Newcomer Workshop

FEMFAT User Meeting 2021

18th May 2021 / Manuel Frank

FEMFAT Support

Overview

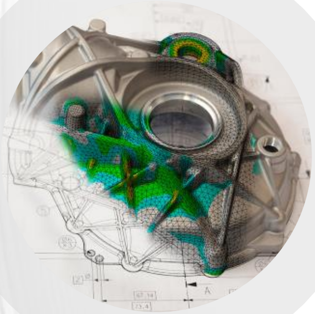
1. FEMFAT Software
2. Analysis in Time Domain
   - FEMFAT basic
   - FEMFAT max
3. Analysis in Frequency Domain
   - FEMFAT spectral
4. Joint Assessment in FEMFAT
   - FEMFAT weld
   - FEMFAT spot
   - Assessment Method for Adhesive Joints
5. Non-metal Fatigue
   - Assessment Method for Short Fiber Reinforced Plastics
   - FEMFAT laminate

FEMFAT Software

Software Products by Magna Powertrain

MAGNA

FEMFAT

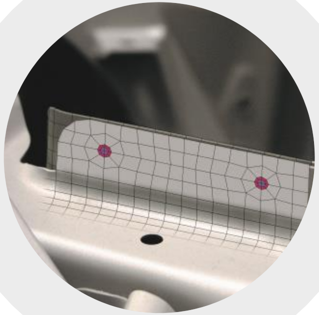


Finite Element Method Fatigue

FEMFAT LAB

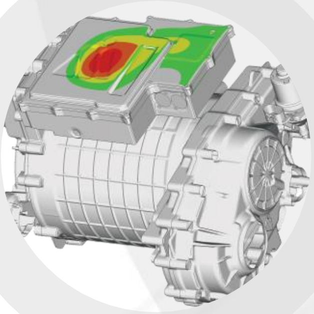
Signal Processing for Fatigue Lab and Simulation

MAMBA



Simulation of Joint Contact Phenomena

MNOISE



Acoustic Simulation Postprocessor

KULI

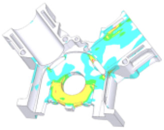
Energy Management Optimization

Depending on the stress state, joining technique or analysis target different FEMFAT modules are used for analysis.

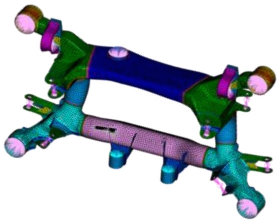
MAGNA

FEMFAT modules

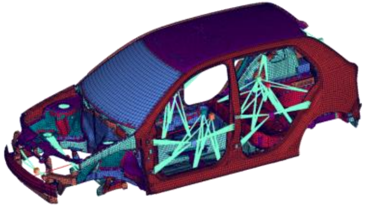
FEMFAT basic
plast
break
MEHD
FEDIS
MNOISE
MAMBA



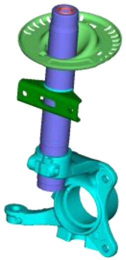
FEMFAT weld



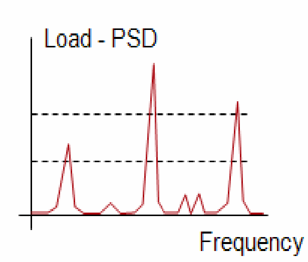
FEMFAT spot
MAMBA



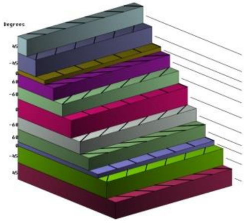
FEMFAT max



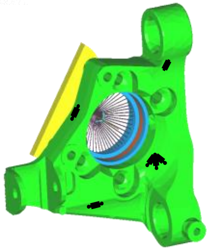
FEMFAT spectral



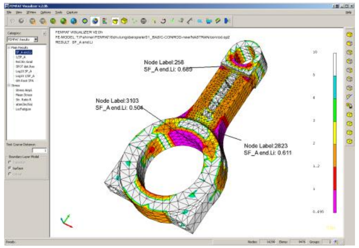
FEMFAT laminate
parallel



FEMFAT strain



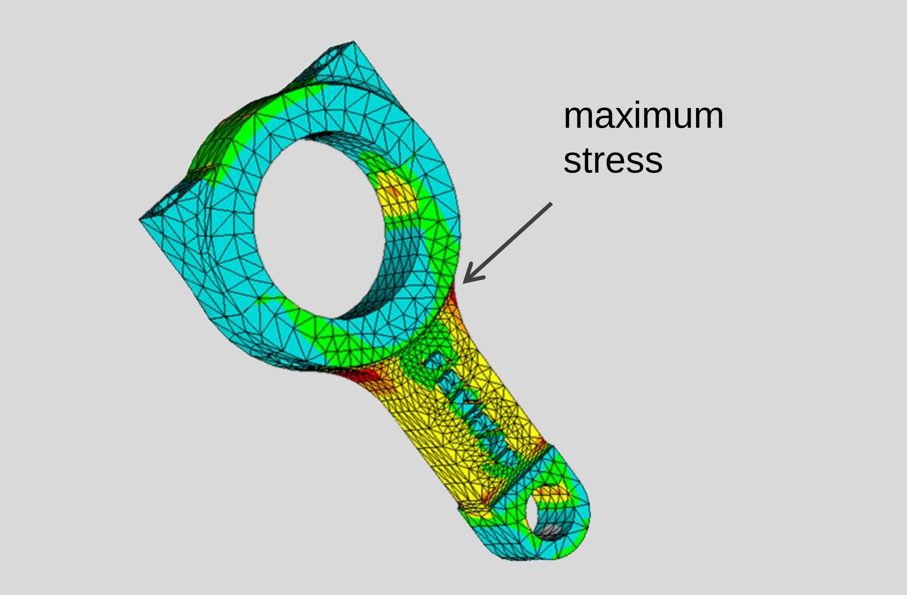
FEMFAT visualizer



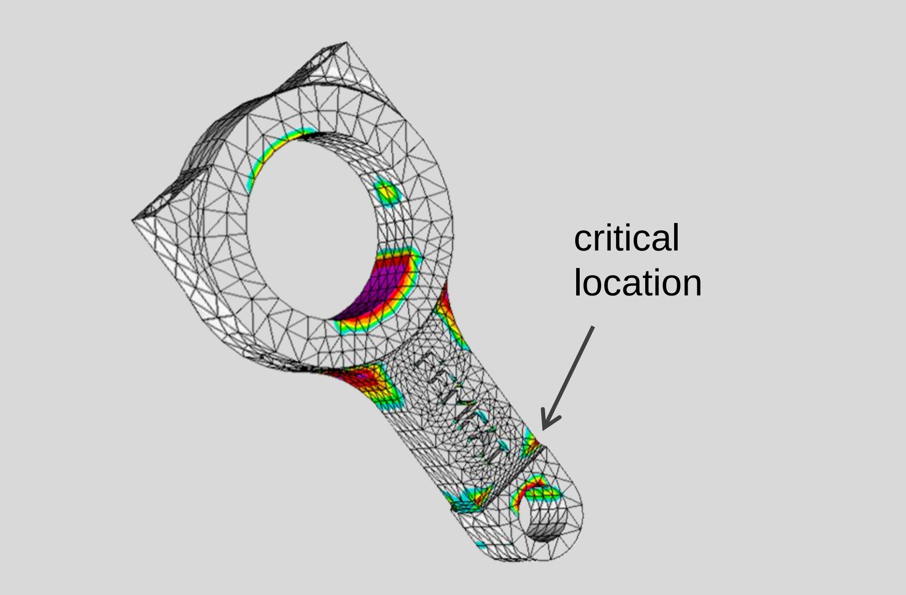
Analysis Concept

The exclusive analysis of stress in a traditional way doesn‘t often reveal damage occurrence at the right point

Traditional view



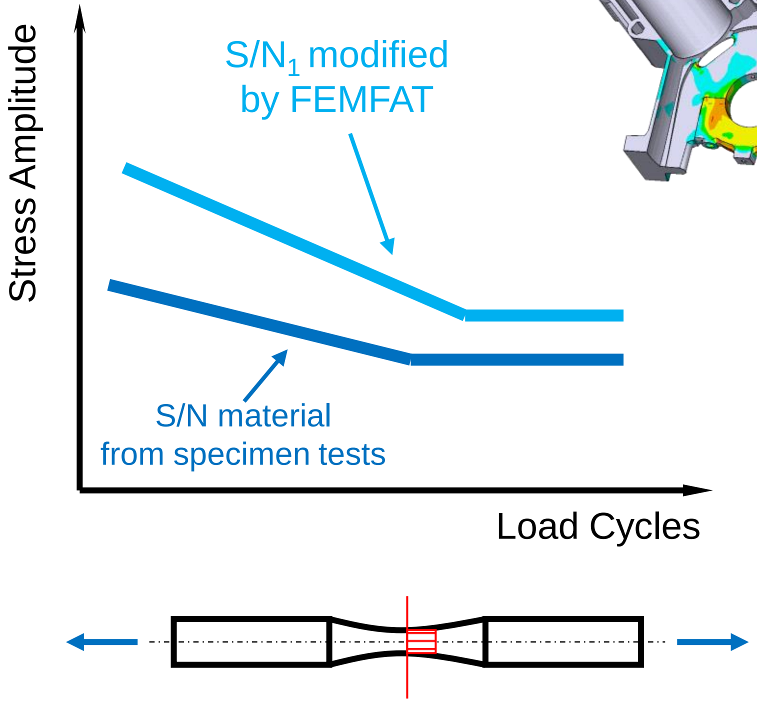
Modern life-cycle stress analysis

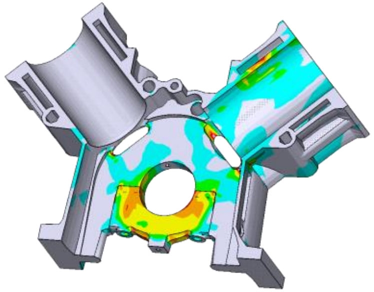


Only modern fatigue analysis tools are capable of predicting critical crack locations and the number of load cycles until failure

Local Stress Concept in FEMFAT

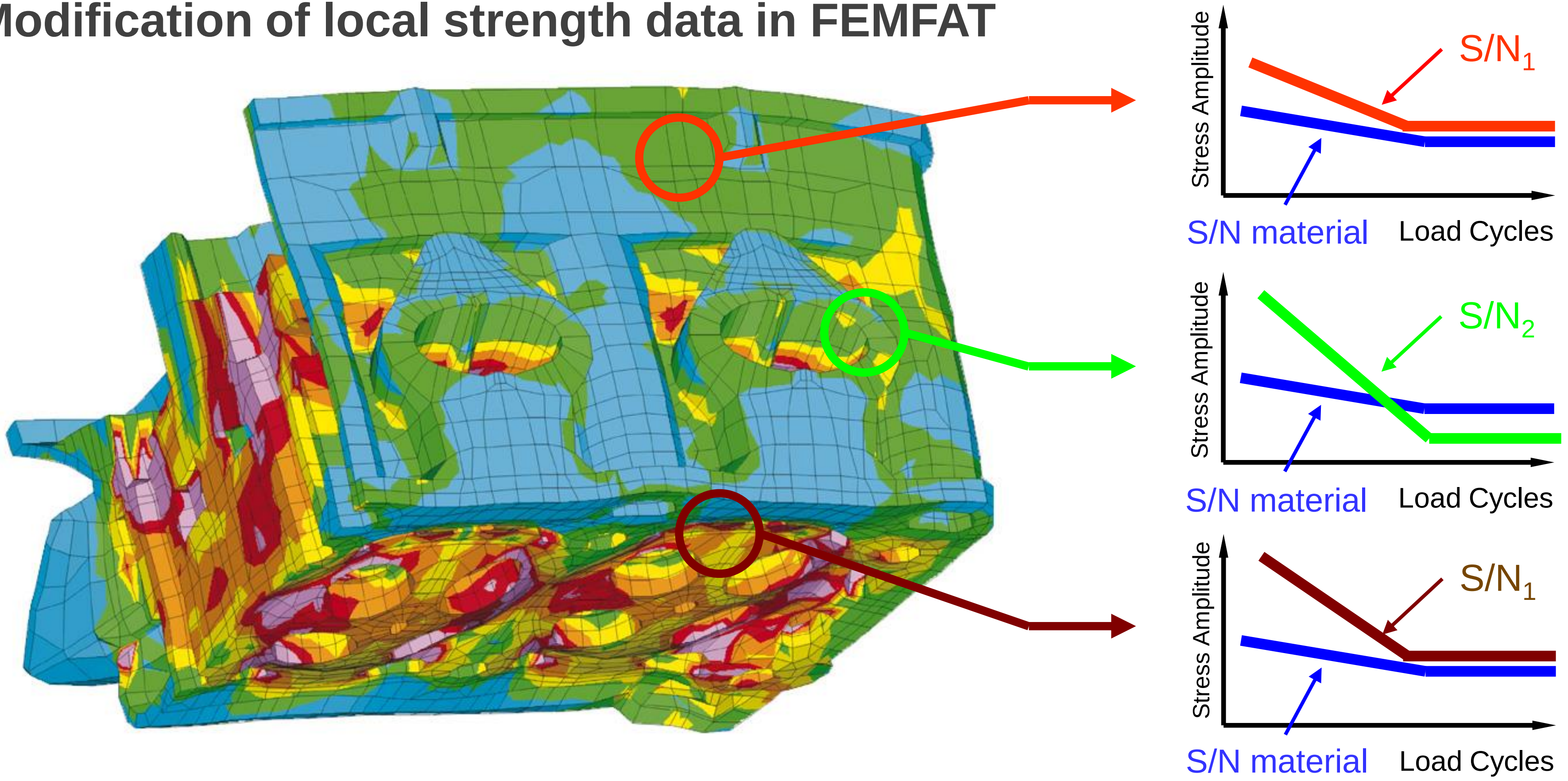
- Stress Tensors
- Material Properties
- Stress Gradient
- Mean Stress Influence
- MultiAXial Load
- Technological Influences
- Size Influence
- Temperature Influence
- PLASTic Deformations
- SPOT Joints
- Anisotropical Behaviour of Arc WELDs
- etc.





For each node of the FE-model a synthetic S/N curve is defined depending on local temperature, surface roughness, stress gradient, …

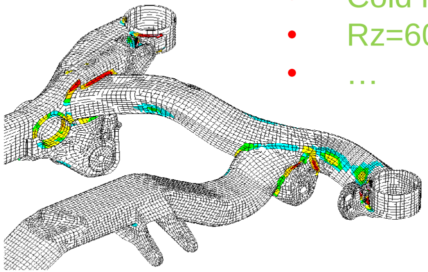
Modification of local strength data in FEMFAT



Data Processing

Data Processing in FEMFAT

FE Entities



- Cold Rolled
- Rz=60µm
- …

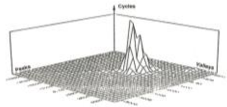
by FEA:

Stress Data

Linear + PLAST
Non-Linear

by MKS or Measurement:

Load Data

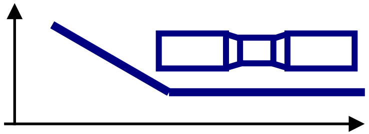


- EN-AW-7020
- DP500
- …

Groups

assigned to

Material Data

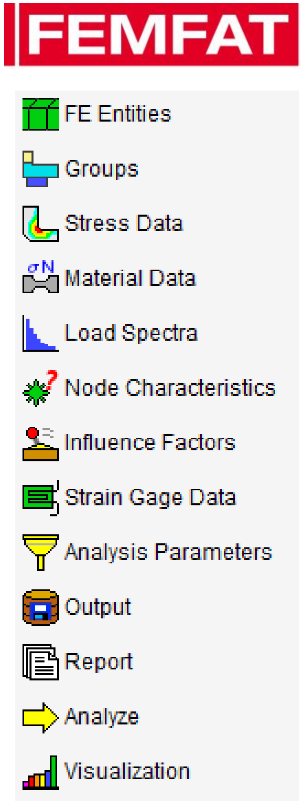


acting on

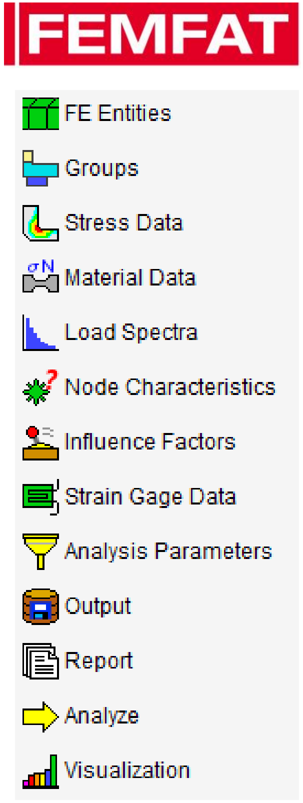
Node Characteristics

considered by

Influence Factors

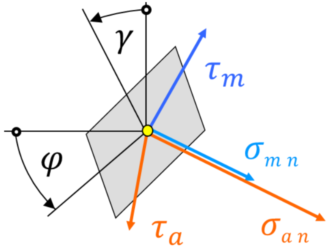


Data Processing in FEMFAT



Analysis Parameters

- Analysis Filter
- Cutting Plane Parameters

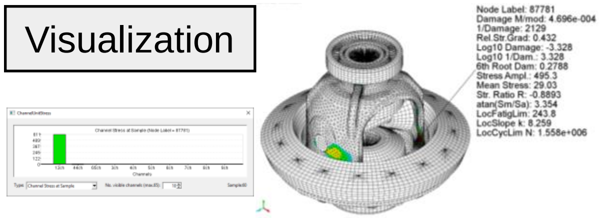


- Analysis Target:
  - Damage
  - Endurance Safety Factor
  - Static Safety Factor
  - Stress/ Strain Comparison
  - Degree of Multiaxiality (MAX)

Output → Output of 6 node results (*.dma, *.op2, …)

Report → Protocol of detailed node results (*.pro)

Visualization



FEMFAT visualizer

- All results
- Detailed evaluation

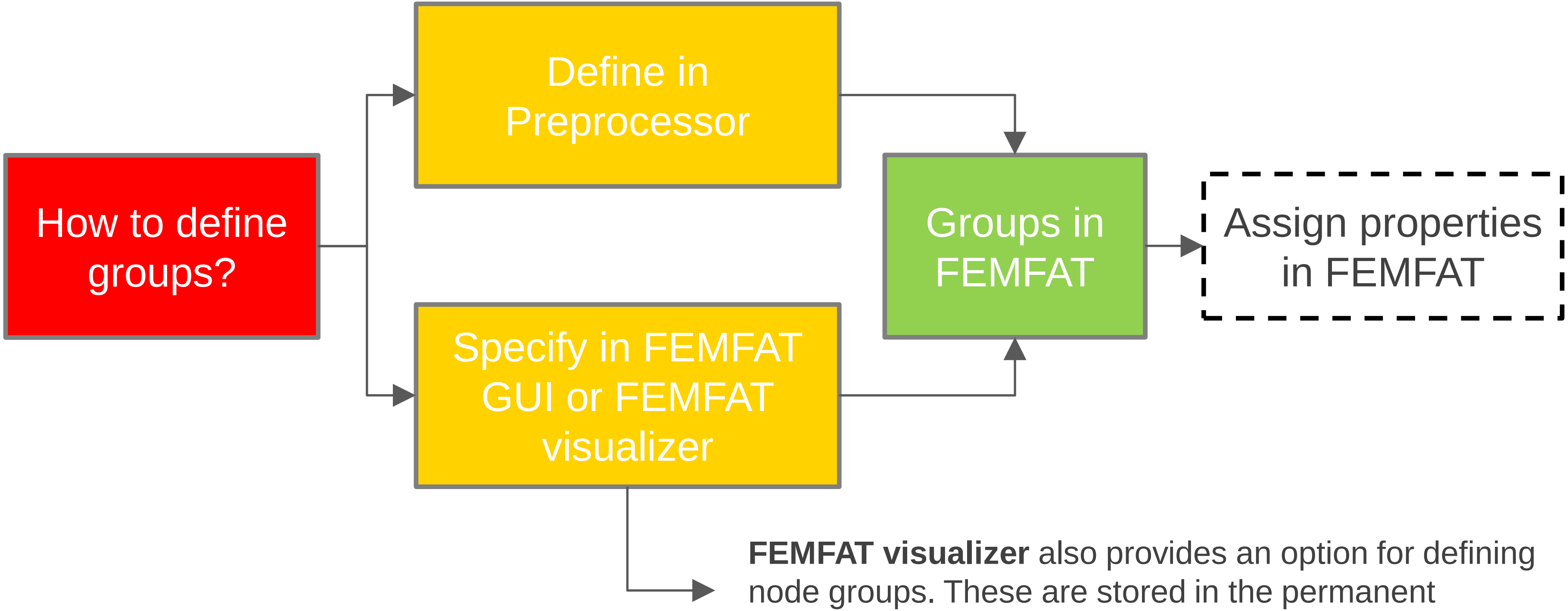
Postprocessor

- 6 results

Groups

Groups can either be previously defined in the finite element application and be imported with FEM data or they can be created directly in FEMFAT

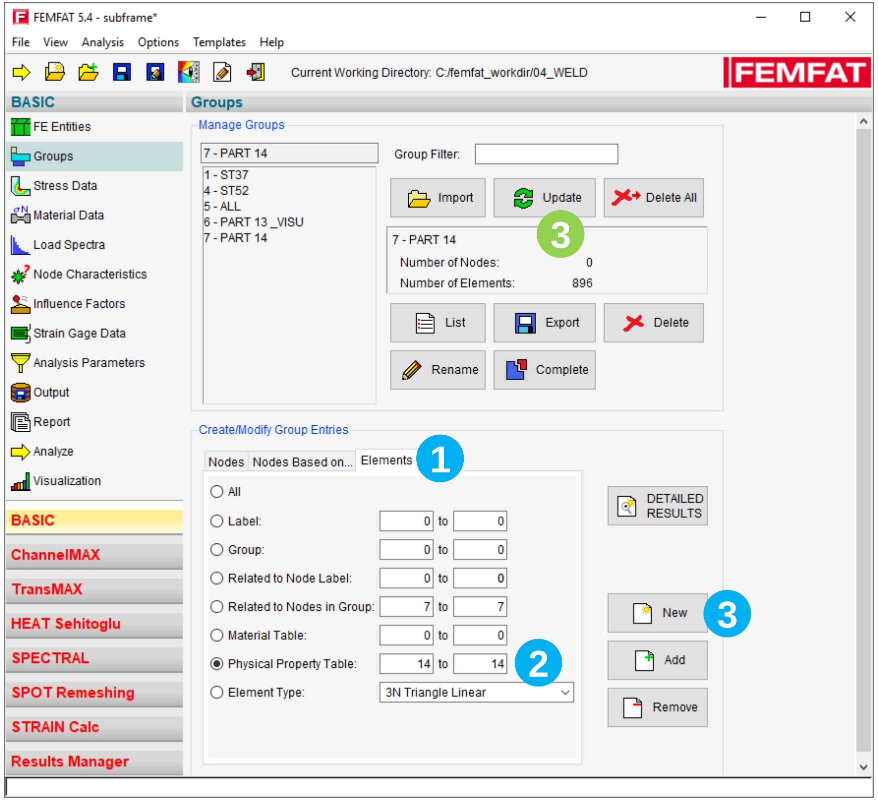
Group-handling in FEMFAT



FEMFAT visualizer also provides an option for defining node groups. These are stored in the permanent scratch file (*.fps) and can be imported into **FEMFAT**.

