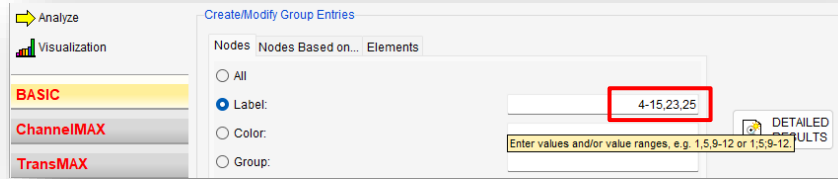


Local extrema search can be done for all result types

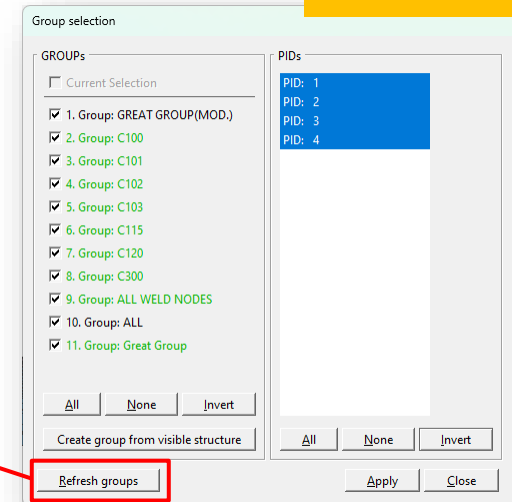
- FEMFAT Results (damage, stresses, fatigue limit, etc.)
- FEA-Stresses
- Animated Results
- Node Characteristics (temperature, SDAS, etc.)

Improved Group Handling

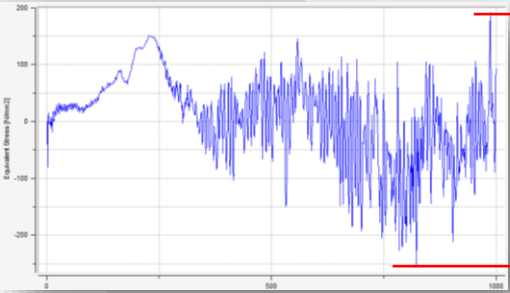
- Enter Label ranges and lists:
- Automatic completion of groups: Group of elements will be completed by adding the nodes automatically when importing the FE model.



- Synchronize FEMFAT and VISUALIZER:
 - FEMFAT: Clicking on the “Sync” button loads the groups created in the VISUALIZER and updates the fps-file.
 - VISUALIZER: Clicking on “Refresh groups” loads the groups created in FEMFAT.



ChannelMAX: Critical Load Case for Several Nodes



Max. amplitude
from equivalent
stress history

jobname CritLC Subcases.dat

```
SUBCASE = 1000001
SUBTITLE = Node 4602: stress at time 6195 (maximum equivalent stress)
LOAD = 1000001
$
SUBCASE = 1000002
SUBTITLE = Node 4602: stress at time 8549 (minimum equivalent stress)
LOAD = 1000002
$
SUBCASE = 1000003
SUBTITLE = Node 4602: amplitude stress between time 6195 and 8549
LOAD = 1000003
```

jobname CritLC Loads.dat

```
LOAD*      1000001 1.000000000000 0.044665 1+
*          0.102884 2 1.277838 3+
*          0.044665 4 1.277838 5
$ Node 4602: load factors for minimum equivalent stress at time 8549
LOAD*      1000002 1.000000000000 0.056763 1+
*          -0.447711 2 -1.715377 3+
*          0.056763 4 -1.715377 5
$ Node 4602: load factors for amplitude stress between time 6195 and 8549
LOAD*      1000003 1.000000000000 -6.048967E-03 1+
*          0.275297 2 1.496607 3+
*          -6.048967E-03 4 1.496608 5
```

Critical Load Case Settings

Node Output Selection

- ☒ Most Critical/Particular Node
- ☒ DETAILED RESULTS Group
- ☐ All Nodes in Group
- ☒ Selected Nodes in Group:

Load Case Number from ...

- ☒ File
- ☐ FEMFAT Load Case Number
- ☐ 1,2,3...