Create a group for a PID

Example

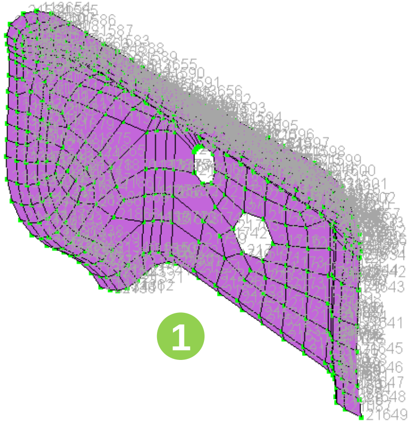


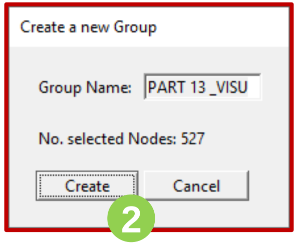
In GUI:

1. Go to Elements/ Physical Property Table
2. Select PID
3. Click New.

In VISUALIZER:

1. Select Nodes
2. Create a Group
3. Update GUI





Material Data

There are two possible ways to define a material in FEMFAT. Read it from the material database or define a material using material generator

MAGNA

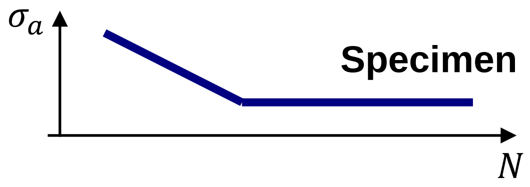
Defining a material

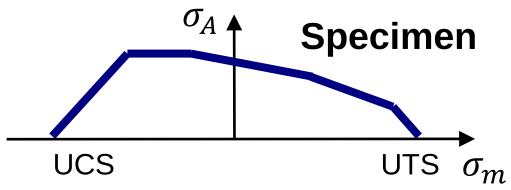
Material Database in FEMFAT

- 450 materials available
- expandable

FEMFAT basic
FINITE ELEMENT METHOD FATIGUE

Material Generator in FEMFAT

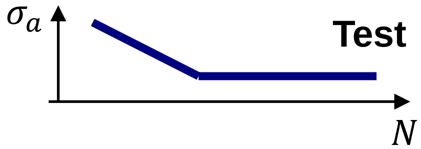




Break points/ Shape of Haigh diagram depends on the material class and come from internal studies of FKM and TGL

Automatic Generation
- Definition UTS

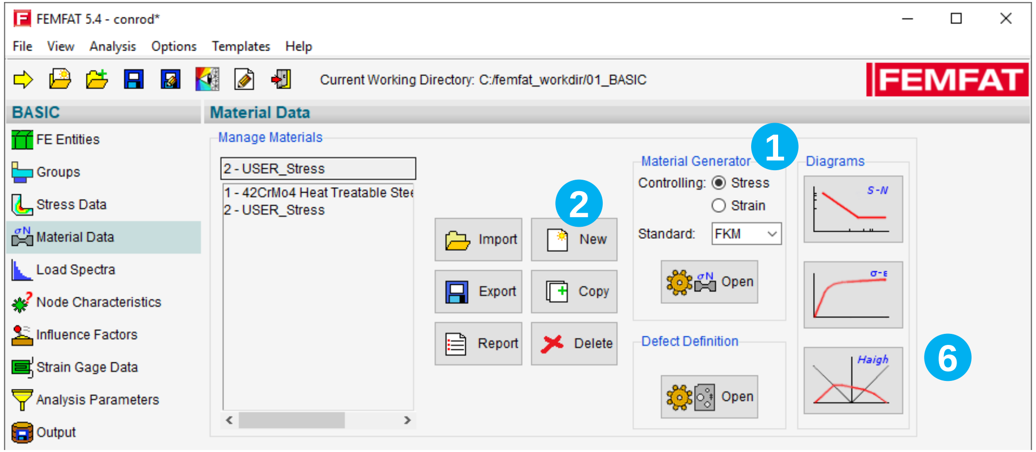
Material Data of Specimen

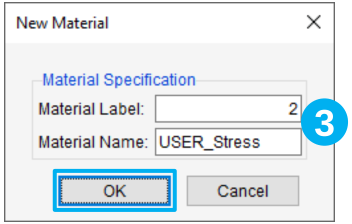


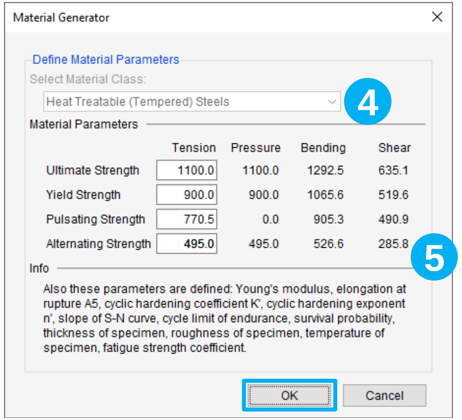
User Defined Material

User Modified Material
- High temp. values; E; UTS; Yield; K’; n’ …

Defining a material using stress data







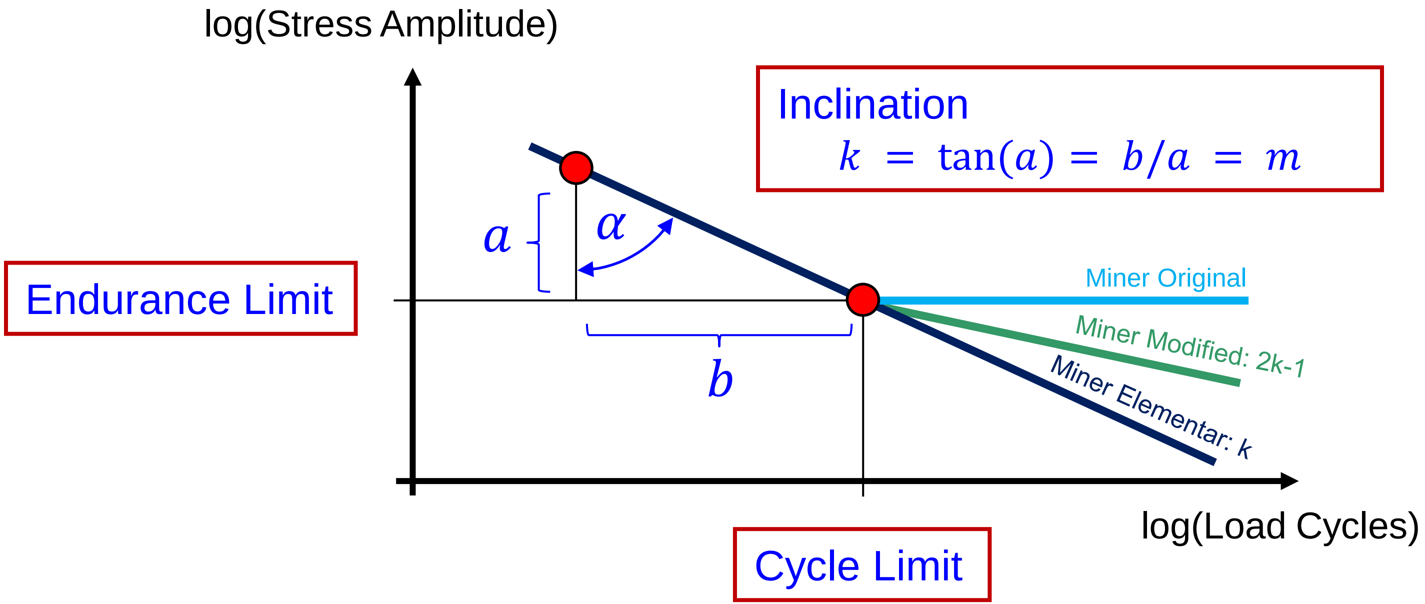
1. Click Stress Controlling with FKM Standard
2. Click New
3. Name it “USER_Stress“. Confirm with OK
4. Change material class to “Heat Treatable Steels“
5. Enter UTS. Enter YS and observe the change. Confirm with OK

6. Check the Diagrams

Influence Factors

S/N curve – Influenced Parameter

S/N curve is defined for $R = -1$



$$\left(\frac{\sigma_A}{\sigma_D}\right) = \left(\frac{N}{N_D}\right)^{-\frac{1}{k}}$$

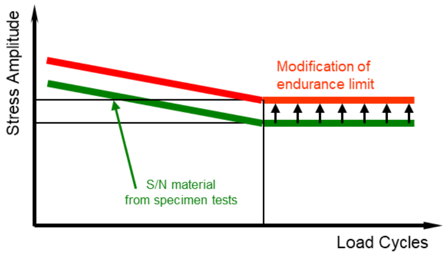
Endurance stress limit, slope and endurance cycle limit of local S/N curve are modified as well as specimen Haigh-diagram

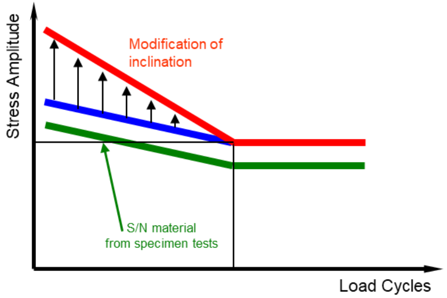
Influence factors

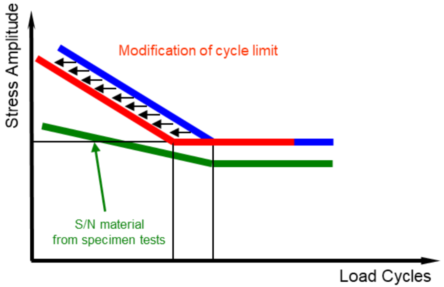
Influences	Endurance stress limit	Slope	Endurance cycle limit
Notch influence (by relative stress gradient)	●	●	●
Mean stress influence	●	●	●
Influence of surface roughness	●	●	●
Technological parameter influence	●	-	-
Tempering condition	●	-	-
Technological surface treatment	●	-	-
- Shot peening	●	-	-
- Rolling	●	-	-
- Carburizing	●	-	-
- Nitriding	●	-	-
- Carbonitriding	●	-	-
- Induction hardening	●	-	-
- Flame hardening	●	-	-
- General surface factor	●	-	-
Temperature influence	●	-	-
Statistical influence	●	-	-
Forging influence (technological factor)	●	●	●
Cast microstructure	●	-	-

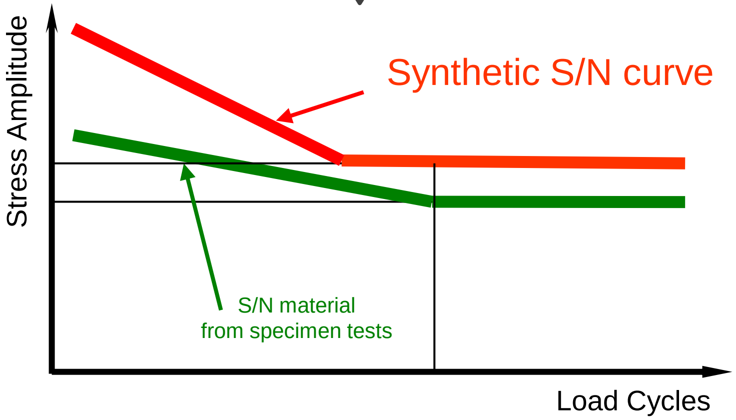
The locally considered S/N curve results from the superimposed modification of all influences activated for the analysis.

Synthetic S/N curve







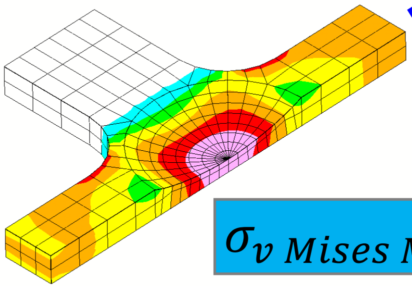


MAGNA

Mean stress influence

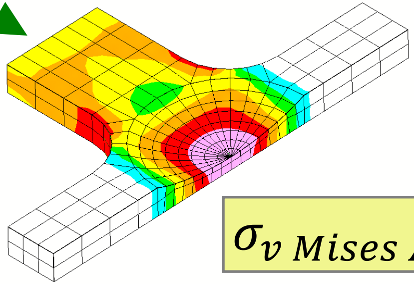
TEST 1

Mean M1



$\sigma_{v\ Mises\ M1}$

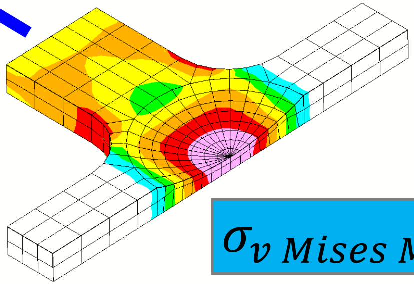
Amplitude A1



$\sigma_{v\ Mises\ A1}$

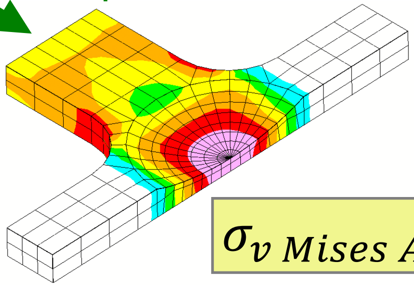
TEST 2

Mean M2



$\sigma_{v\ Mises\ M2}$

Amplitude A2



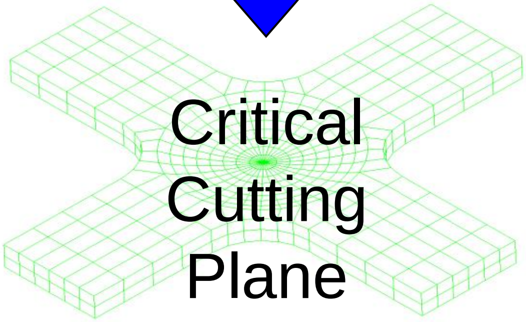
$\sigma_{v\ Mises\ A2}$

$\sigma_{v\ Mises\ M1} = \sigma_{v\ Mises\ M2}$

$\sigma_{v\ Mises\ A1} = \sigma_{v\ Mises\ A2}$

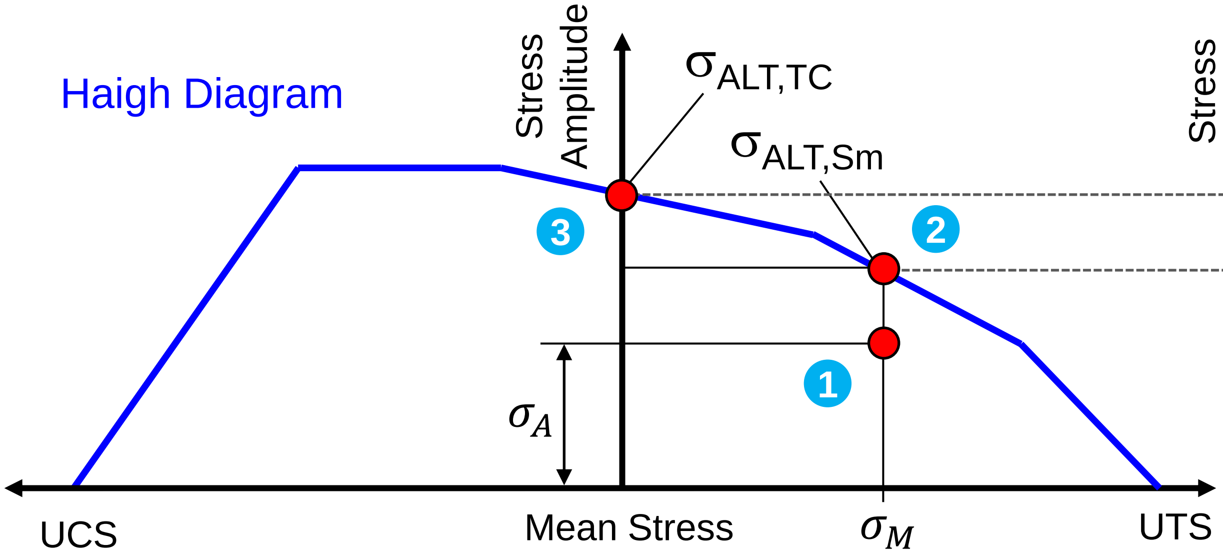
But:

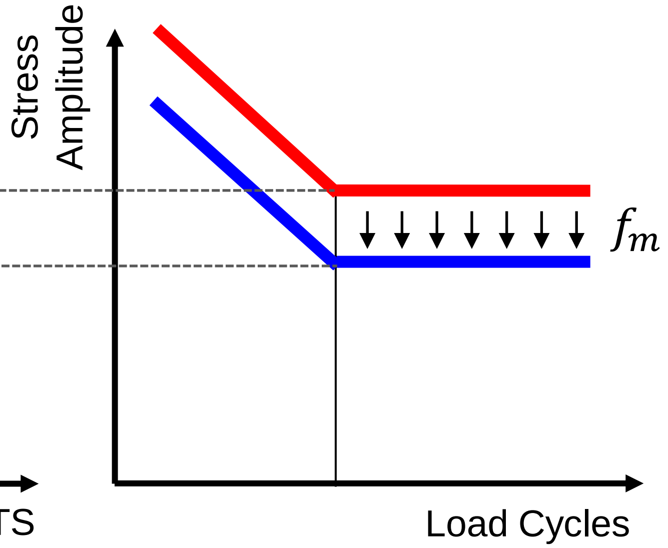
Damage 1 ≠ Damage 2



S/N curve is transformed due to mean stress at node. Stress amplitude is not taken into account for the modification

Mean stress influence – Influence on endurance limit





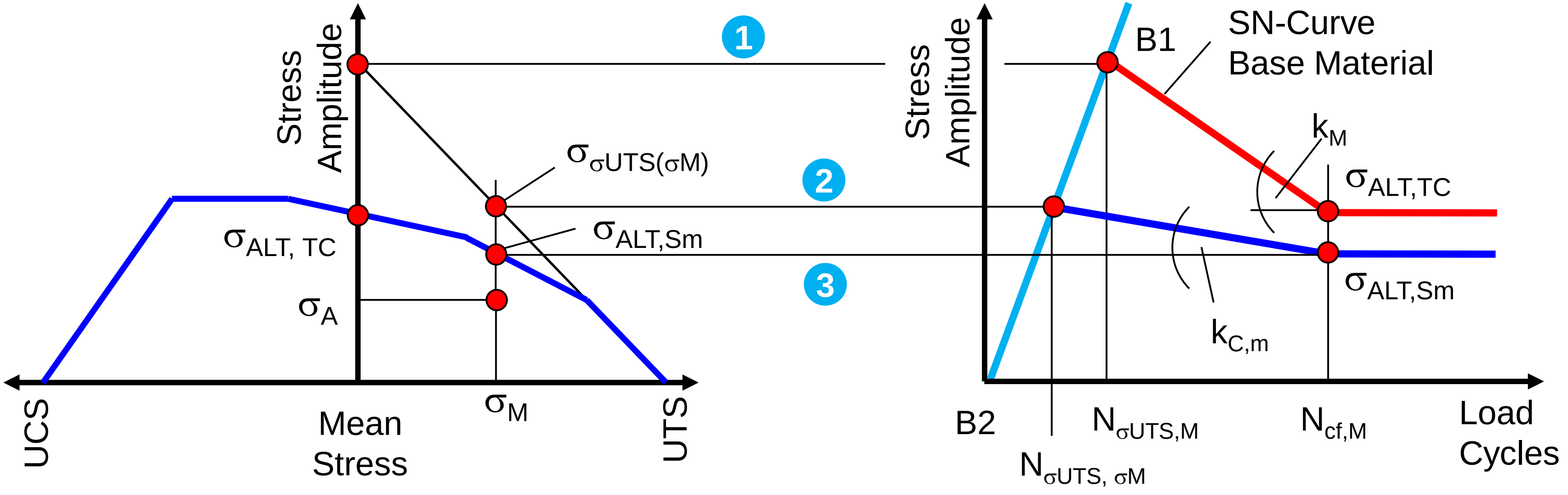
1. A stress state is defined by its mean and amplitude stress
2. Determine endurance limit $\sigma\_(ALT,S\_m)$ for this given point
3. Take endurance limit at zero mean stress for influence factor f_m