Load Case Definition

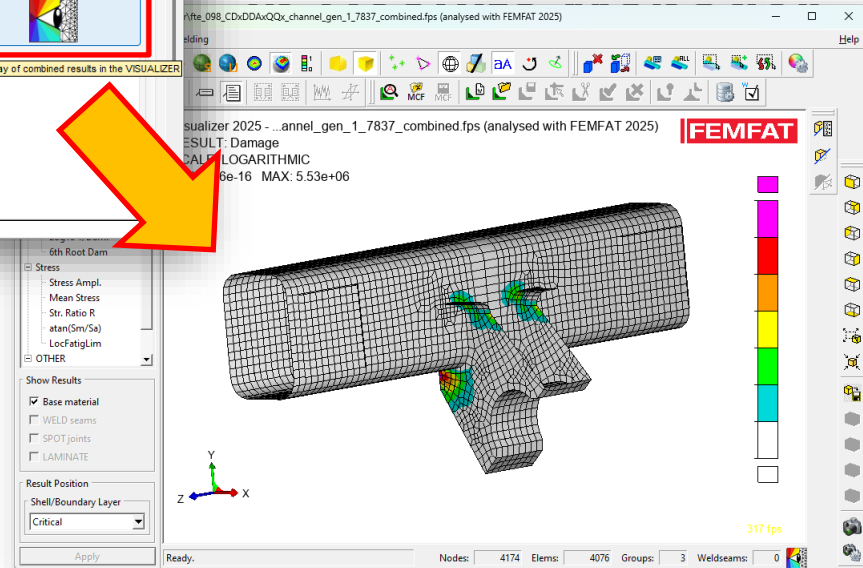
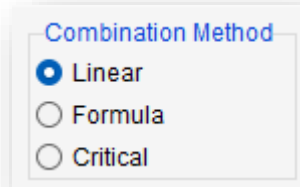
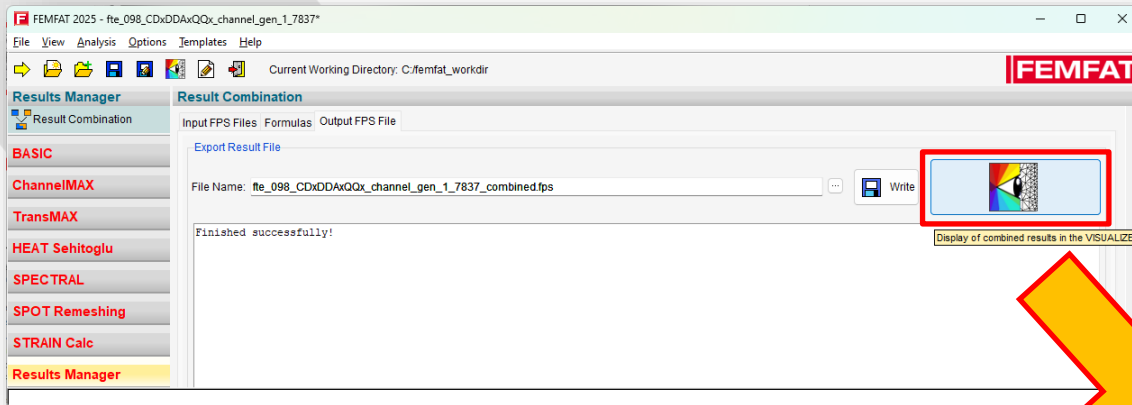
- ☒ Maximum Value
- ☒ Maximum Amplitude
- ☒ Minimum Value
- ☐ at Time Point

Base Name of Output Files

 Write

- Include these files in your Nastran dat-file with FE-structure
- Rerun Nastran
- Visualize deformation of critical load case (e.g. in Meta, HyperView, etc.)

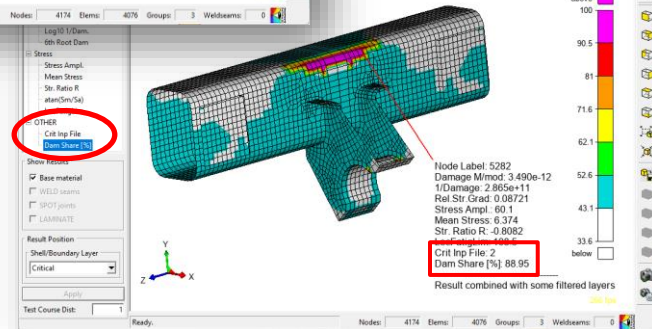
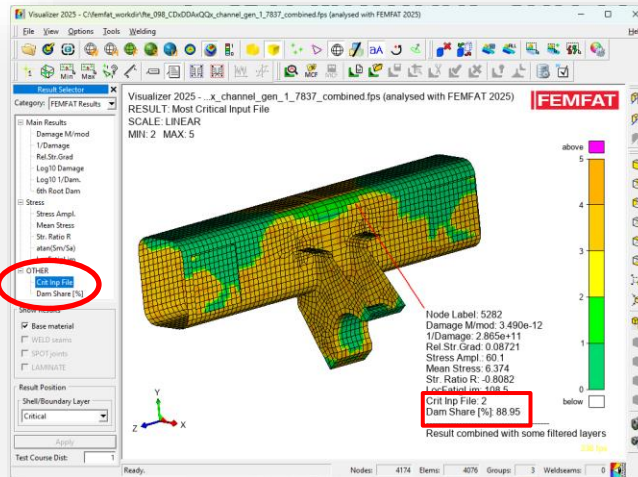
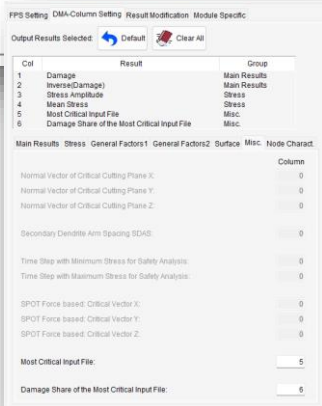
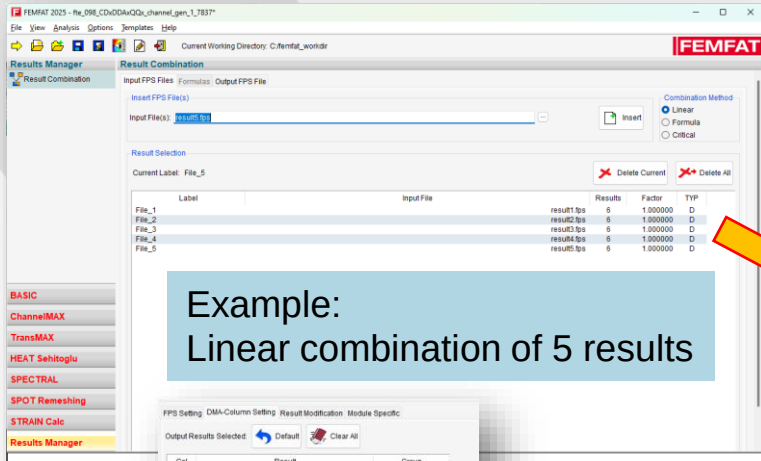
One Click Visualization of Combined Results in Results Manager