Modification of Fatigue Stress due to mean stress by the factor

$$f_m = \frac{\sigma_{ALT,S_m}}{\sigma_{ALT,TC}}$$

Inclination of S/N curve is modified for a given mean stress at node

Mean stress influence – Influence on inclination of S/N curve

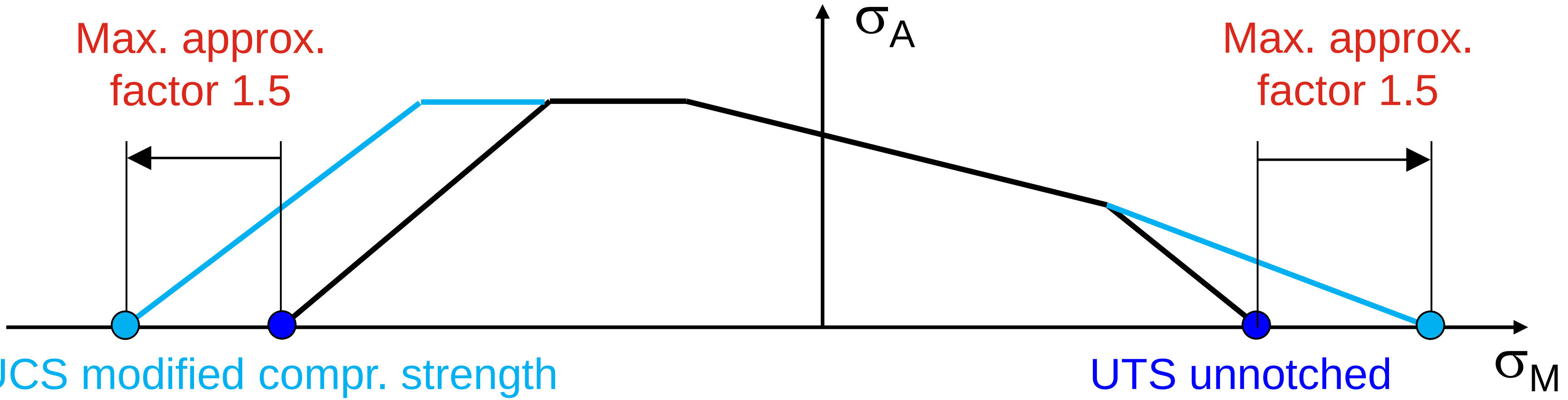


1. Determine the original S/N curve of the base material
2. Map the new ultimate tensile stress on the line B1-B2
3. Determine the inclination of the modified S/N curve

$$k_{C,m} = \frac{\log\left[\frac{N_{cf,M}}{N_{\sigma UTS(\sigma_m)}}\right]}{\log\left[\frac{\sigma_{\sigma UTS}(\sigma_m)}{\sigma_{ALT,Sm}}\right]}$$

Modified Haigh diagram

Modification of Haigh diagram:

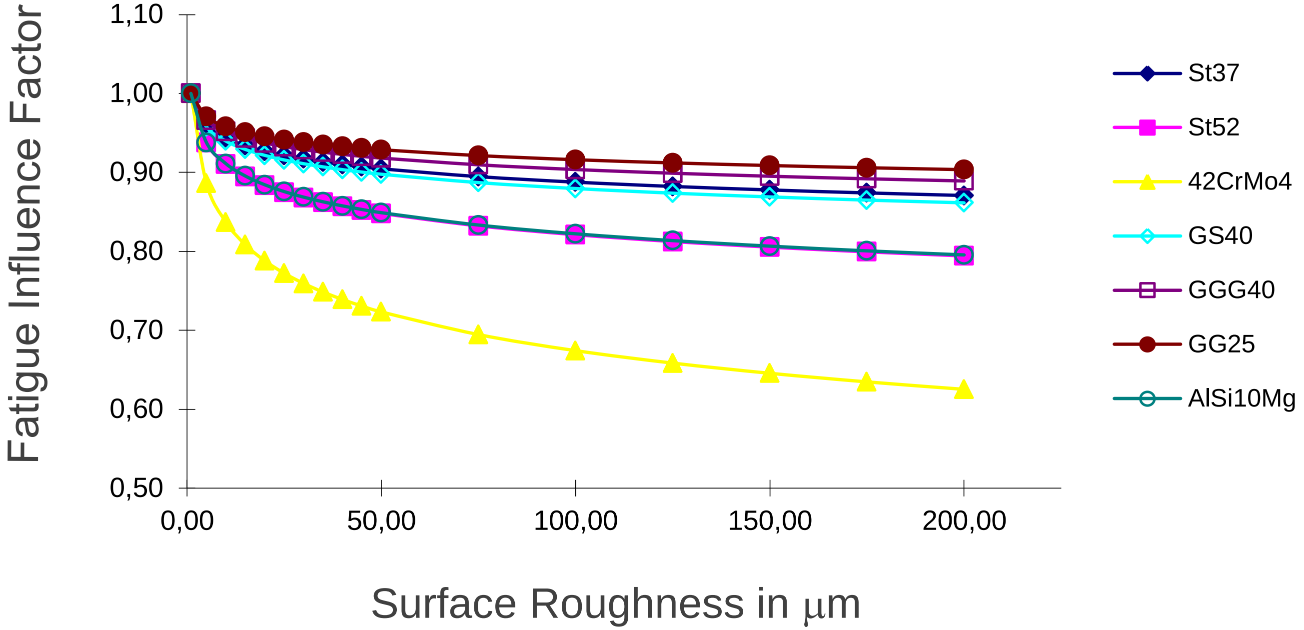


- Increase of the UTS due to support effect in notches
- Different for tensile and compression stress

The standards use different roughness values (R_t for IABG/ R_t for FKM and TGL).
If a material class is not covered in a standard, the influence is not considered.

Surface Roughness

FKM takes the Average Surface Roughness R_z

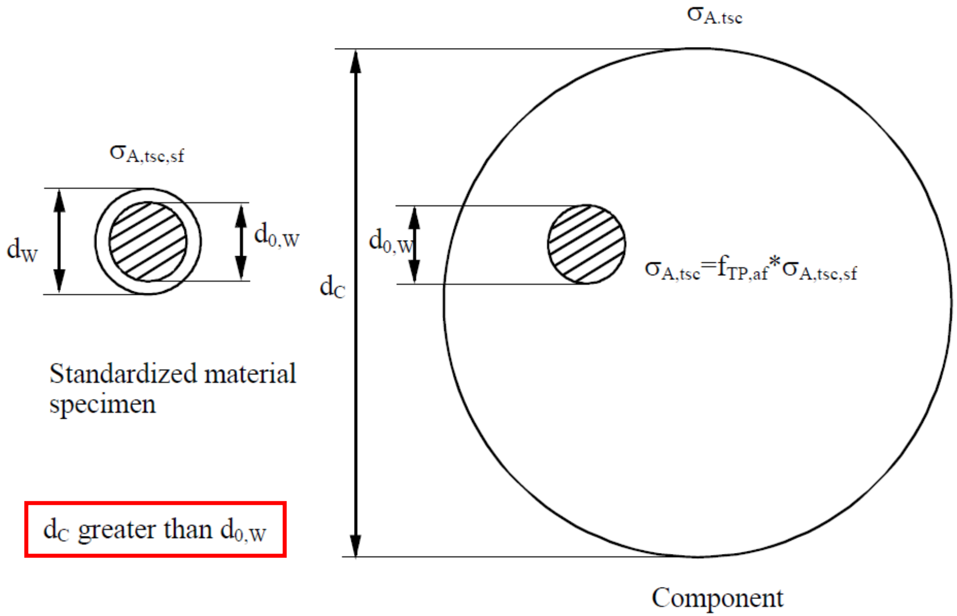


Implemented Standards:

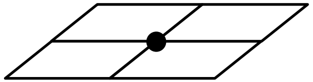
- IABG
- TGL
- FKM

The different strengths of materials as a function of the effective diameter of semis, the type of material and the technological treatment are considered

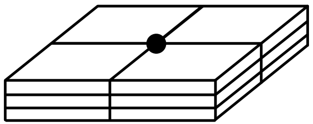
Technological Size at 3D Nodes



FEMFAT determines the technological parameter influence factor on the basis of the **FKM** guideline. This takes into consideration the differing strengths of materials as a function of the effective diameter of the semis or the unfinished castings, the type of material and the technological treatment, e.g. tempering.



Thickness results from adjacent shell elements

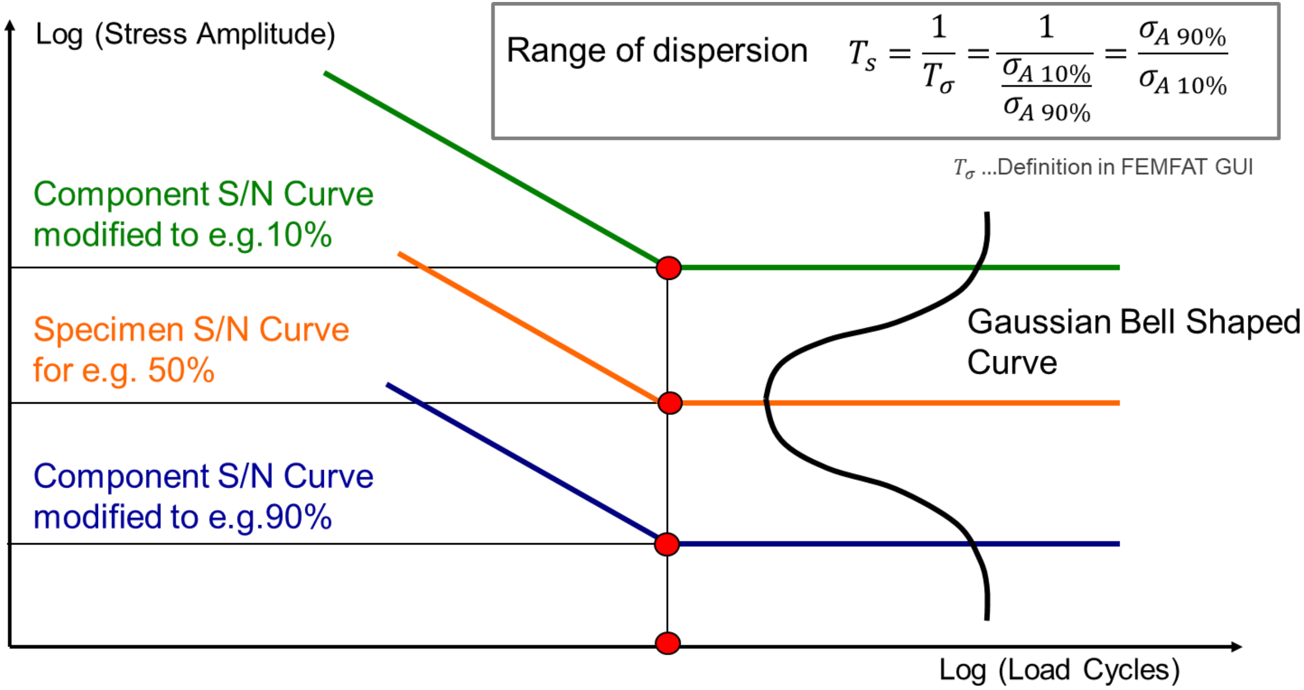


Thickness must be defined by node characteristic

The wall thickness defined here applies only to nodes on 3D elements. For shell element nodes, the wall thickness is determined by the average thickness of adjacent shell elements.

Component S/N curve is considered to follow a log-normal distribution. S/N curve is modified according to the demanded survival probability.

Statistical Influence



Range of Dispersion is used to transform the S/N curve to the desired survival probability.

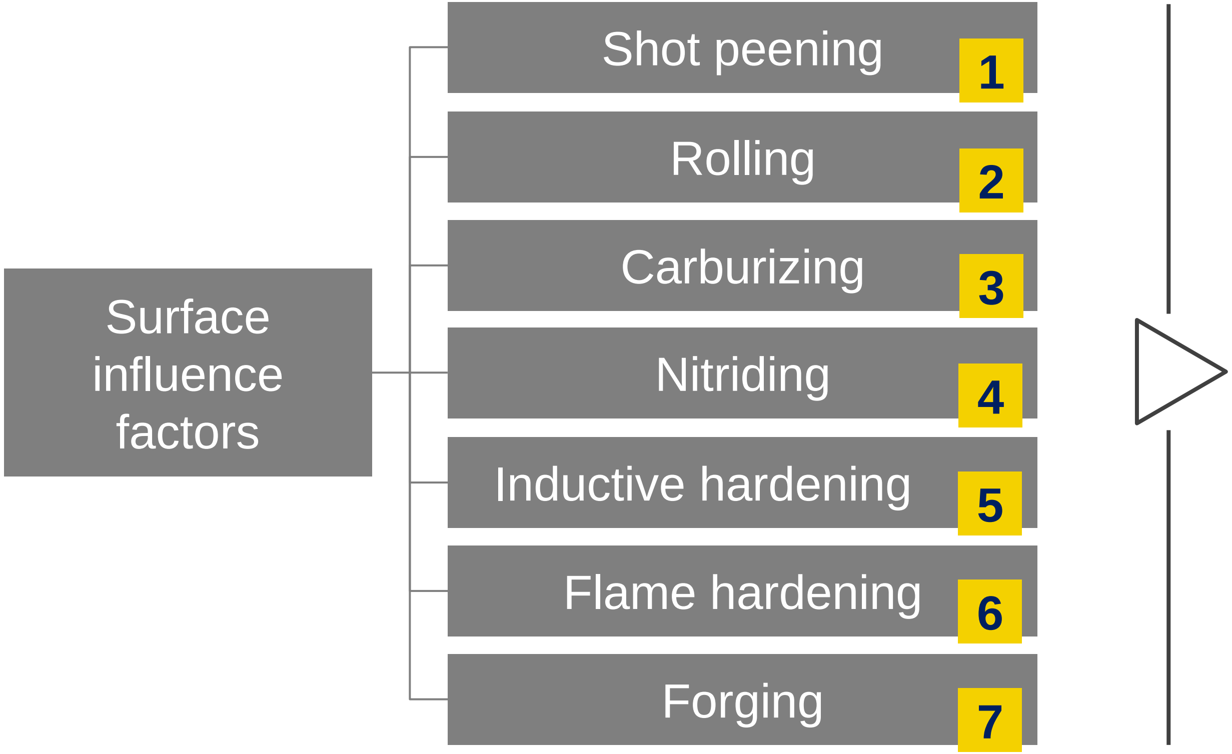
Isothermal Temperature Influence

Implemented Methods:

FKM	If the influence is solely considered using FKM guideline endurance limit of S/N curve is modified by the fatigue influence factor.
FEMFAT 4.5	The S/N curve and the Haigh diagram are modified. Modification of the static and cyclic material parameters is carried out acc. to FKM.
User Defined	User defined temperature dependent material data. The S/N curve, Haigh diagram, cyclic stabilized σ-ε curve, support factor (= stress gradient influence factor) and the equivalent stress are modified.
FEMFAT 4.6	The S/N curve, Haigh diagram, cyclic σ-ε curve are modified. Modification of the static and cyclic parameters acc. to FKM. The cyclic coefficient of hardening K' is modified proportional to the tensile strength (acc. to UML). The cyclic hardening exponent n' is not altered.

The surface treatment influence is only considered as far as the endurance fatigue limit is concerned.

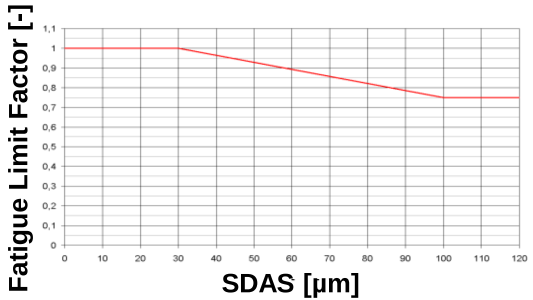
Process Influence Surface Treatment



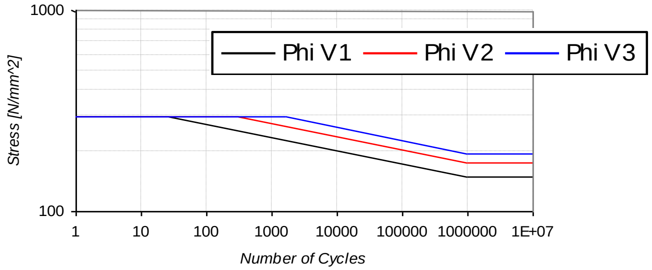
Considered acc. to **FKM** guideline by modifying the endurance limit.

Process Influence

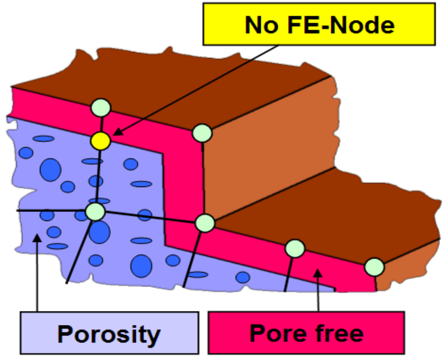
- **Microstructure Parameter**



- **Effective Plastic Strain**



- **Boundary Layer Analysis Model**

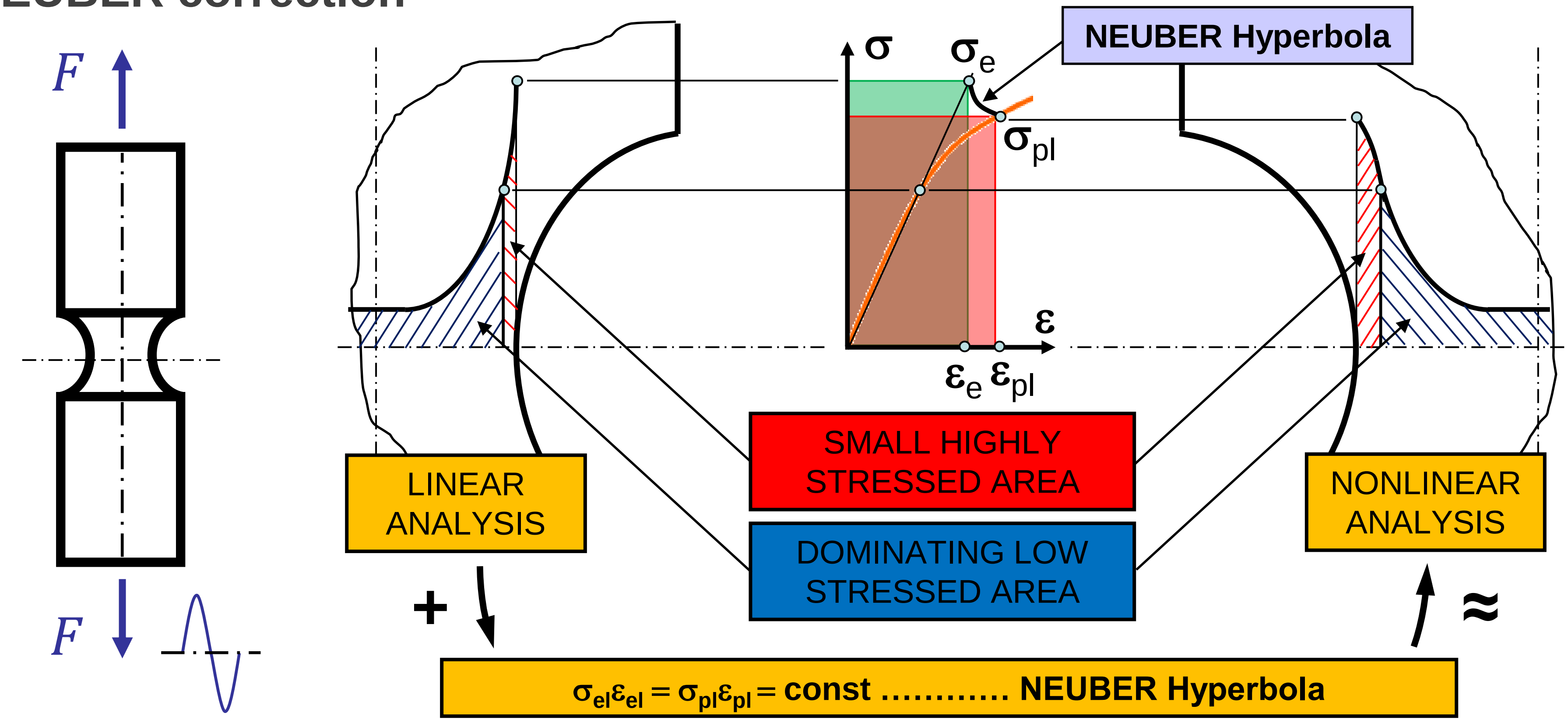


- **Tempering Condition**
- **Surface Residual Stress**
- ...

FEMFAT plast

NEUBER correction using NEUBER hyperbola works well in small highly stressed areas.