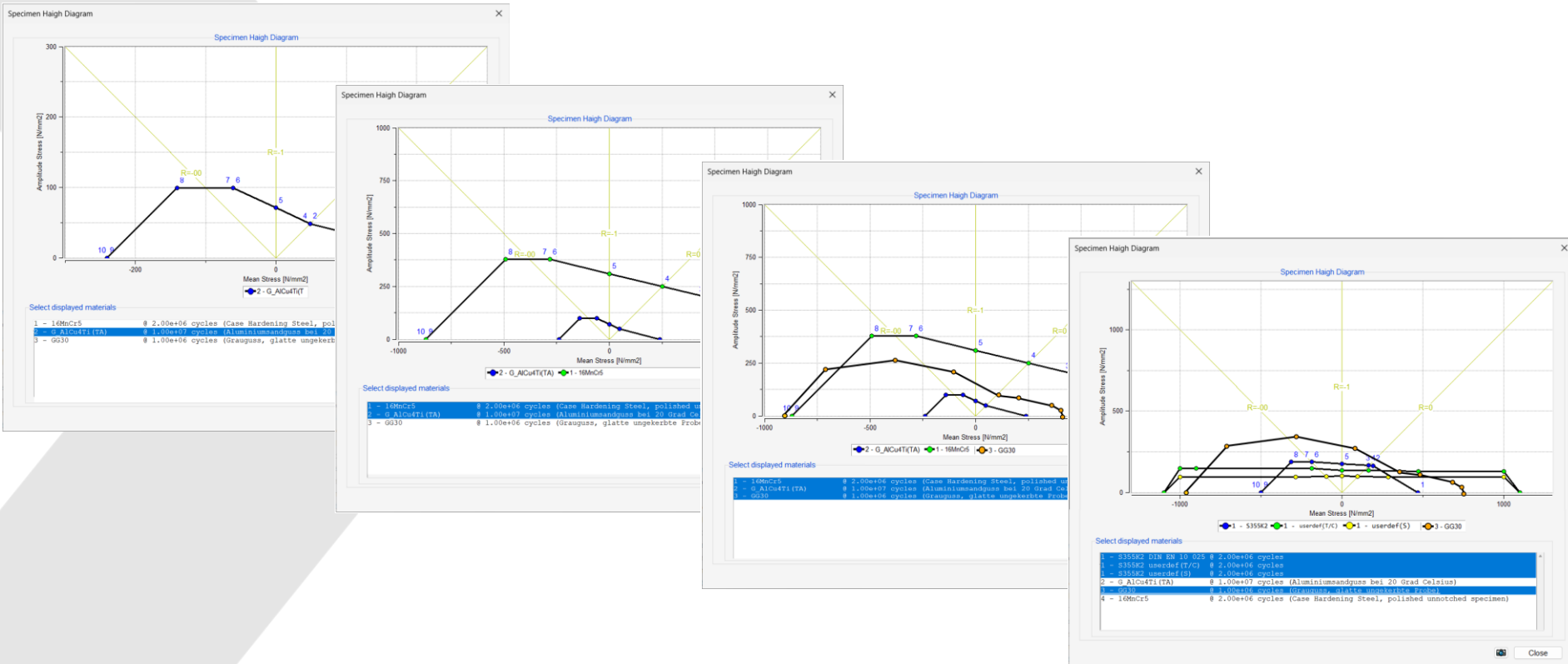
Results Manager, New Datasets: Critical Load Case Number – Damage Share [%]

Example:
Linear combination of 5 results

Output as
dma-File

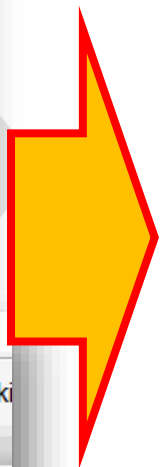
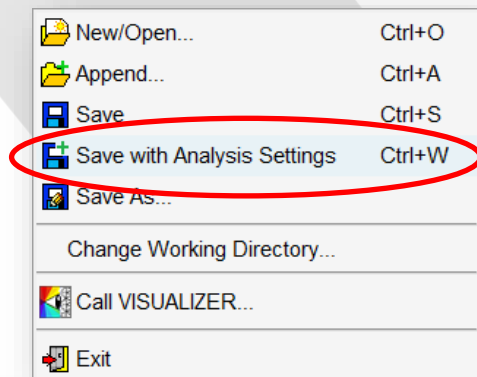


Display of Multiple Material Haigh Diagrams



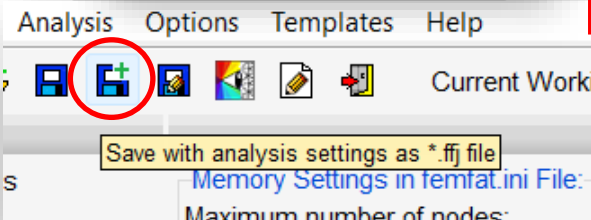
Summary of Settings can be Saved even Without Analysis