NEUBER correction

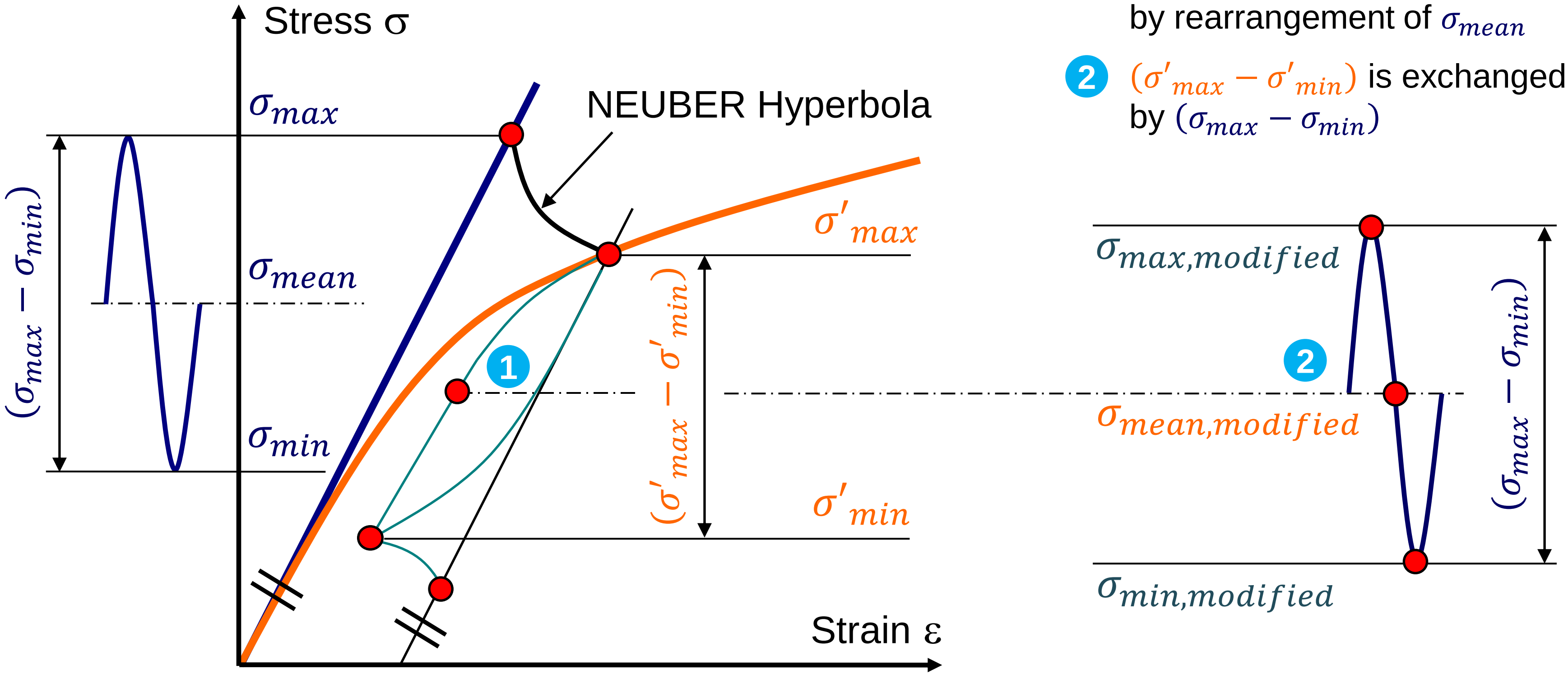


$\sigma_{el}\varepsilon_{el} = \sigma_{pl}\varepsilon_{pl} = const$ ………… NEUBER Hyperbola

After NEUBER correction of upper and lower stress FEMFAT modifies the mean stress but uses the original amplitude to remain conservative.

Modification of mean stress and amplitude

1. $\sigma_{mean,Modified}$ is determined by rearrangement of σ_{mean}
2. $(\sigma'_{max} - \sigma'_{min})$ is exchanged by $(\sigma_{max} - \sigma_{min})$

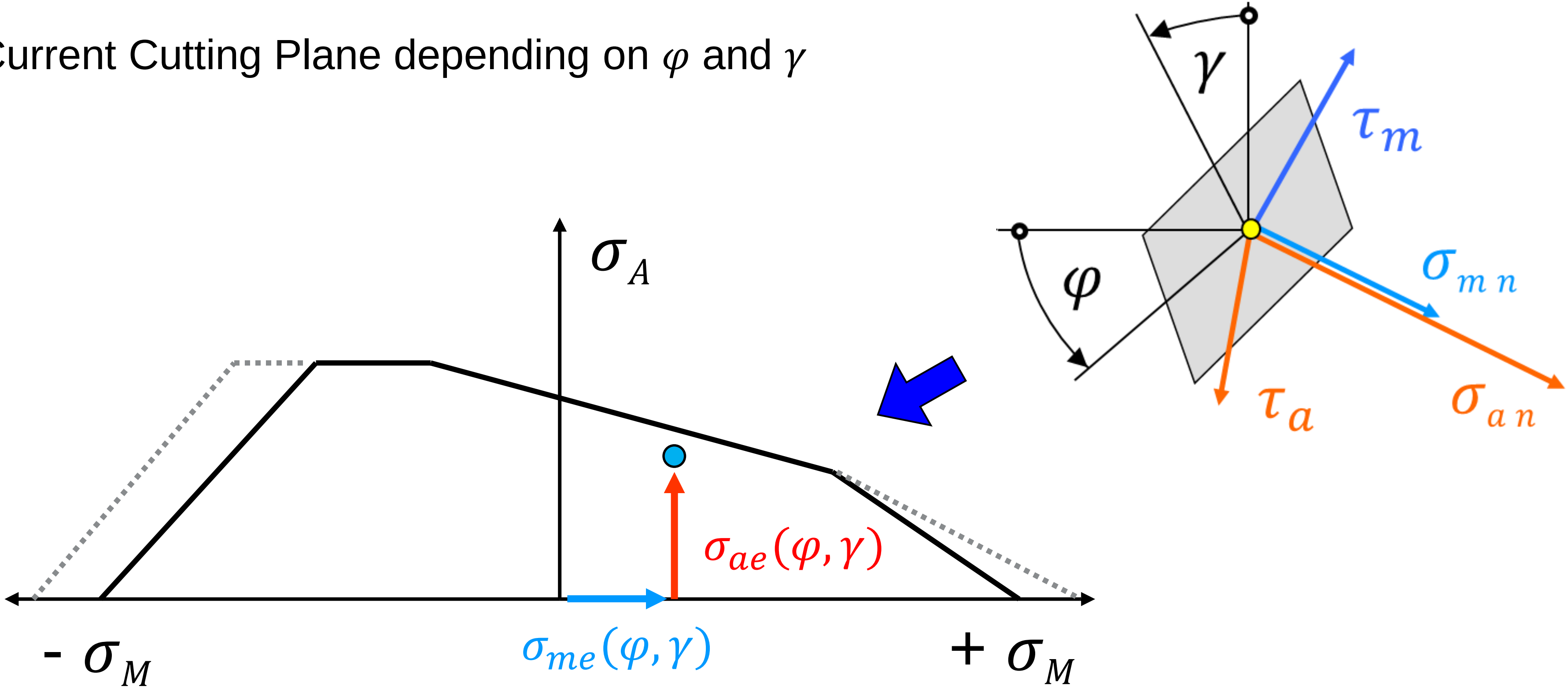


Critical Plane and Stress Computation

Depending on the cutting plane (φ,γ) the equivalent stresses are calculated for middle stress and amplitude stress. Haigh diagram is used for evaluation.

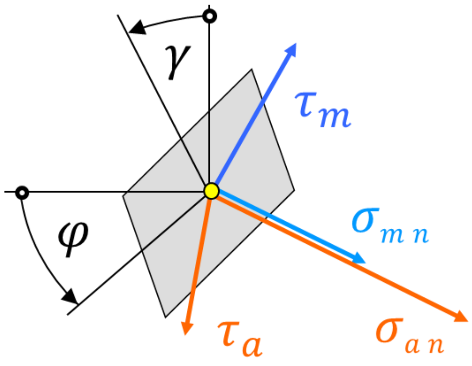
Cutting plane criterion (1/3)

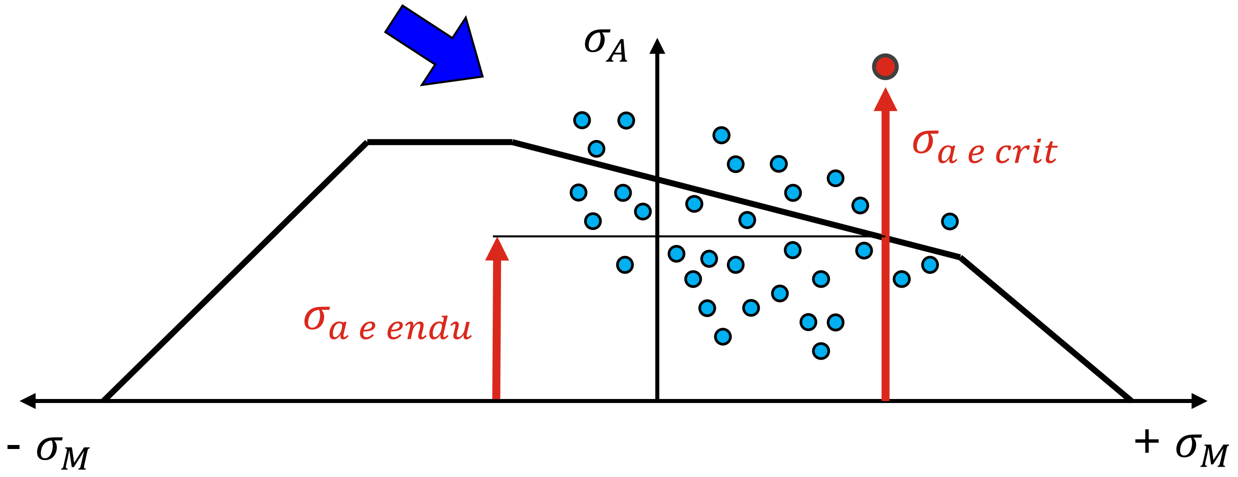
Current Cutting Plane depending on φ and γ



Cutting plane criterion (2/3)

HAIGH diagram and loading points are calculated for all cutting planes

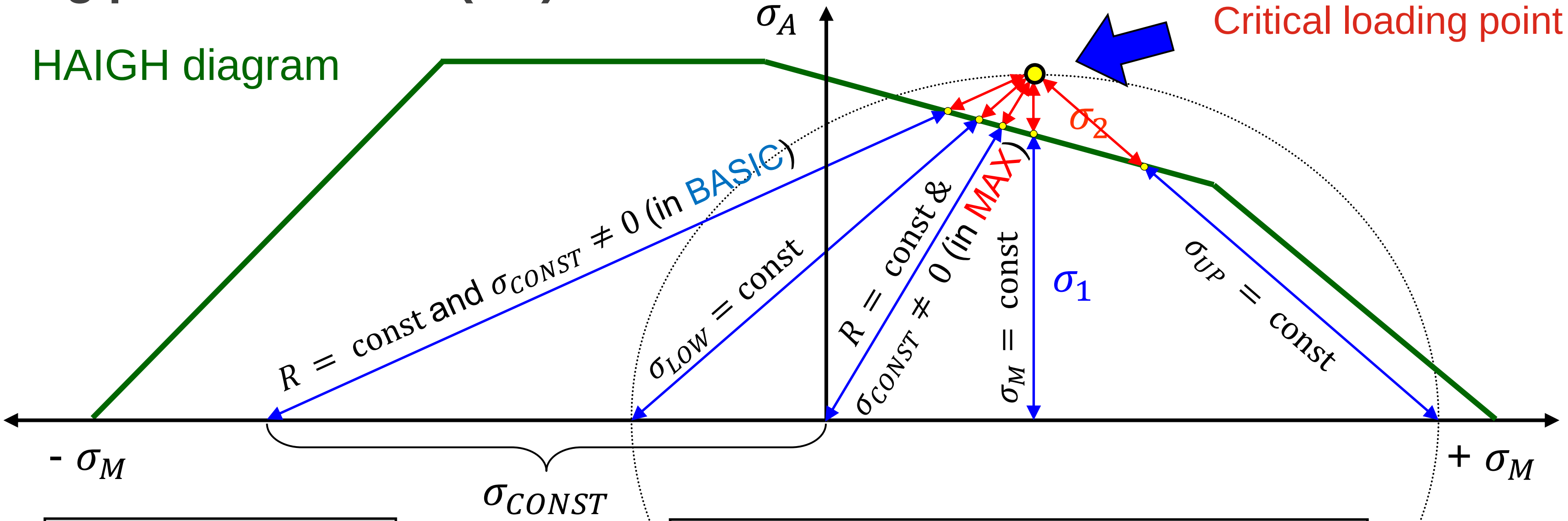




Criteria ($\sigma_m = const$) for critical loading point:

$$\frac{\sigma_{a\ e\ endu}}{\sigma_{a\ e\ crit}} \rightarrow \text{Min}$$

Cutting plane criterion (3/3)



σ_M = const
σ_{LOW} = const
σ_{UP} = const
R = const

Criteria for critical loading point: $\frac{\sigma_1}{\sigma_1 + \sigma_2} \rightarrow \text{Min}$

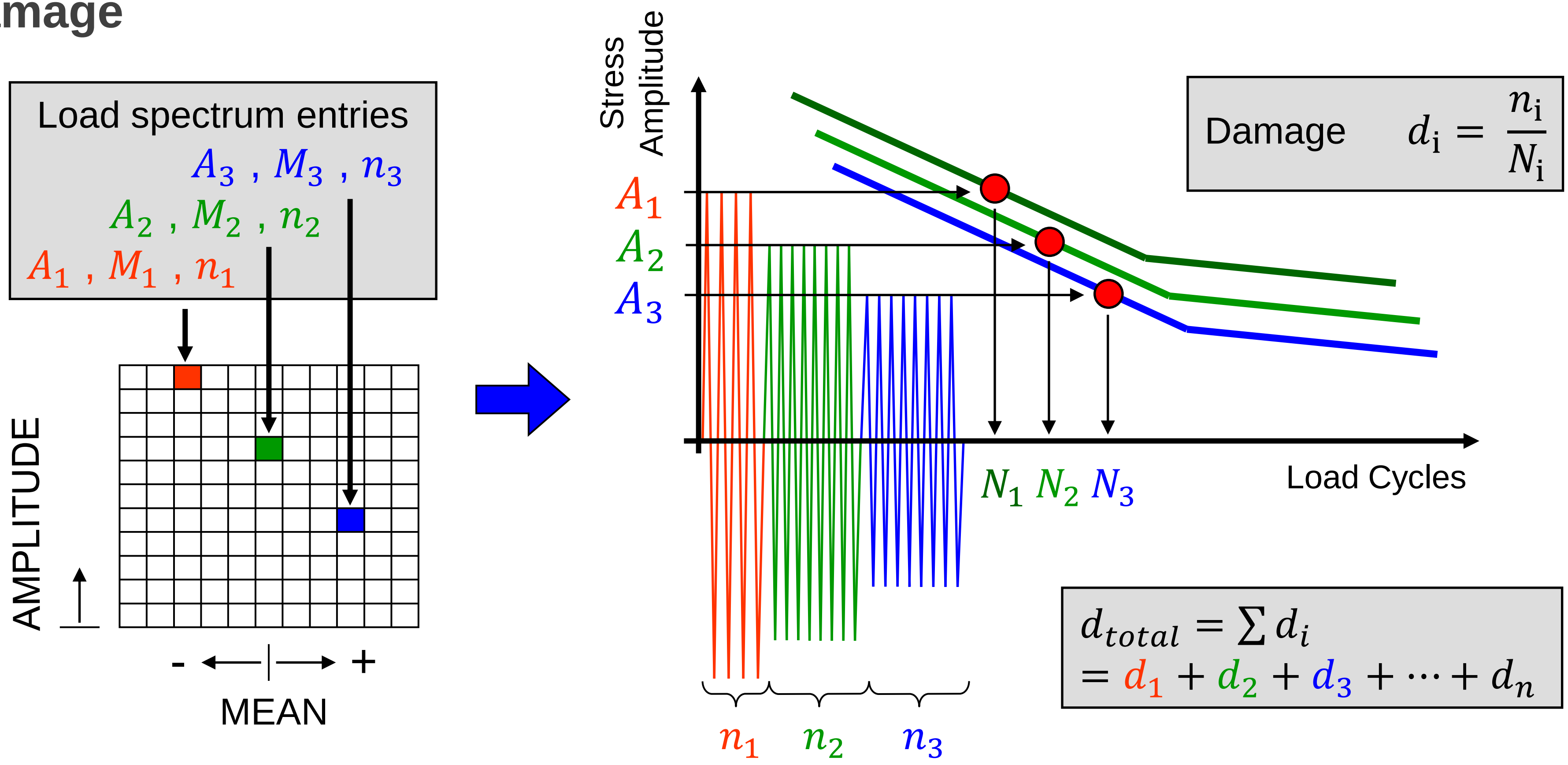
Note: Different behavior for R = const & $\sigma_{CONST} \neq 0$ between BASIC and MAX

Analysis Target

For each entry in the rainflow matrix a S/N curve is derived and a damage increment is calculated. All increments are summed up to a total damage value.

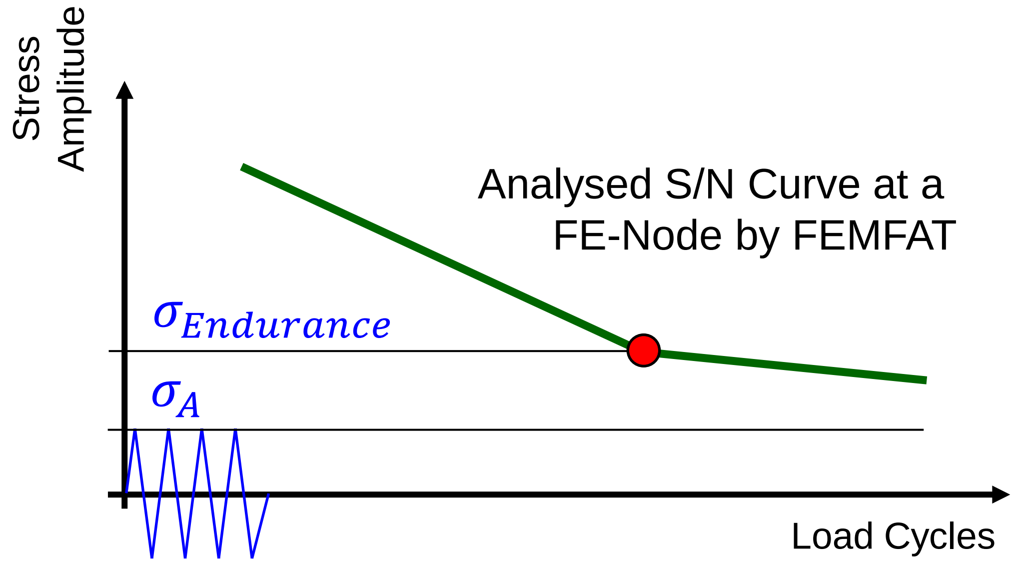
MAGNA

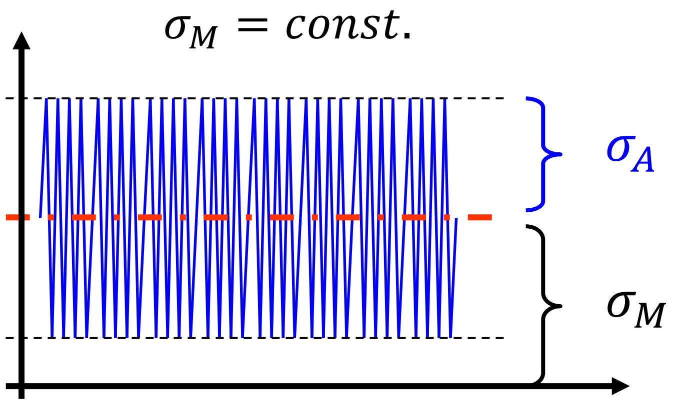
Damage



Endurance safety factor SF_A is calculated with respect to the ultimate cycle limit and the stress amplitude . Safety factor SF_B is related to the upper stress.

Endurance Safety Factor





$$SF_A = \frac{\sigma_{endurance}}{\sigma_A}$$

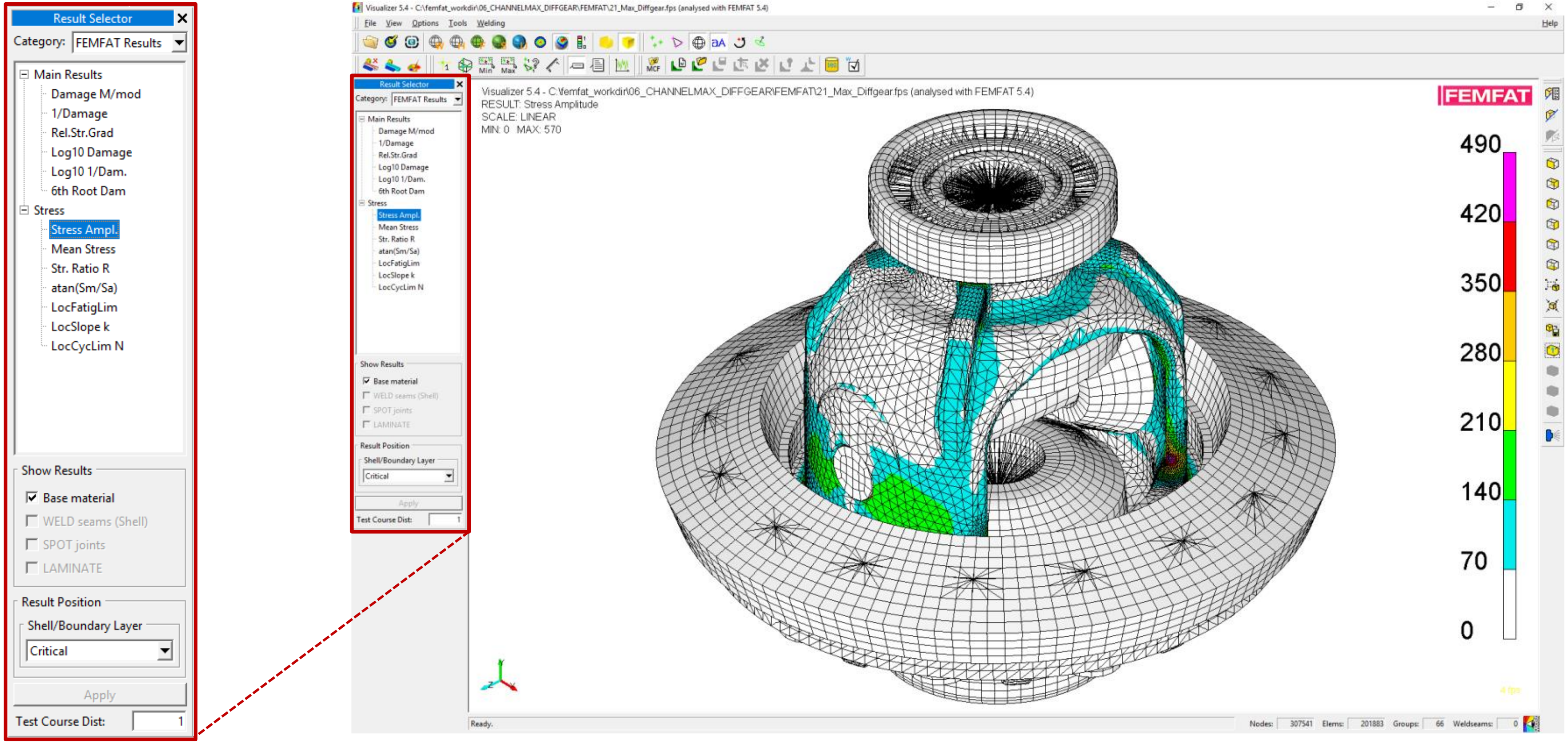
$$SF_B = \frac{\sigma_{endurance} + \sigma_M}{\sigma_A + \sigma_M}$$

FEMFAT visualizer

A fast 3D post processor for displaying the structure of FEM models, definition of weld seams and for viewing fatigue results and FEA stresses.

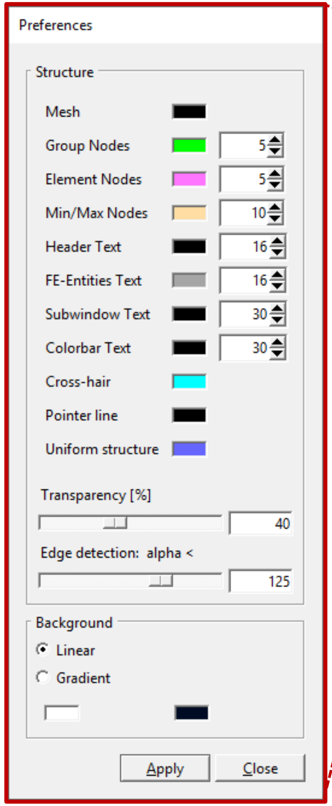
MAGNA

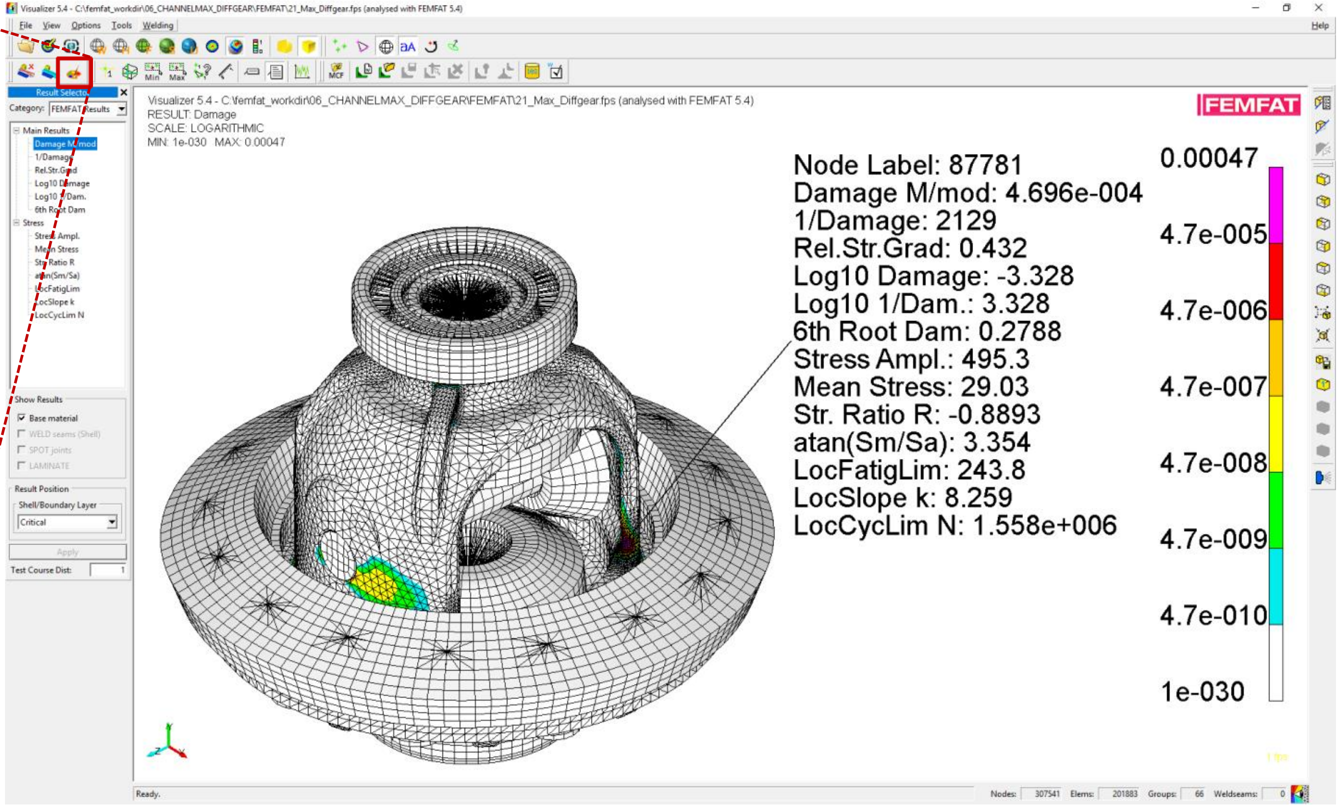
FEMFAT visualizer – Displaying FE Entities and Results



In case you need detailed information about a node for your report it‘s possible to add a subwindow with detailed information.

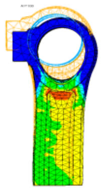
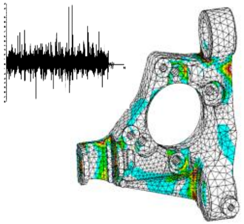
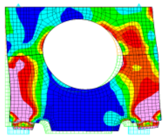
FEMFAT visualizer – Detailed node information and preferences





Analyses in Time Domain

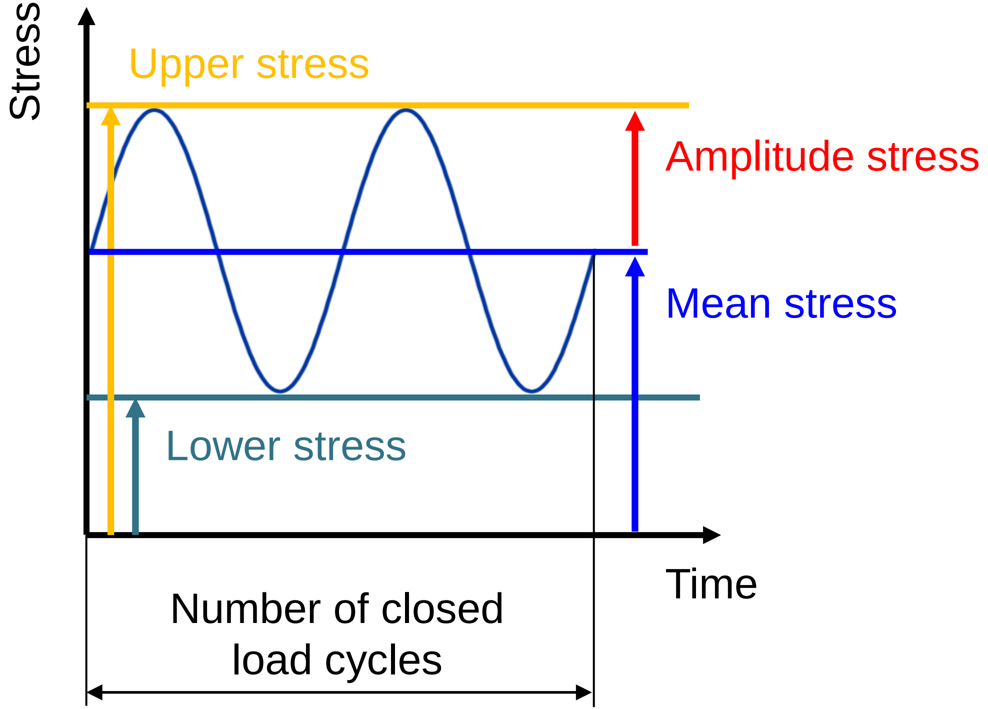
Which module to use?

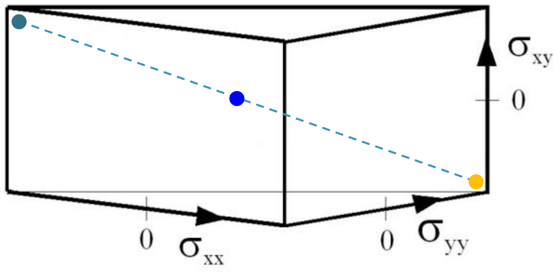
Module	When to use	Examples	What to Import
BASIC 	• Direction of principle stresses are constant • 2 load conditions • max. one constant loading	• Conrod with dominating load cases ignition and inertial force • Engine bulkhead and bearing cap (assembly and ignition) • Shafts with torque history • Matrix of combinations (railway)	• Two stress results, which can be: – Upper and lower stress or – Amplitude and mean stress • Load spectra for damage analysis
CHANNEL MAX 	• Direction of principle stresses may change permanently • Direction and location of forces and boundary conditions are constant • Existing load histories • Multiple channels which are generally not in phase	• Chassis parts like: – Knuckles, subframes, H-Arms,... – Body in White – Crankshaft with modal approach	• One stress result for each channel (e.g. for unit load) • One load history for each channel • Modal stresses from CMS and modal coordinates from MBS
TRANS MAX 	• Load application point and/or boundary conditions are altering with time • Transient sequence of stress results	• Engine bulkhead and bearing cap or crankshaft with stress result each n° crankangle • safety factor from 2 of n loading conditions	• Sequence of stress results

FEMFAT basic

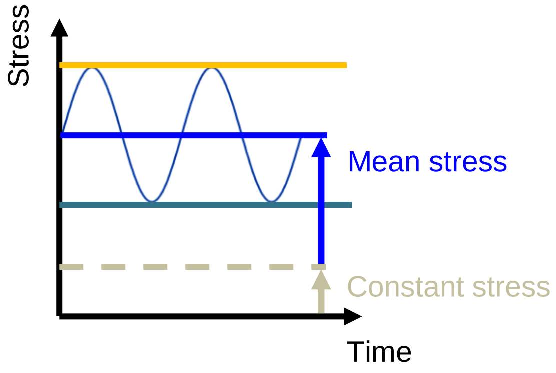
FEMFAT basic is used when the load consists of only 2 load conditions, which may include an optional constant load.

FEMFAT basic for Pure Uniaxial Loading





… with Constant Stress Data Set:



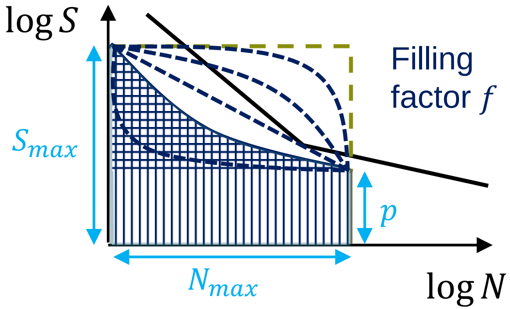
Load Spectra for damage analysis are made by scaling mean stress and stress amplitudes. Thus, the Rainflow matrix can be constructed.

Load Spectra

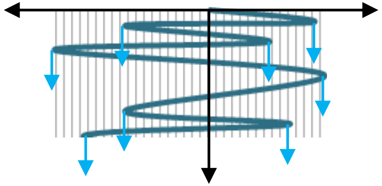
Stepped Load Spectra

Step	N	Fact.Ampl	Fact.Mean
1	500	2.0000	1.0000
2	2500	1.5000	0.5000
3	7000	0.7500	0.4000

Synthetic Load Spectra

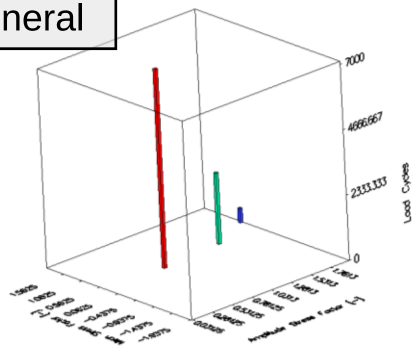


Load-Time History

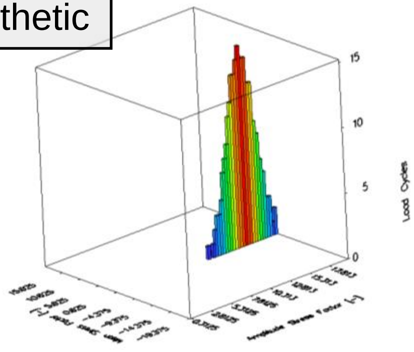


Rainflow Counting e.g. in FEMFAT LAB

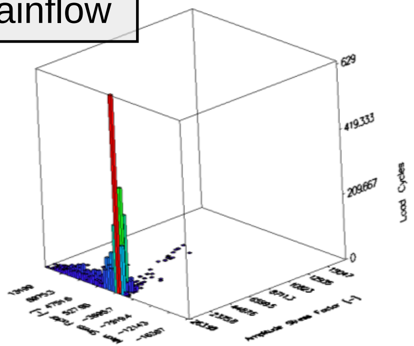
General



Synthetic



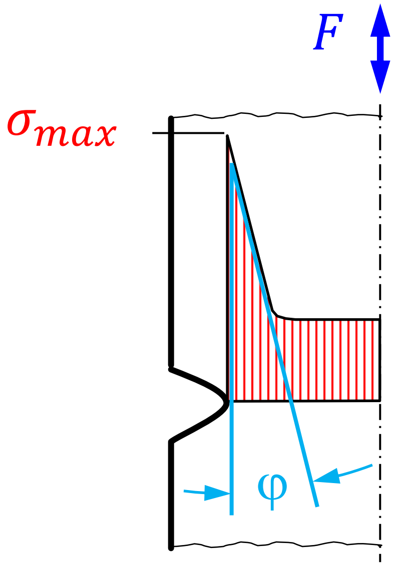
Rainflow



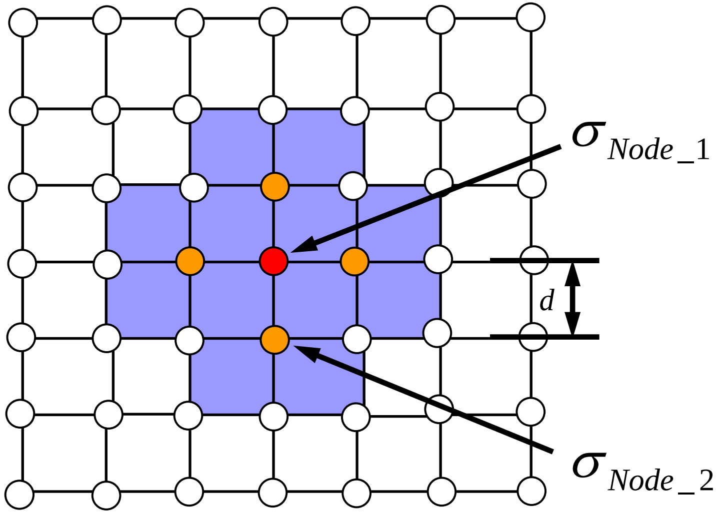
In order to calculate the gradient a minimum number of nodes around the node of interest needs to be taken into consideration

MAGNA

Relative Stress Gradient (1/2)



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



Following FE-Entities must be part of the analysis group:

- Node of Interest
- Neighbor nodes
- Elements related to the node of interest

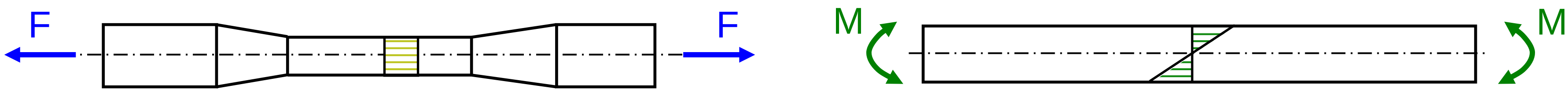
Relative Stress Gradient: $$\chi' = \frac{1}{\sigma_{Node\_1}} \left(\frac{\sigma_{Node\_1} - \sigma_{Node\_2}}{d} \right)$$

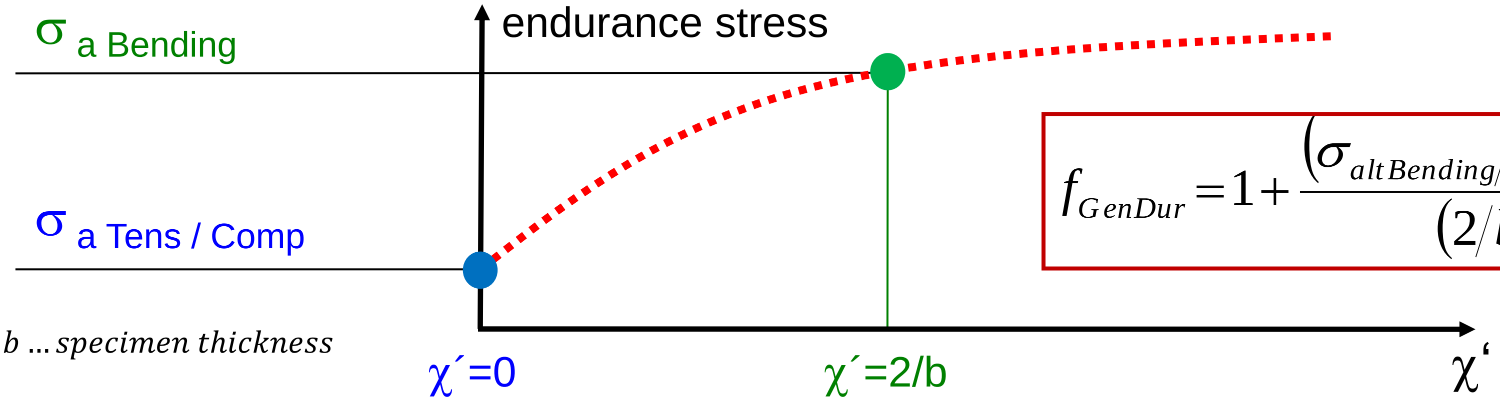
The influence of the relative stress gradient on the endurance limit is determined by experiments at different relative stress gradients and notches

Relative Stress Gradient (2/2)

Relative Stress Gradient
Tension / Compression $\chi'_{alt\,TC} = 0$

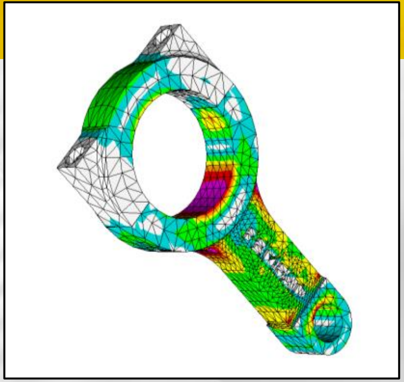
Relative Stress Gradient
Bending $\chi' = 2/b$





$$f_{GenDur} = 1 + \frac{\left(\sigma_{altBending}/\sigma_{altTC} - 1\right)}{\left(2/b\right)^{\nu}} \chi'^{\nu}$$

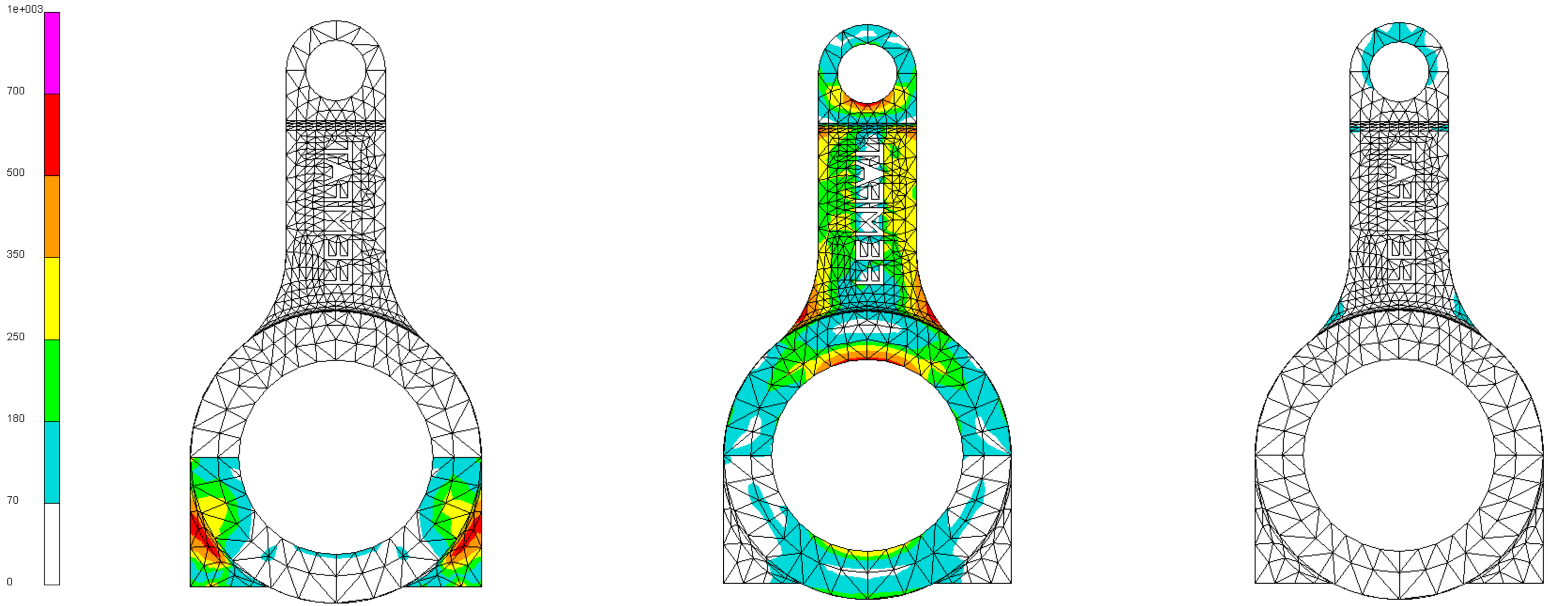
BASIC – Example



In this example a damage analysis of a connecting rod is performed. Loading is characterized by three different load cases.