- Summary of Settings will be appended to an existing job file or saved to a new job file.



```
#COM ===== FEMFAT job file *.ffj
#COM  Analysis Settings
#COM =====
#COM
#COM : General Factors
#COM
setValue {} {} GradientComputationMethod 33; # 16:FEMFAT 2.4 33:Vector Reconstruction
setValue {} {} InfluenceStressGradientEnduranceLimit 1
...
#COM
#COM : Surface Treatment
#COM
setValue {} {} InfluenceShotPeen 0
setValue {} {} InfluenceShotPeenMethod 17; # 17:FKM 56:Eurocode / BS 7608
...
#COM
#COM : Analysis Target
#COM
setValue {} {} DamageCalculationAim 2; # 1:MINER Original 2:MINER Modified ...
...
#COM
#COM : Global Parameters
#COM
setValue {} {} BasStressSelection 0; # 0:Automatic 1:Von Mises ...
setValue {} {} SurvivalProbability 90.0
...
```

FEMFAT job file *.ffj



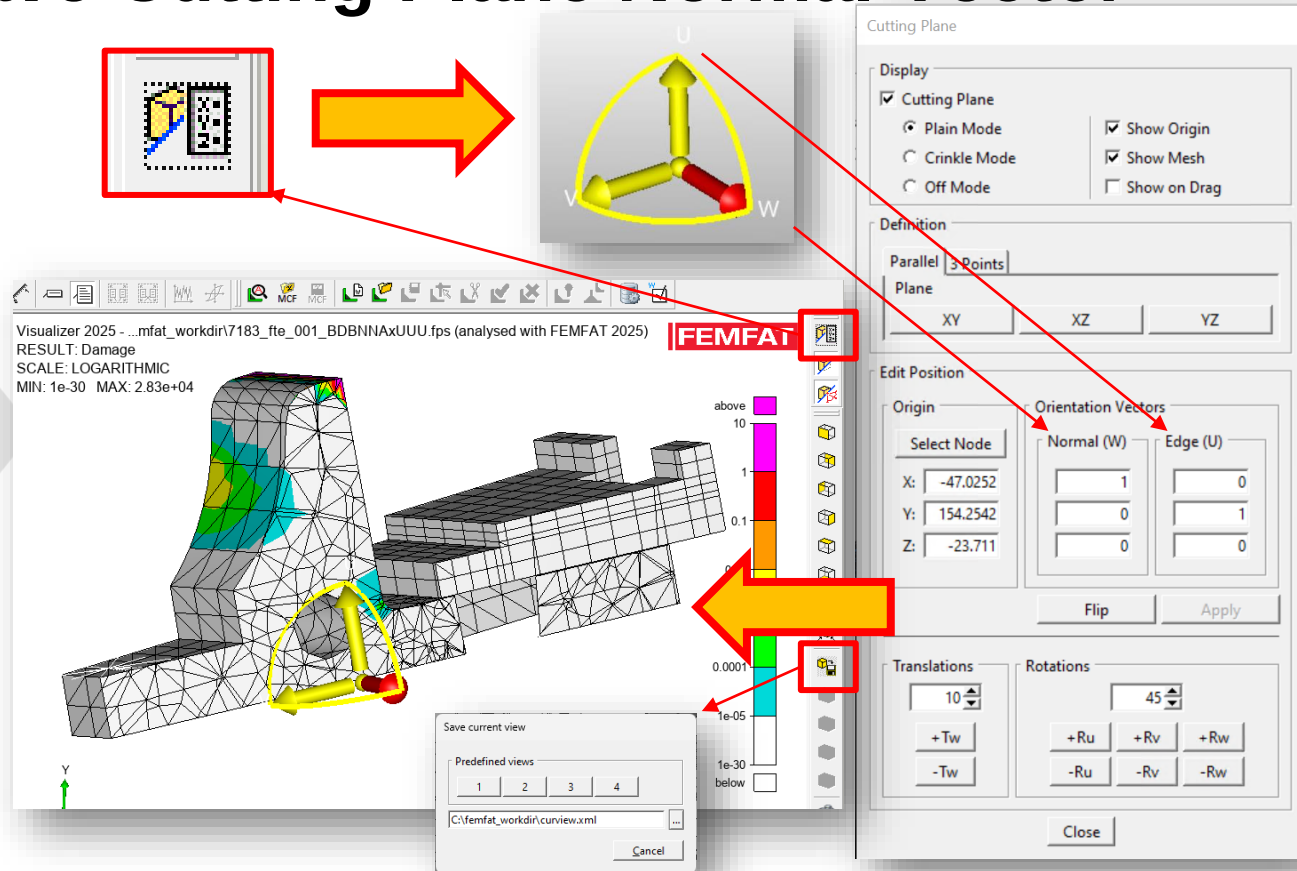


04

VISUALIZER

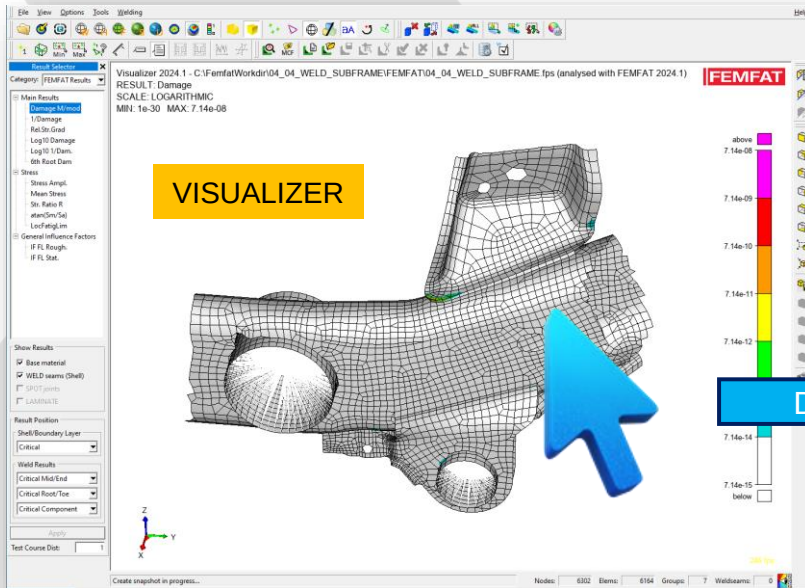
Specify and Save Cutting Plane Normal Vector

- Normal vector (W) and depending vector (U) can be set (and stored) in cutting plane dialog.
- Both vectors are unit vectors and perpendicular to each other.
- By moving the plane in the graphic window, the values are updated automatically.
- The values are greyed out until „Apply“ is hit.



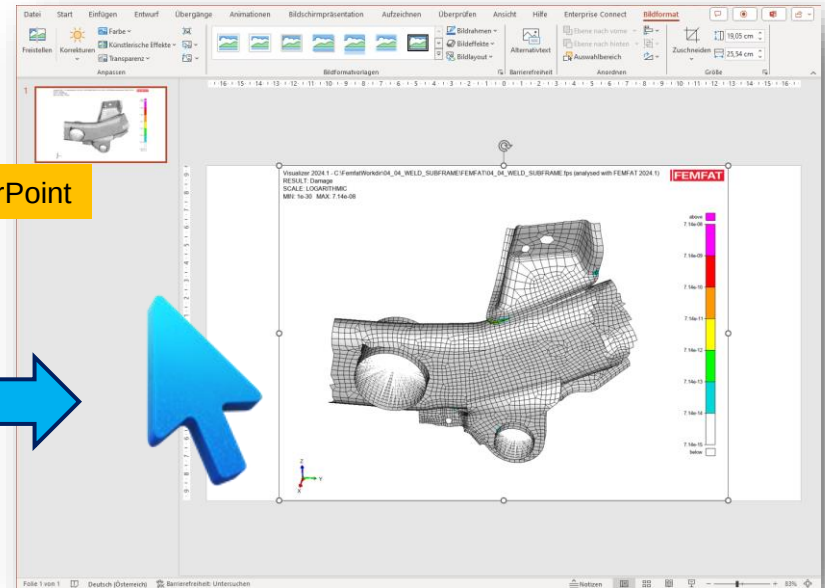
Drag & Drop for Screenshots

- Click in the graphics window and drag into office application (PowerPoint, Word, Paint, Explorer).
- With the shift key a png-file will be generated in the directory where the fps-file is located.



PowerPoint

Drag & Drop



Display of SolidWELD and 3D-WELD Nodes

- SolidWELD:
 - without modelled notch radius
 - Definition via special named sets
- 3D-WELD-Nodes
 - with modelled notch radius
 - Definition via node color 200-299

Analysis Report
Max. Damage

Result: 1.360e-06

in Node: 16991

with Material: 1 st

Component: SolidWELD Node

with Material: 2 ROOT_SUS444_Suzuki

Component: RefRadiusWELD Node

Node Label 16991N
Node Color 0
Structural Component SolidWELD
Internal Specimen Material Number 1

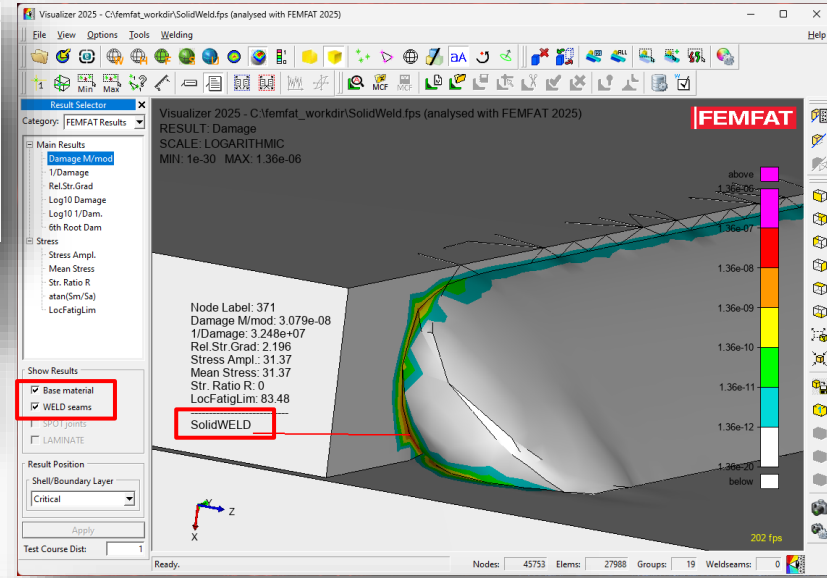
MAX. DAMAGE

Max. Damage 1.360E-06
in Node 16991
with Material 1 st
Component 7 SolidWELD

Critical Locations:

Location	NodeLab	Damage	Node Type	Result	Pos.
1	16991	1.360e-06	SolidWELD	top / surf	
2	6	5.093e-09	SolidWELD	top / surf	
3	38	7.070e-11	SolidWELD	top / surf	
4	33	3.024e-14	Base	top / surf	
5	35	3.028e-16	Base	top / surf	
6	9	1.377e-16	Base	top / surf	
7	47	1.261e-16	Base	top / surf	
8	40	3.135e-25	Base	top / surf	
9	49	7.343e-26	Base	top / surf	
10	50	4.949e-26	Base	top / surf	

End - Critical Locations



Critical Locations:

Location	NodeLab	Damage	Node Type	Result	Pos.
1	6679003	5.769e+01	RefRadiusWELD	top / surf	
2	6678935	1.933e+01	Base	top / surf	
3	6678972	1.432e+01	RefRadiusWELD	top / surf	
4	6910185	4.723e+00	Base	top / surf	
5	6910251	2.450e+00	Base	top / surf	
6	6678964	2.301e+00	Base	top / surf	
7	7008114	1.474e+00	Base	top / surf	
8	7007728	6.681e-01	Base	top / surf	
9	6678960	5.310e-01	Base	top / surf	
10	6678941	5.188e-01	Base	top / surf	

End - Critical Locations

Report file



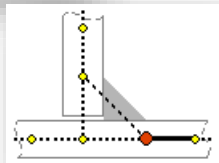
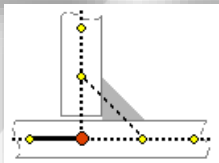
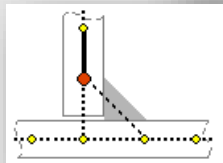
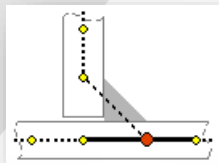
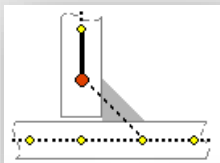
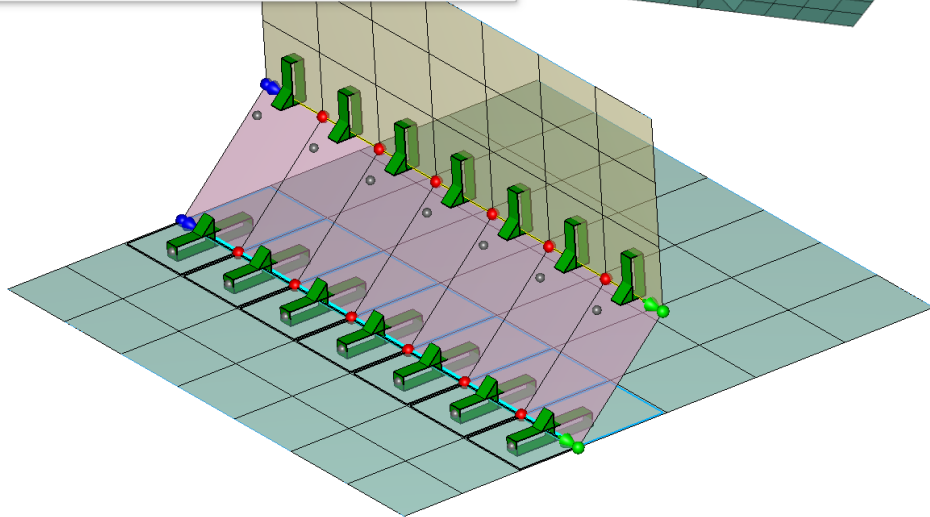
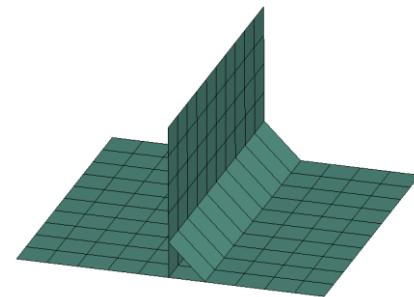
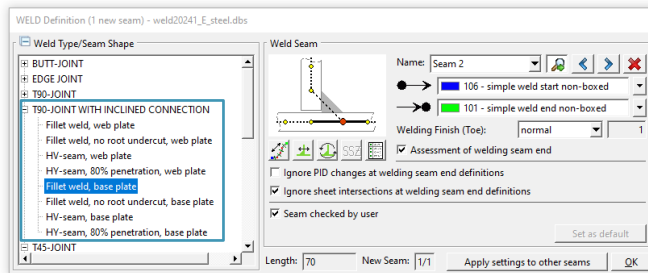
05