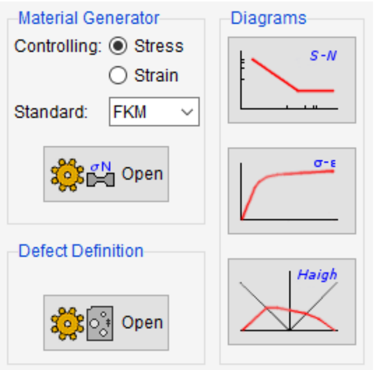
Conrod: Uniaxial analysis based on load spectra

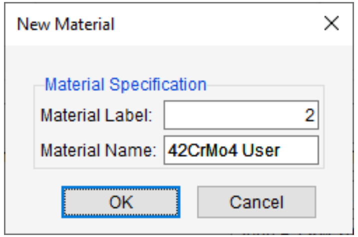
Load case 1 Stress from bolts	Load case 2 Stress from ignition	Load case 3 Stress from mass inertia
Constant	**Lower**	**Upper**



Materials that are not included in the FEMFAT database (> 450 materials) can be requested from FEMFAT support or generated by the material generator.

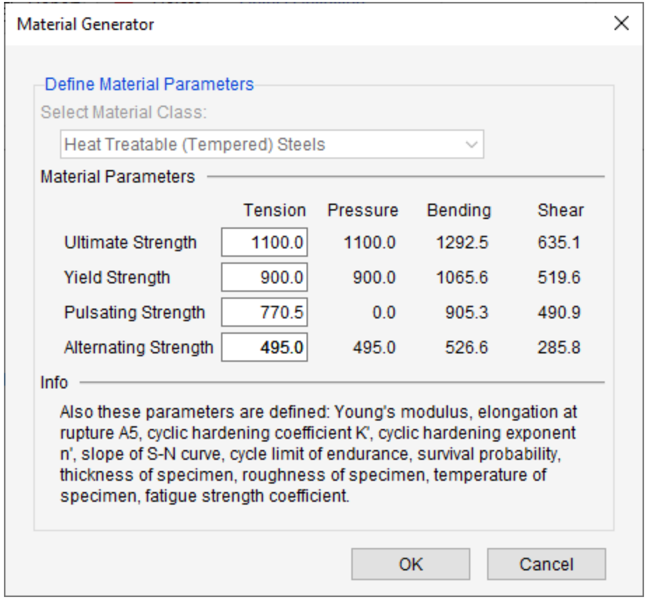
Material Data

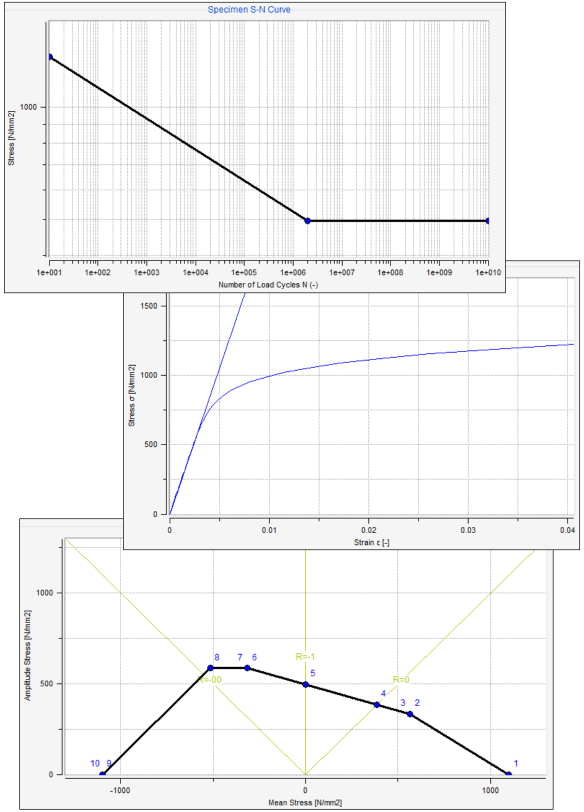




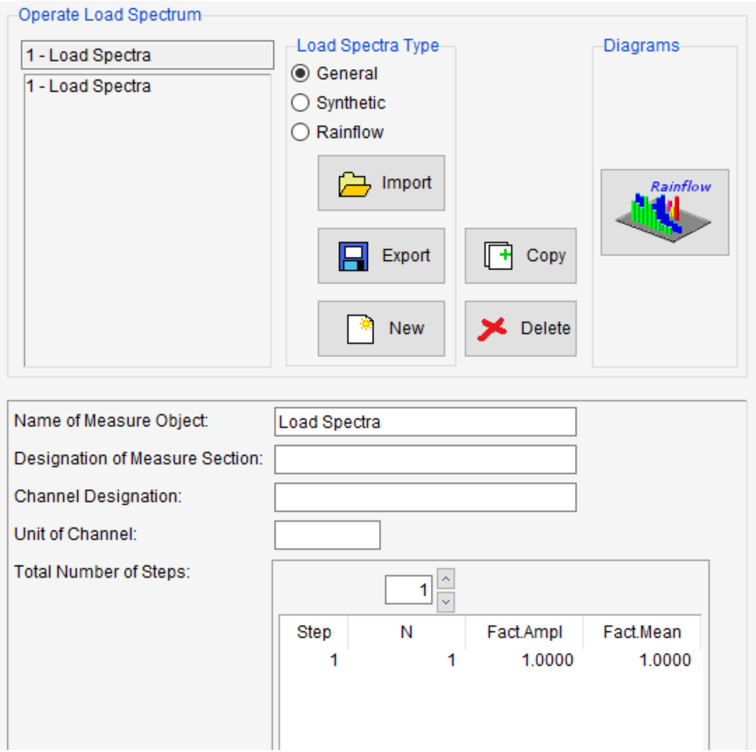
Material 42CrMo4:

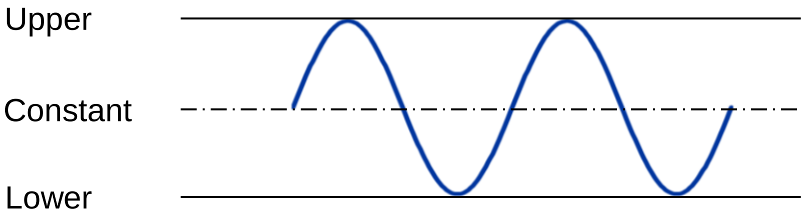
- Heat Treatable (Tempered) Steel
- $UTS = 1100 \text{N/mm}^2$
- $YTS = 900 \text{N/mm}^2$



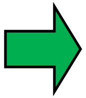


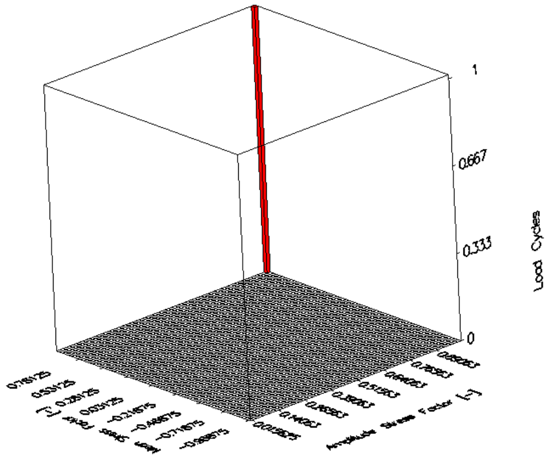
Load Spectra





Rainflow Matrix:





If your stress results come from a nonlinear FE calculation you must turn off the Mean Stress Rearrangement!

Node Characteristics & Influence Factors

Assign to Group 'ALL':

Material:	1 - 42CrMo4 User	
Surface Roughness:	Smoothed (= 60µm)	60.000 [µm]

Assign to Group 'hardening':

Material:	1 - 42CrMo4 User	
Surface Roughness:	Polished (= 2µm)	2.000 [µm]

Surface Treatment

Inductive Hardening

General Surface Treatment Factor:	1.000
Forge Influence Factor:	1.000

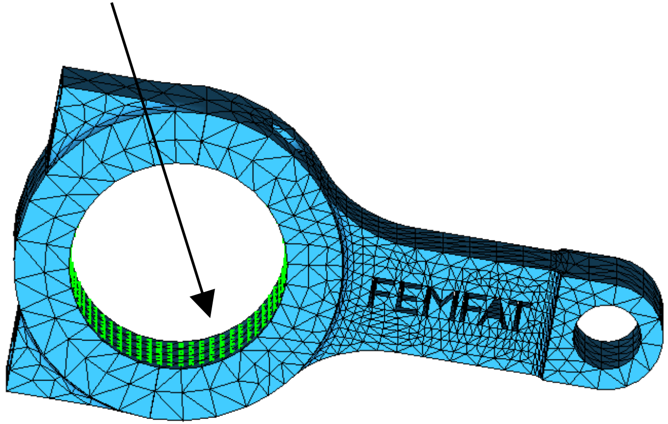
Activate Influence Factors:

☑ Surface Roughness	FKM / IABG (Rz)
☑ Constant Stresses	FEMFAT 2.0
☑ Inductive Hardening	FKM

Influences on fatigue strength:

- Inductive Hardening
- Surface Roughness
  $R_z = 2\mu m$ (polished)

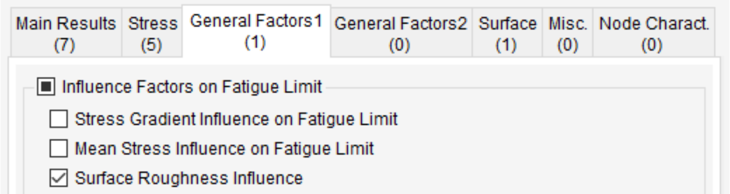
} acc. to FKM

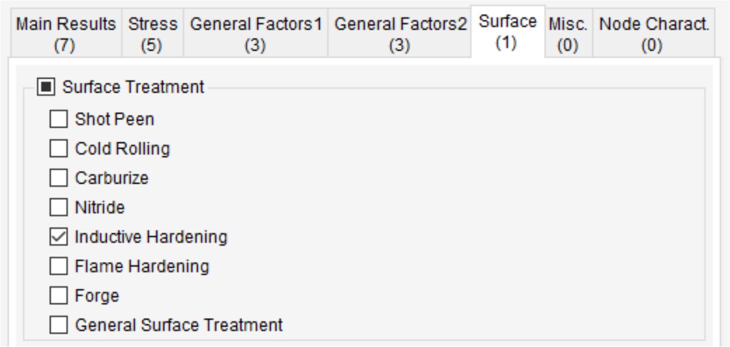


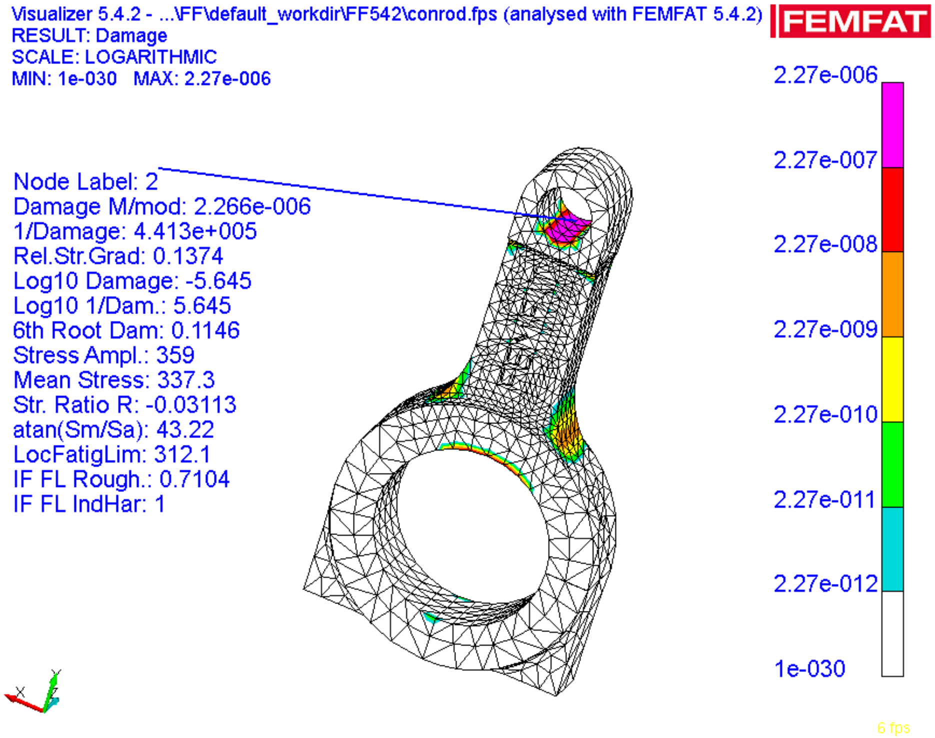
The FEMFAT visualizer is a post-processor for displaying analysis results. In addition, the welds and groups used in the analysis can be defined here.

Output & Visualization

Output of Influence Factors:







FEMFAT max

MAGNA

FEMFAT max for Multiaxial Loading

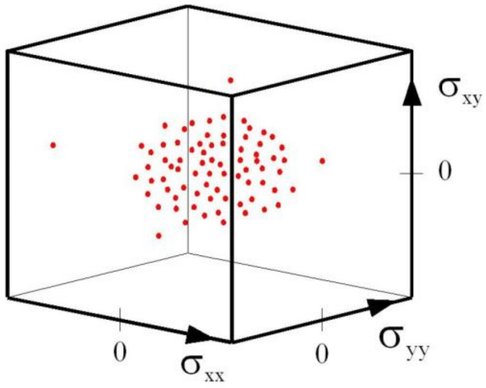
Time steps: Stress tensor at FE node:

0 $$\begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix}_0$$

1 $$\begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix}_1$$

...

m $$\begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix}_m$$



Planar stress:

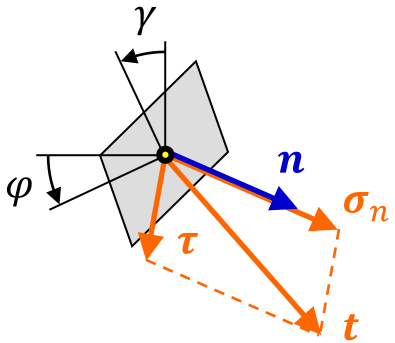
The number of time steps in the load history corresponds to the points in the 3D stress space (σ_{xx}, σ_{yy}, σ_{xy}).

Multiaxial loads:

- are composed of at least 3 time points and
- are characterized by the fact that the points do not lie on a straight line in the 3D stress space.

Criterion of critical cutting plane in FEMFAT max

1) Transformation of all stress tensors into several planes with normal vector $\boldsymbol{n}$
2) Filtering of interesting planes
3) Generation of the load histories of the stress components
4) Rainflow counting in all selected planes
5) Stress Rearrangement (FEMFAT plast)
6) Damage analysis (Influence Parameter Method)
7) The cutting plane with maximum damage is assumed to be the critical plane for fatigue failure



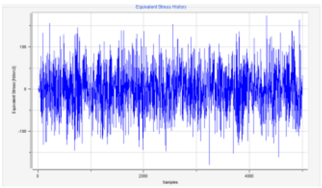
$$\boldsymbol{t}_0 = \begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix}_0 \cdot \boldsymbol{n}$$

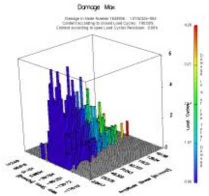
$$\boldsymbol{t}_1 = \begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix}_1 \cdot \boldsymbol{n}$$

$$\dots$$

$$\boldsymbol{t}_m = \begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix}_m \cdot \boldsymbol{n}$$

$\boldsymbol{\sigma}_n$

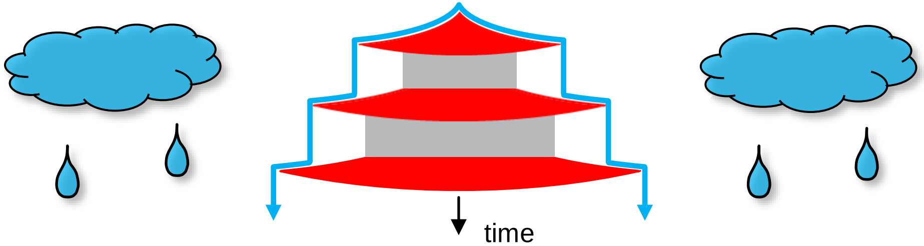


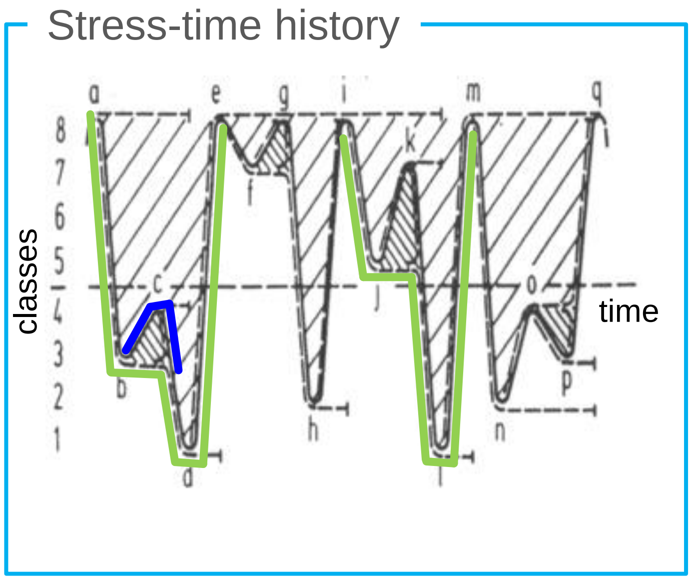


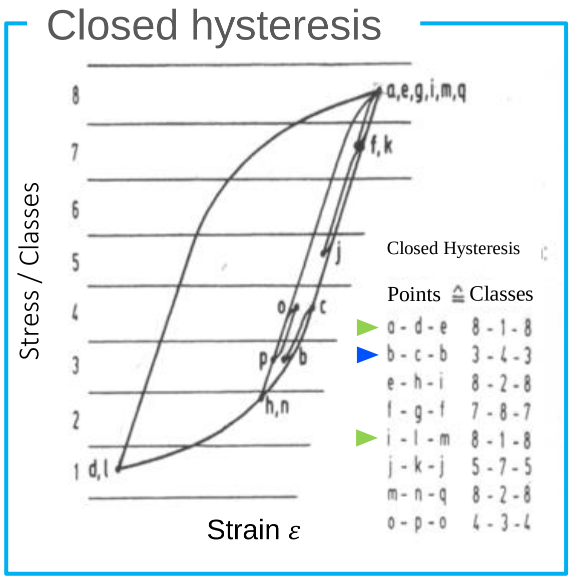
Rainflow counting is a two-parameter procedure. The full stress history must be reflected in the rainflow matrix by counting the closed hysteresis.

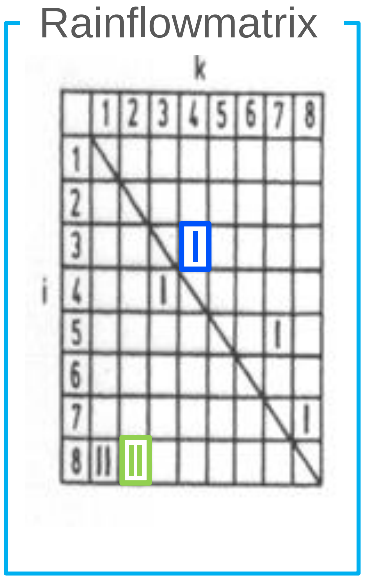
Rainflow counting

"Rainwater flows over pagoda roofs"







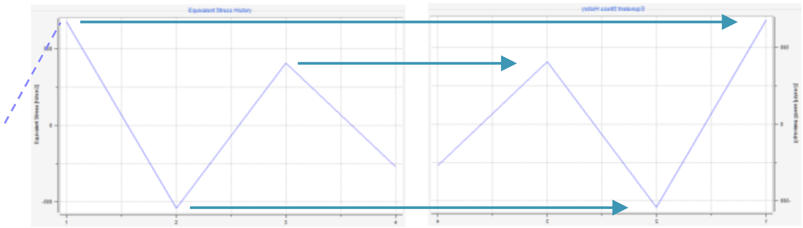


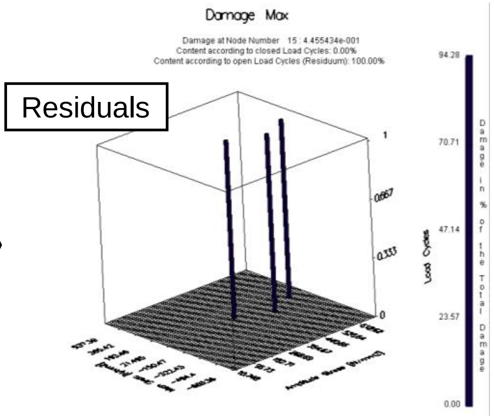
In FEMFAT basic a Load Spectra is used. In FEMFAT max, the local stress history is Rainflow counted, whereby two methods are available.

Rainflow counting methods in FEMFAT max

method 5.0

Results in closed & open hysteresis. (= residuals).

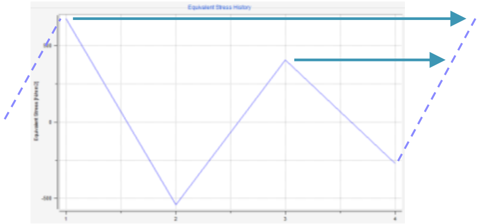


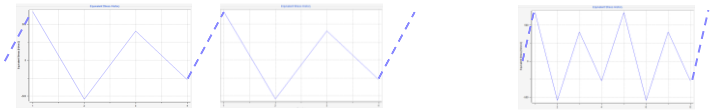


method 5.1

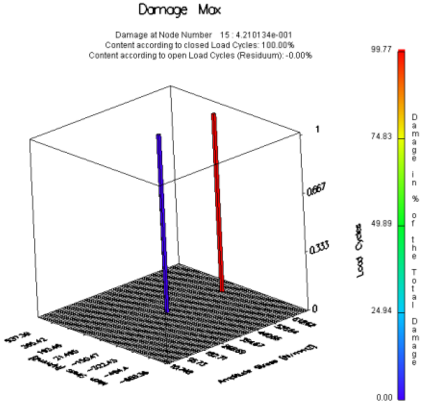
Results in closed hysteresis only.

Default





$$D_1 + D_1 = D_2$$



channelMAX

Channel based modeling requires the definition of unit loadcases for each loading direction.

MAGNA

Data processing in channelMAX

Stresses from Unit Loadcase

Load-Time History

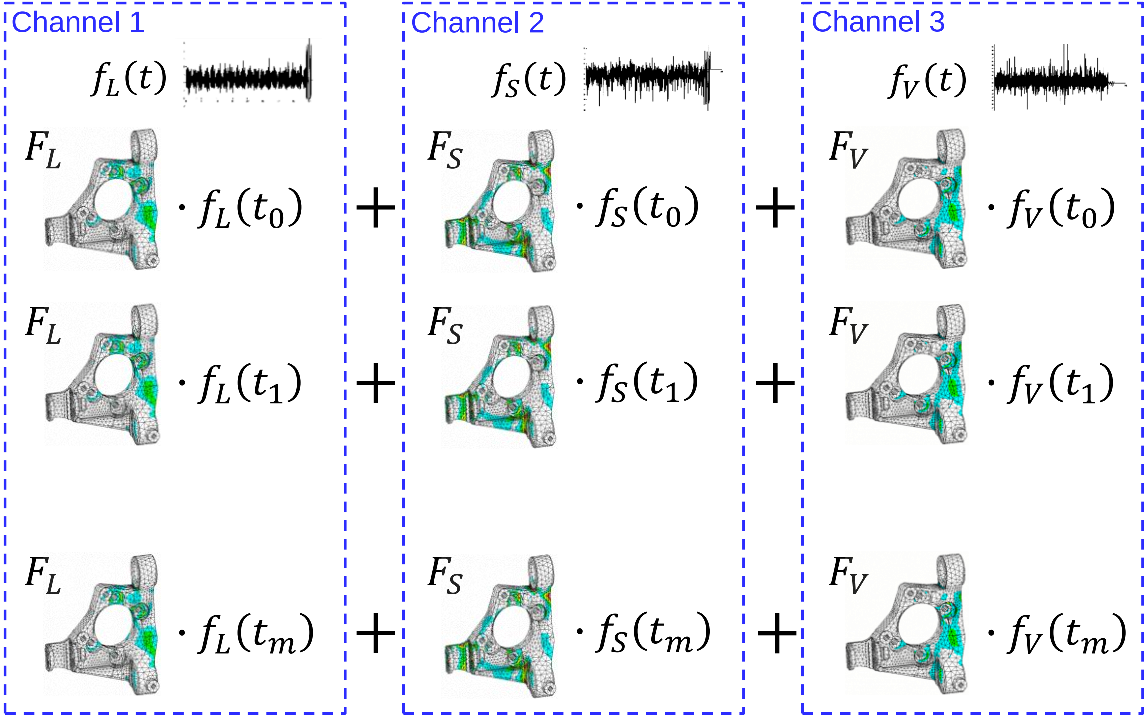
Time steps: | Stress tensor at FE node:

0: $\begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix}_0 \leftarrow$

1: $\begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix}_1 \leftarrow$

$\dots$

m: $\begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix}_m \leftarrow$

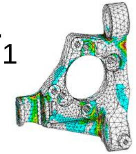
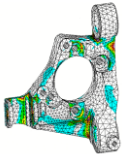


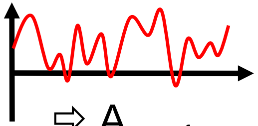
In five steps a weighted stress gradient is calculated over all channels. The weighting factor is the maximum stress amplitude per channel.

Average Relative Stress Gradient

1. Imported stress distribution
2. Gradient calculated for each channel in each node
3. Finding maximum in each force history
4. Calculation of weightening factor
5. Calculation of weighted stress-gradient

Channel 1

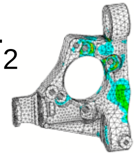
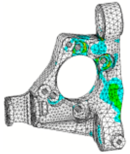
σ_1  ⇨ χ'_1 

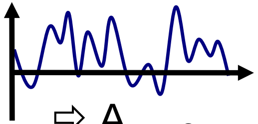


⇨ A_{max1}

$\sigma_{max1} = A_{max1} \cdot \sigma_{Mises1}$

Channel 2

σ_2  ⇨ χ'_2 

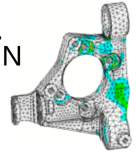
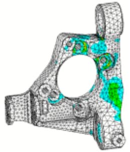


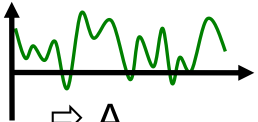
⇨ A_{max2}

$\sigma_{max2} = A_{max2} \cdot \sigma_{Mises2}$

⋮

Channel N

σ_N  ⇨ χ'_N 



⇨ A_{maxN}

$\sigma_{maxN} = A_{maxN} \cdot \sigma_{MisesN}$

$$\overline{\chi}' = \frac{\sum_{i=1}^{N} \sigma_{max\,i} \cdot \chi'_i}{\sum_{i=1}^{N} \sigma_{max\,i}}$$

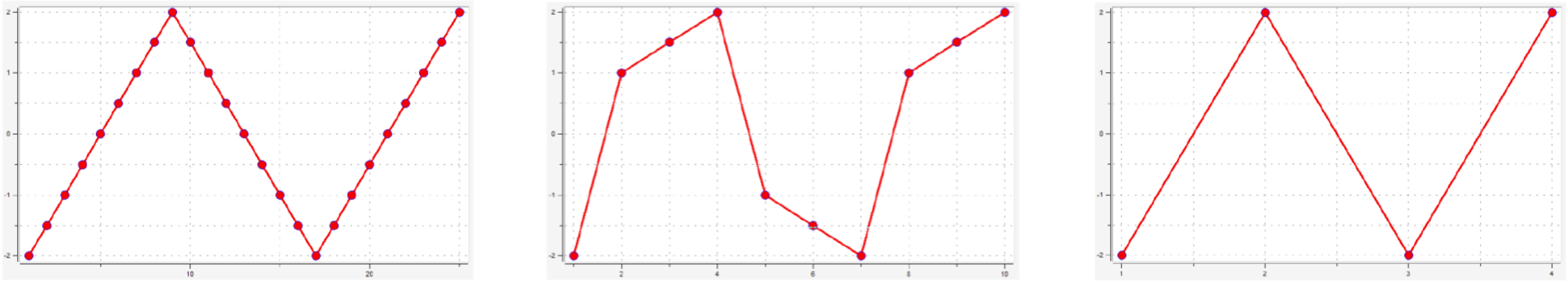
N...Number of channels

Data reduction by compressing the load history is possible. It's possible to omit intermediate points or small partial cycles.

Compression of load-time histories

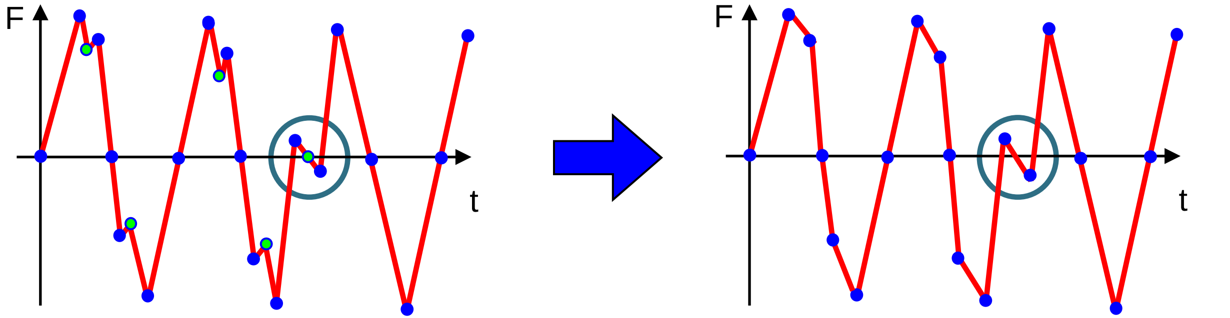
In this case all three load histories are equivalent

Disregarding Intermediate points



Neglect small cycles ⇨ smoothing

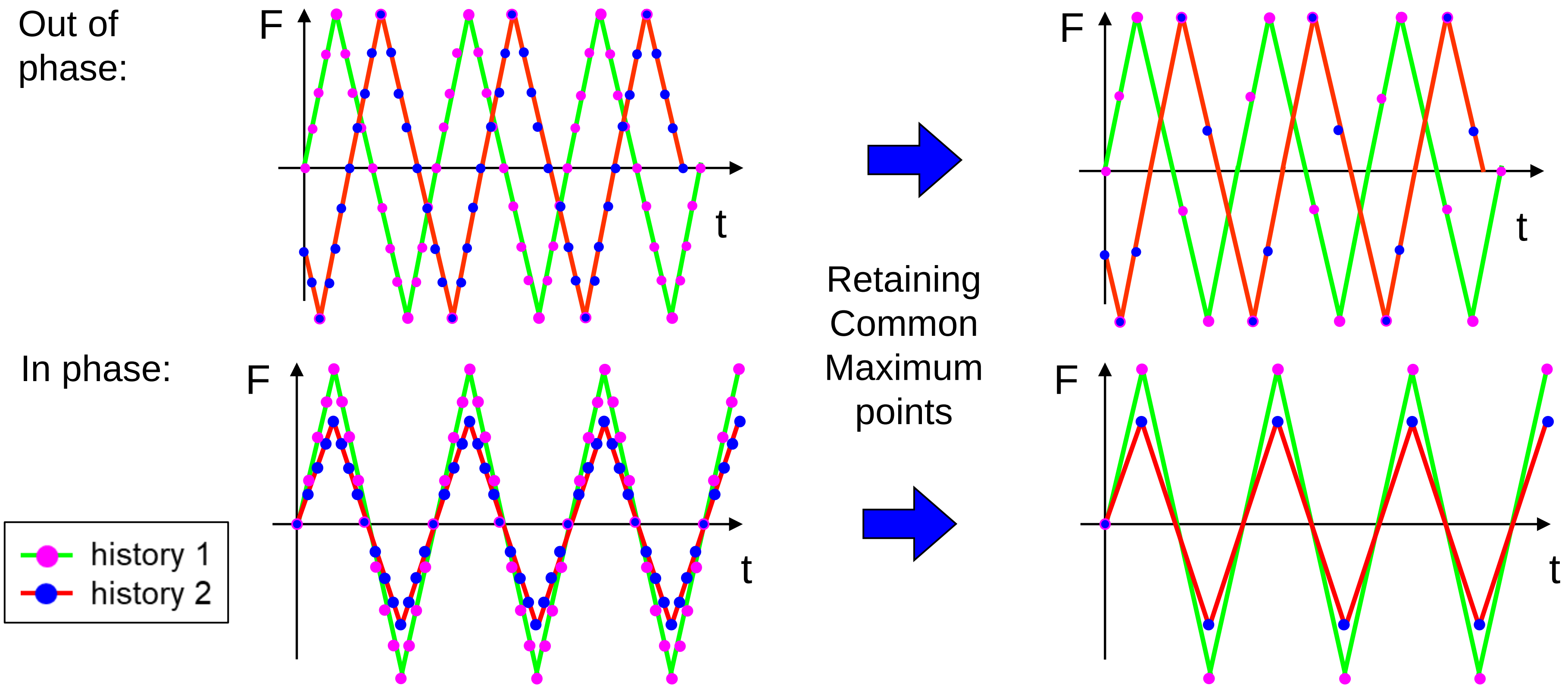
Disregarding small partial cycles



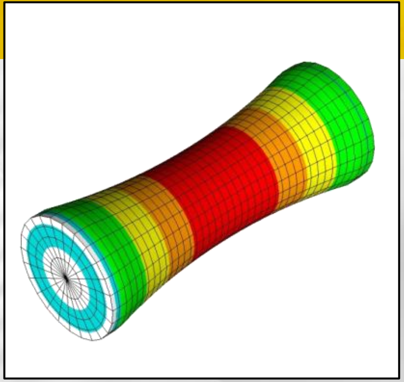
- Disregarding small cycles (multiaxial scanning)
- Weighting of loading histories according stress level!

Each time no channel has a peak points are omitted. As soon as one single channel has a peak, the point is taken into consideration.

Compression of load-time histories - Disregarding intermediate points



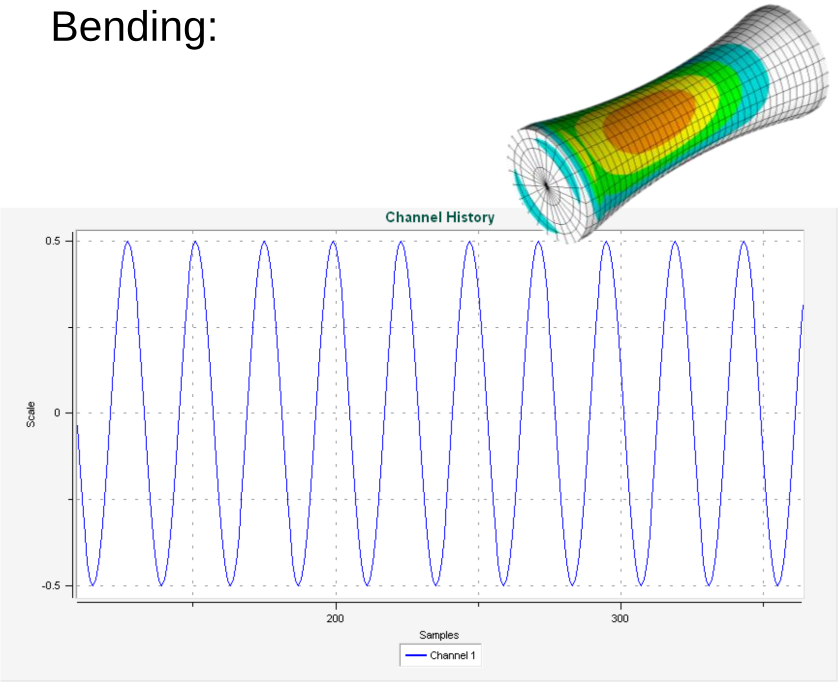
channelMAX – Example



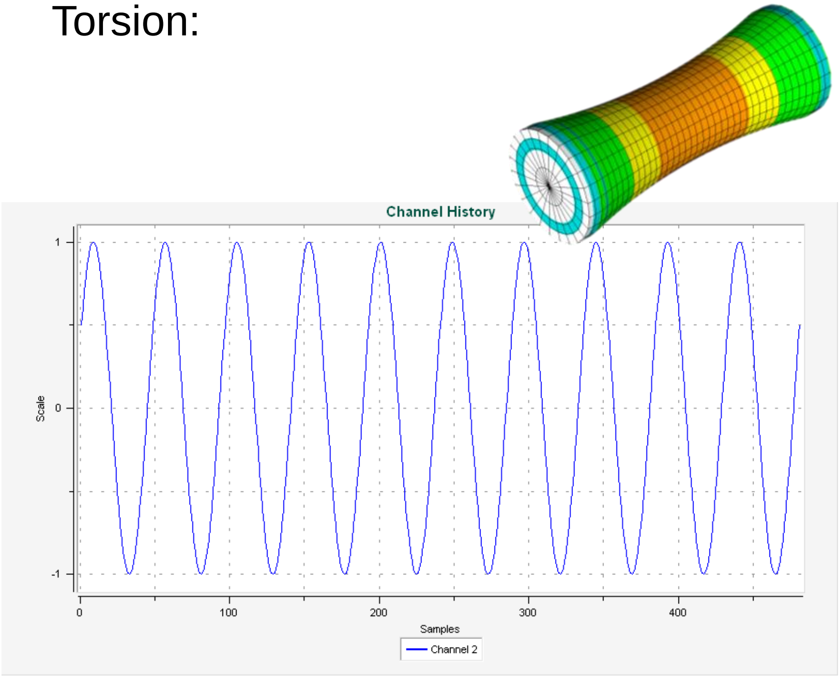
In this example, a multiaxial fatigue analysis is performed for a drive shaft. The unit load cases are combined with the corresponding load histories.

Drive Shaft: Multiaxial analysis based on unit loads & load-time histories

Bending:

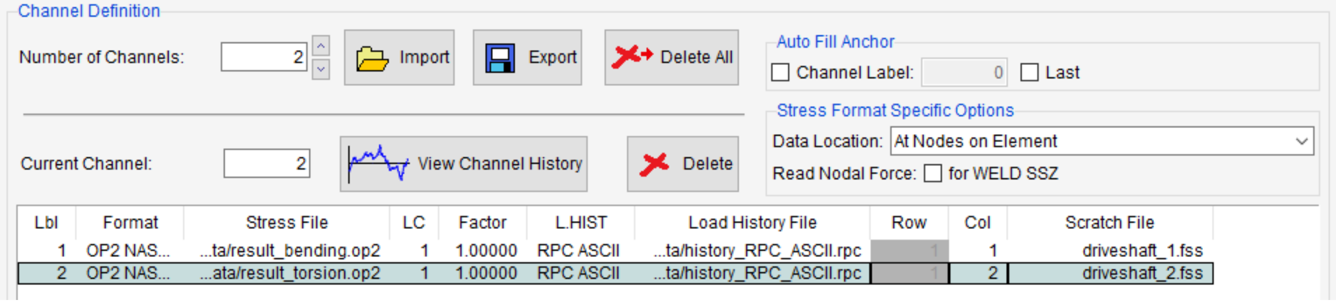


Torsion:



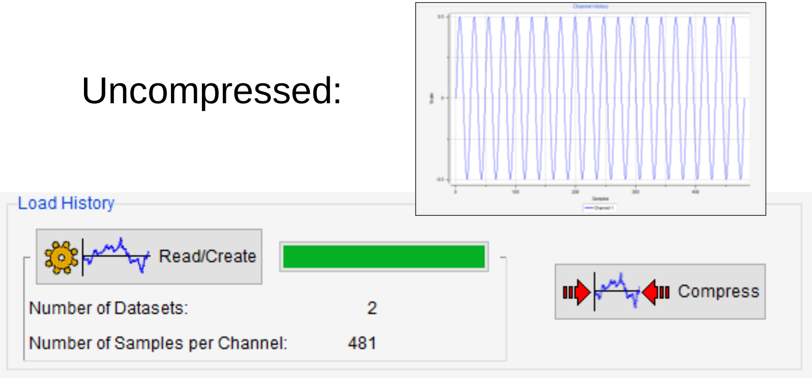
The compressed load time histories can also be exported and imported again.