WELD

T-Joints with Inclined Connector Element

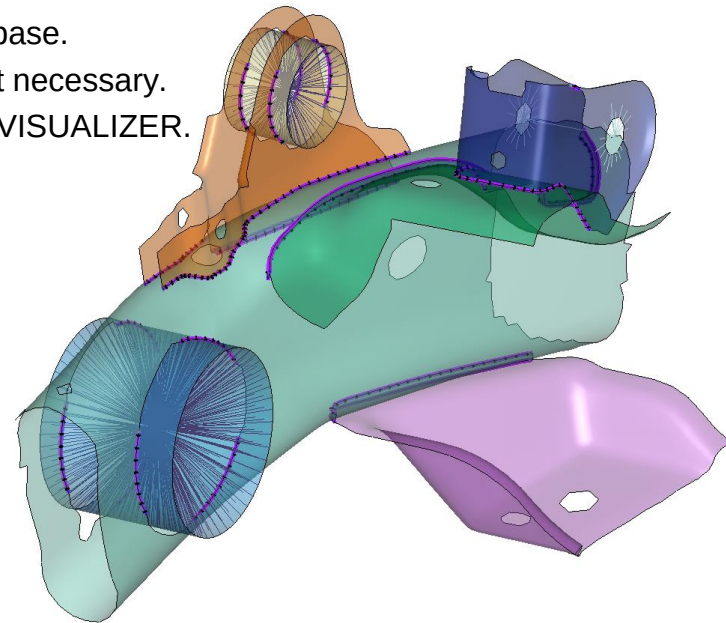
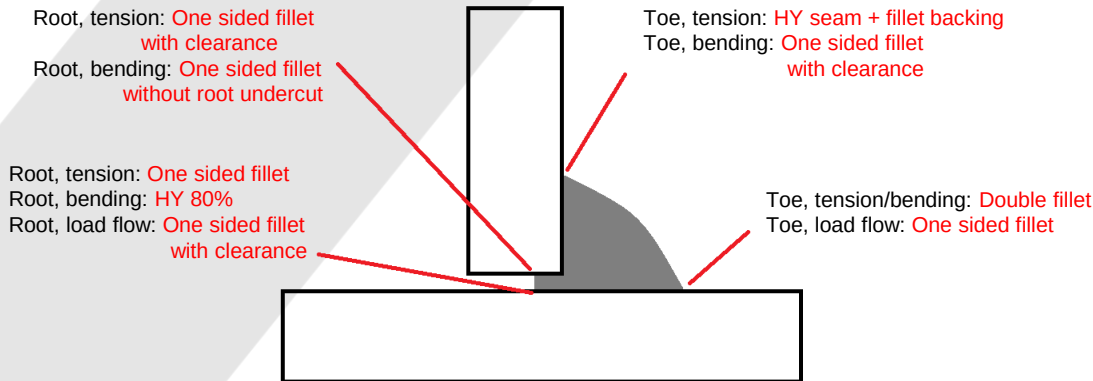
For better stiffness representation an inclined connector element can be used.

- New single sided T-joints has been implemented in the WELD databases.
- The inclined element is not assessed.
- VISUALIZER: Each seam side must be defined separately.



Extended WELD Database by Worst Case Notch Factors for Fast Identification of Critical Areas

- A detailed definition of the welding seam type (weld side, fillet, clearance, undercut, etc.) can be very time-consuming.
- For a fast fatigue analysis to identify critical areas, it is sufficient to use worst case notch factors for each welding seam position (toe and root at weld elements) of a weld type.
- WORST CASE weld types have been included in the standard WELD database.
- All seam types have been mirrored so that a change of the seam side is not necessary.
- WORST CASE weld types can be used for a scan of welding seams in the VISUALIZER.
- Example T90-Joint:

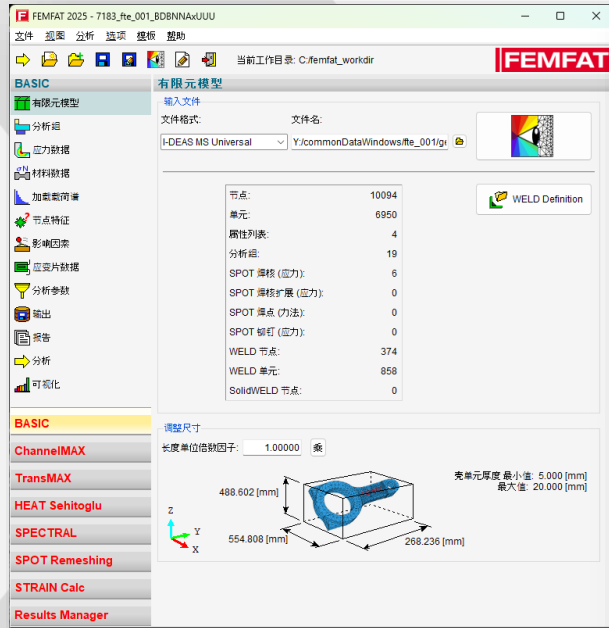




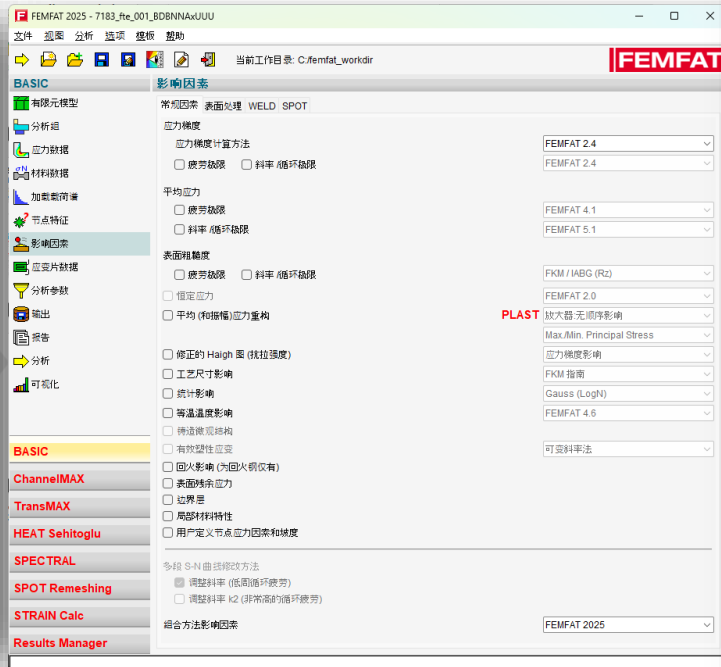
06

Miscellaneous

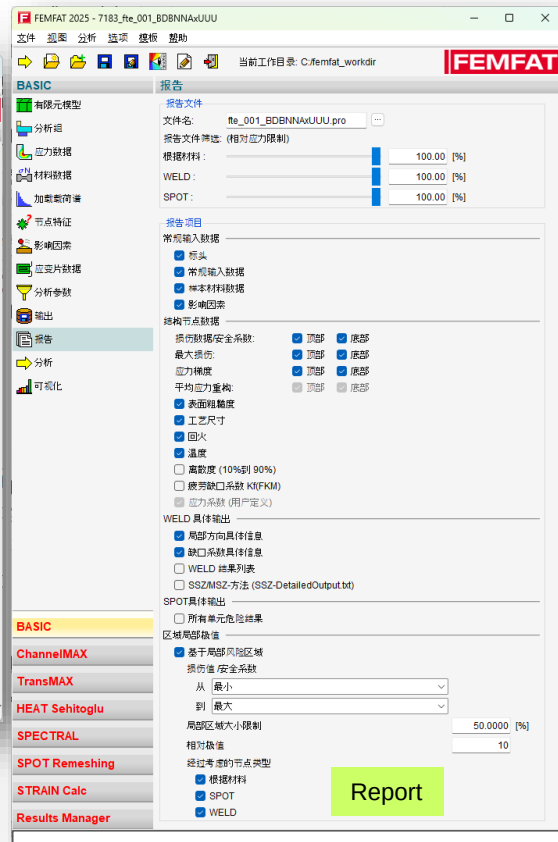
FEMFAT Graphical User Interface (GUI) in Chinese Language



FE Entities



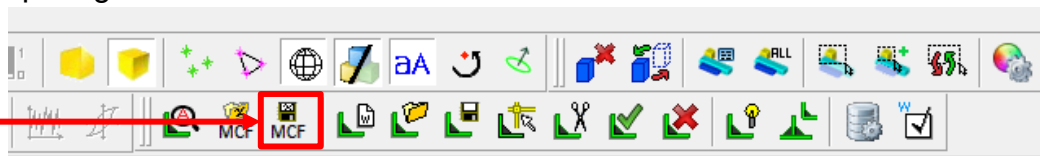
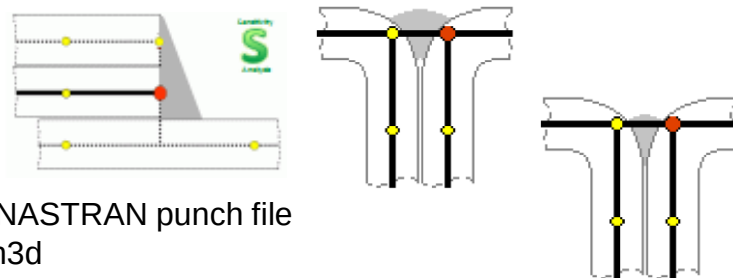
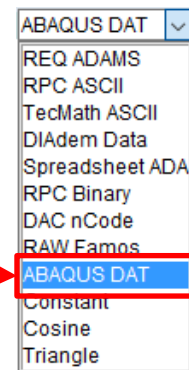
Influence Factors



Report

And Even a Lot of Many More Features ...

- New / extended FE-structure interfaces for LS-Dyna.key, Ansys.rst, NX Nastran.op2
- New / extended stress interface Permas.hdf, NX Nastran.op2, Altair OptiStruct.op2, Marc.t16, MSC Nastran.h5
- Support linear pyramids in FEMFAT and VISUALIZER
- Import of shell thickness from Altair.h3d Files
- ChannelMAX: New load history interface for Abaqus.dat file
- All nodes of "DETAILED RESULTS" group will be reported in the *.pro File
- WELD: New database according to the guideline DVS1608 (Welded aluminum constructions for railway vehicles)
- WELD: Extension of the WELD database by new joint type (double lap joint, double flared joint)
- WELD & VISUALIZER: Export of weld seam definitions in xMCF-format
- WELD: New message file dedicated for WELD messages (*.msgw)
- SPOT: An advanced methodology for rivet assessment
- SPOT: Flexible definition of load direction limits in XML-SPOT-database
- SPECTRAL: Enhanced scaling Option for PSD Diagram
- SPECTRAL: Support of the polar representation for modal coordinates in NASTRAN punch file
- SPECTRAL: Modal stresses and transfer functions from Altair OptiStruct.h3d
- PLAST: Consideration of tempering influence for K'



A man in silhouette stands on the left, looking towards a dark car on the right. The car's rear light bar is illuminated with a red glow. The background is a sunset sky with a large, light gray arrow pointing right. The text "Forward. For all." is centered in the upper half of the image.

Forward.
For all.