Channels

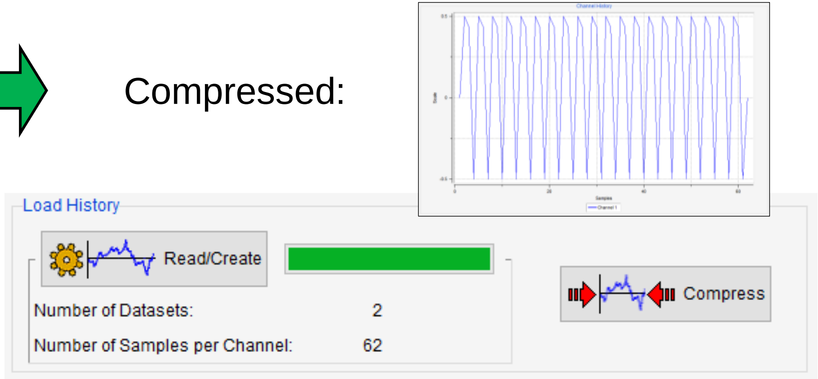


Channel Definition

Uncompressed:

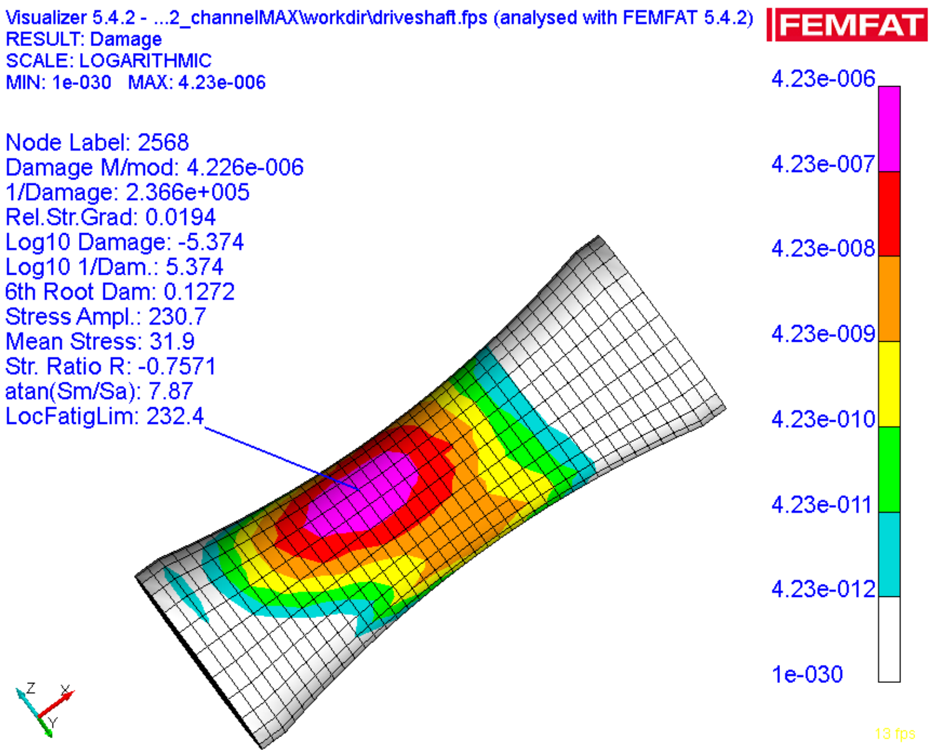


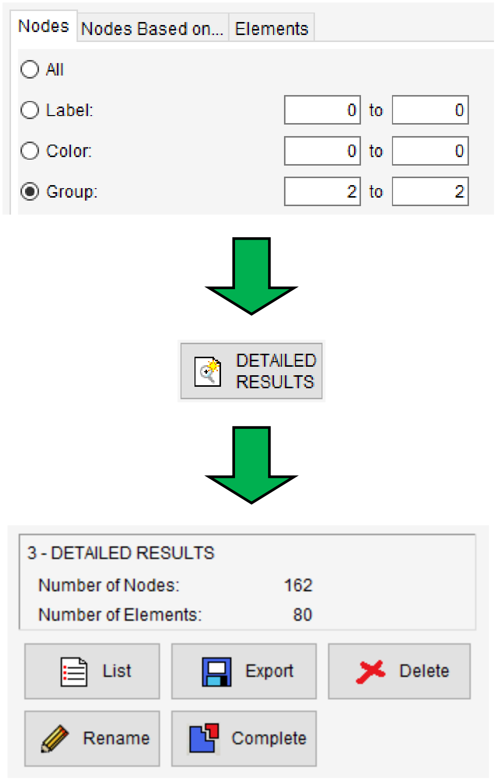
Compressed:



To save memory, detailed results are usually only exported for critical areas. For this purpose, an Analysis Group named 'DETAILED RESULTS' is defined.

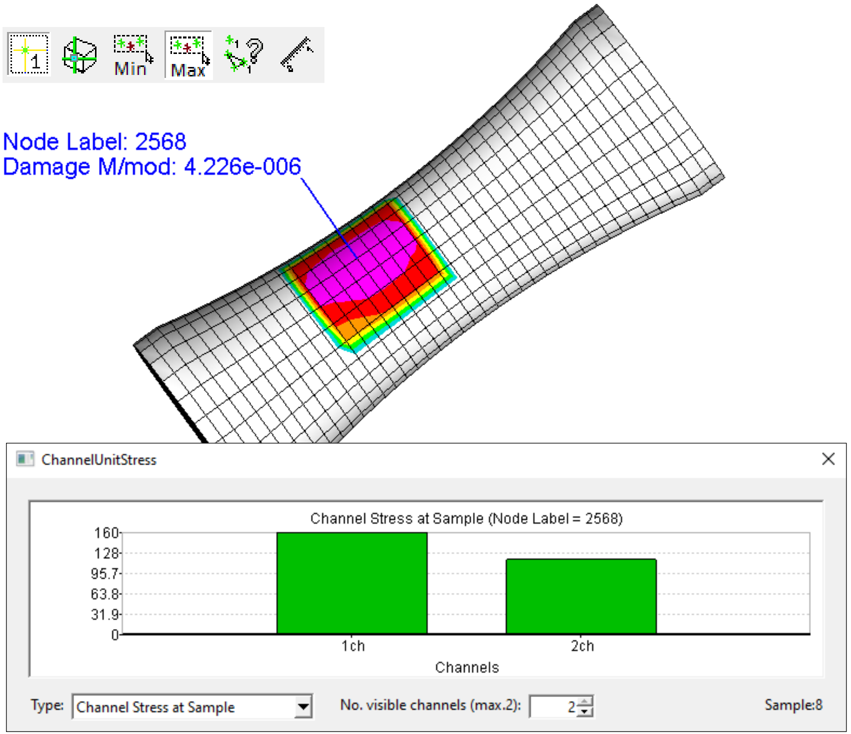
Visualization & Detailed Results Group

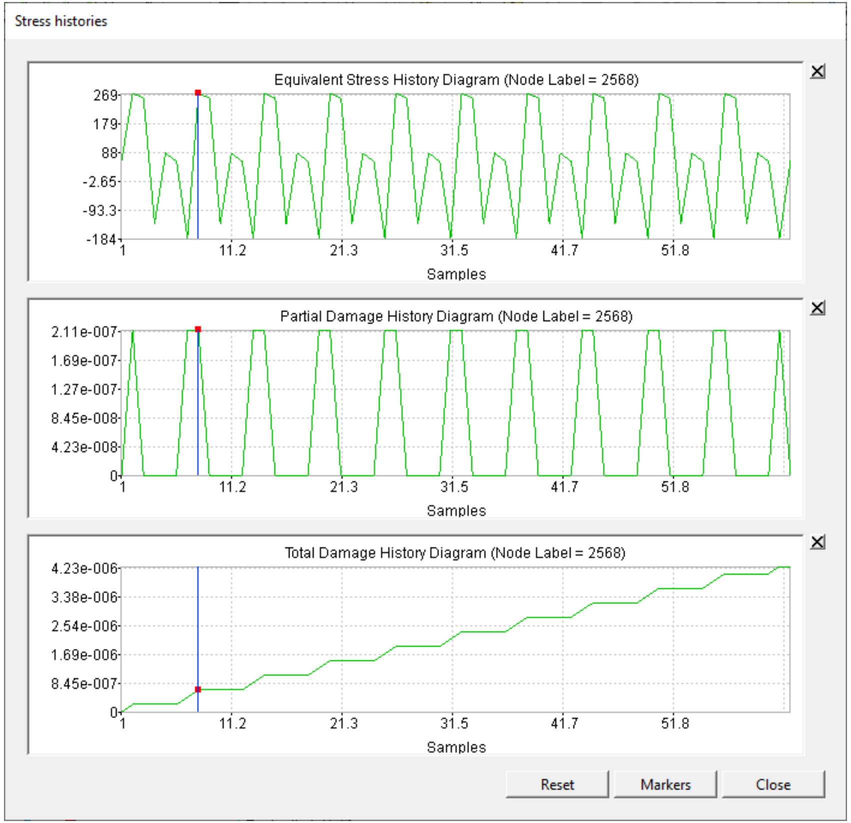




The detailed result output can be defined in the module-specific output menu.
CSV and RPC binary format is also available.

Detailed Results

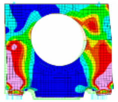


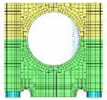


transMAX

A sequence of stress results must be defined for transient analyses. Applications are alternating loads or non-linearities (material, geometry, temperature).

Data processing in transMAX


Stresses at time


Load Case

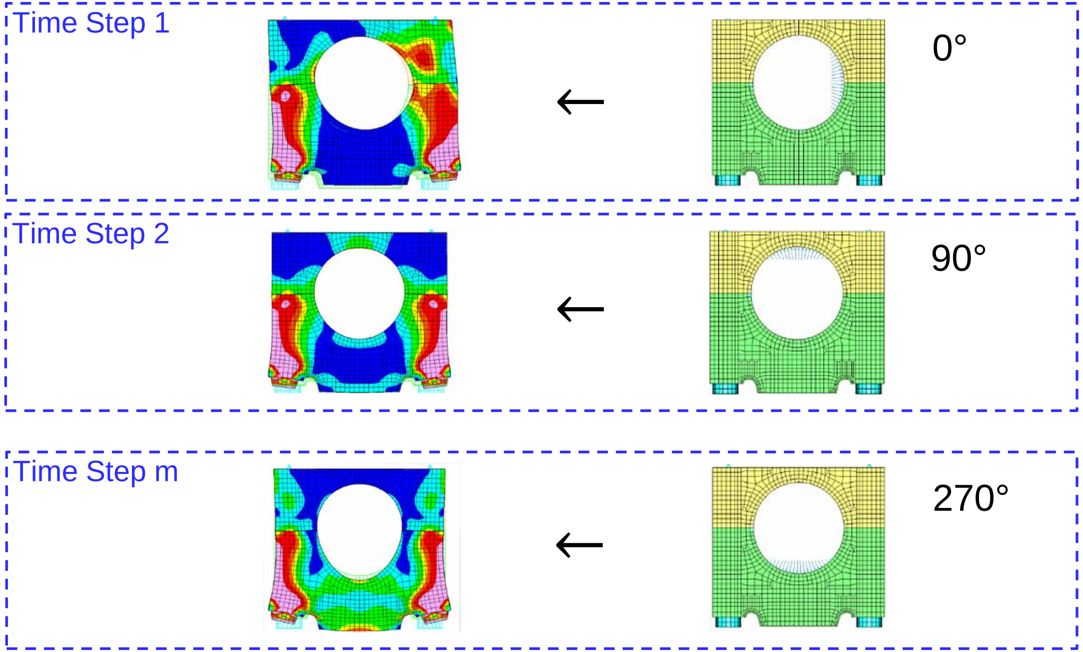
Time steps: | Stress tensor at FE node:

$$0: \quad \begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix}_0 \leftarrow$$

$$1: \quad \begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix}_1 \leftarrow$$

$$\dots$$

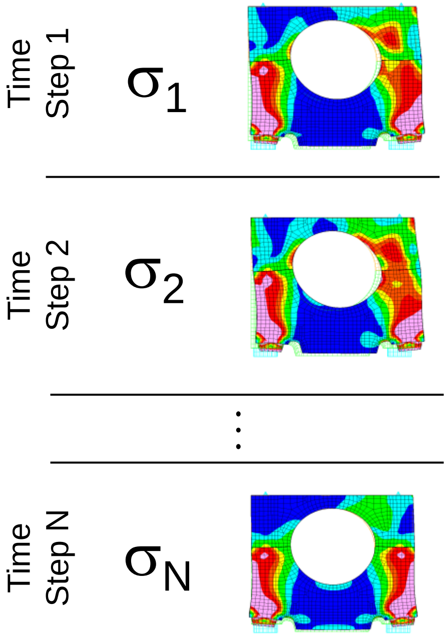
$$m: \quad \begin{pmatrix} \sigma_{xx} & \sigma_{xy} & \sigma_{xz} \\ \sigma_{yx} & \sigma_{yy} & \sigma_{yz} \\ \sigma_{zx} & \sigma_{zy} & \sigma_{zz} \end{pmatrix}_m \leftarrow$$



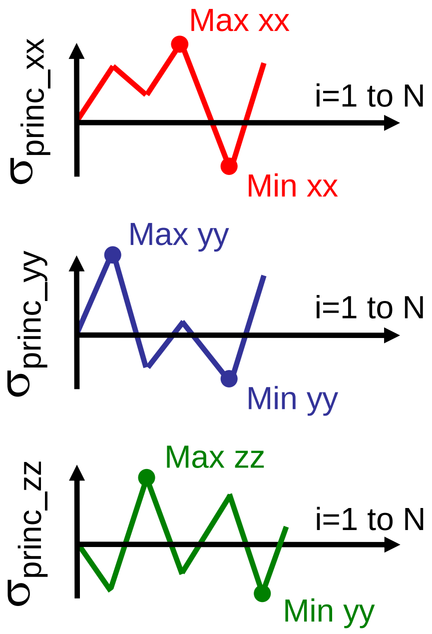
FEMFAT searches the extreme principal stresses. The maximum difference is relevant for relative stress gradient.

Relative Stress Gradient

1 Imported stresses are added



2 Calculation of principal stress histories



3 Find 3 σ_{Max} and 3 σ_{Min}, one for each principal stress history

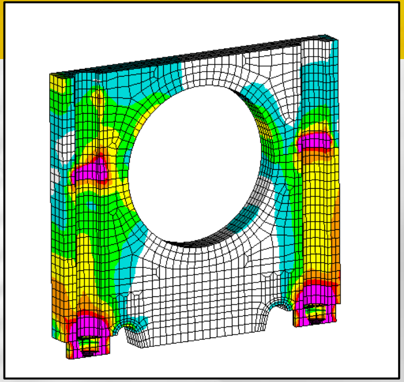
4 From nine possible combinations of 3 σ_{Max} and 3 σ_{Min} …

5 … calculate Nine „Difference-Tensors“ $\Delta\sigma = \sigma_{Max} - \sigma_{Min}$

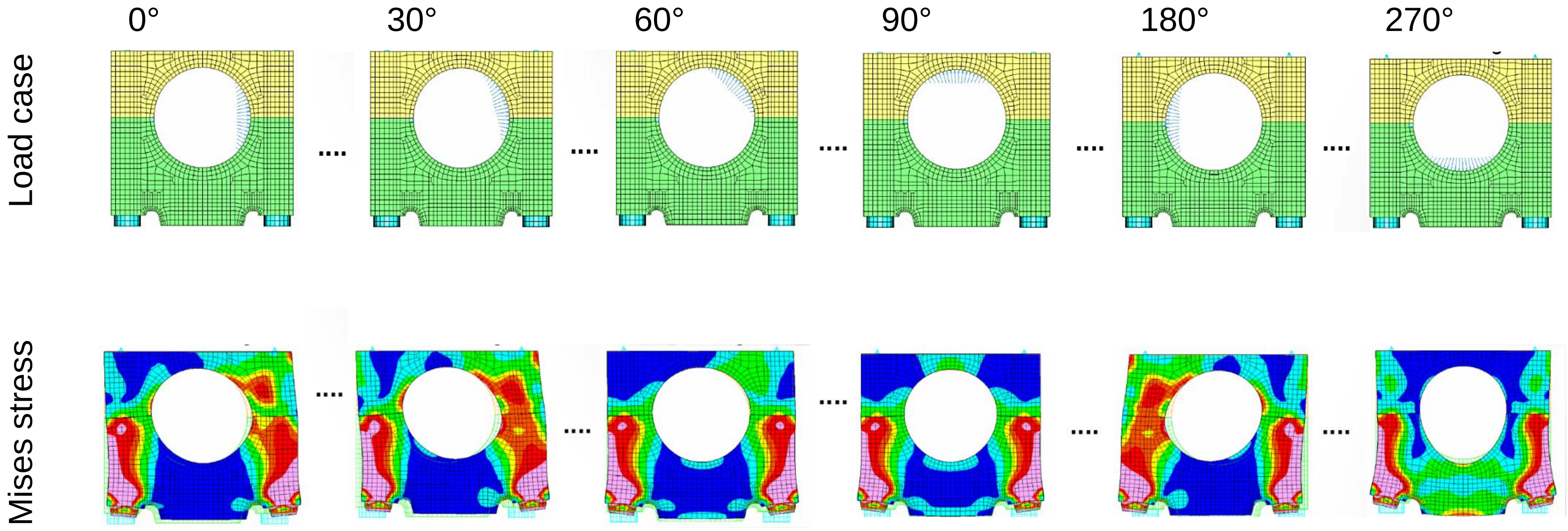
6 Calculation of nine Mises stress amplitudes

Gradient is calculated for the two time steps that cause the biggest Mises stress amplitude.

transMAX – Example

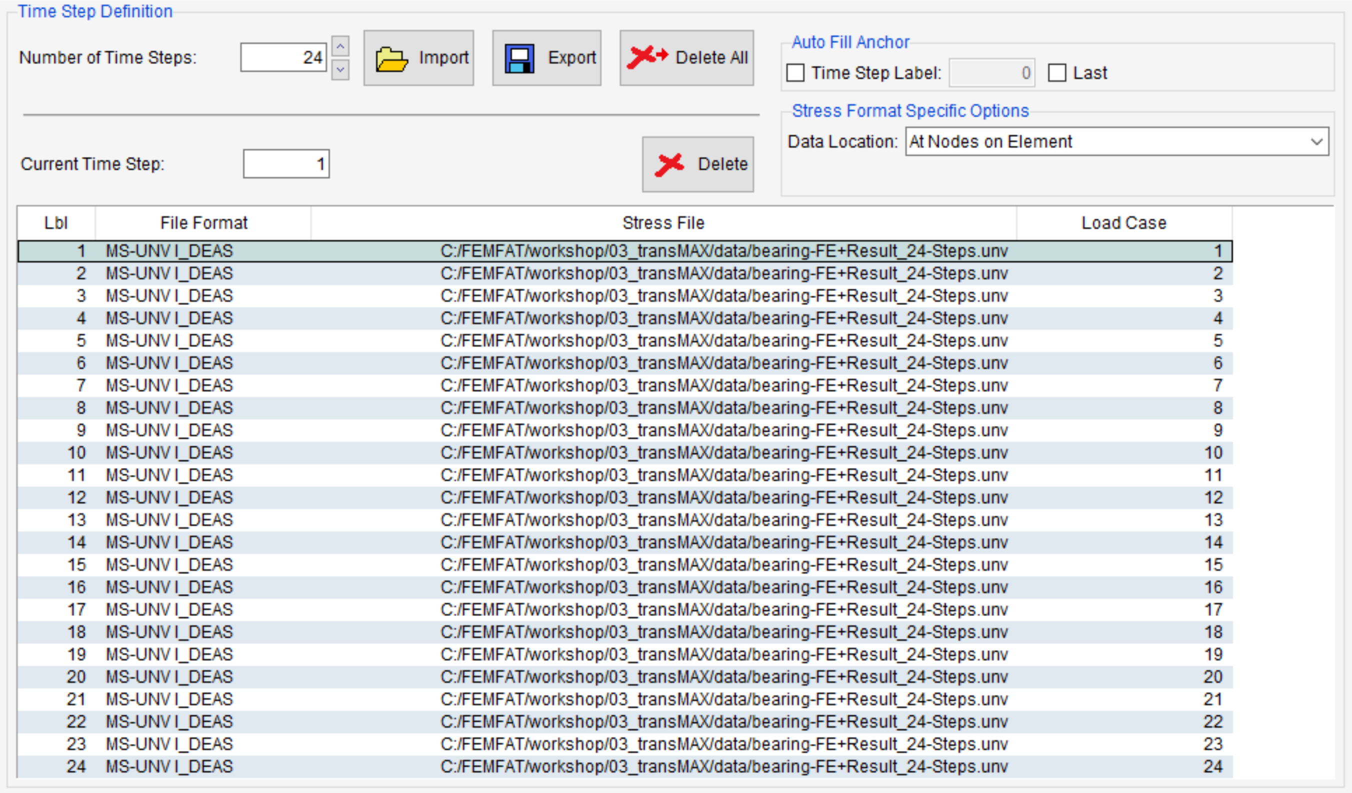


Bulkhead & Bearing Cap: Multiaxial analysis of transient stress states



In FE-Analysis is applied every 15° (360°/15°= 24 steps).

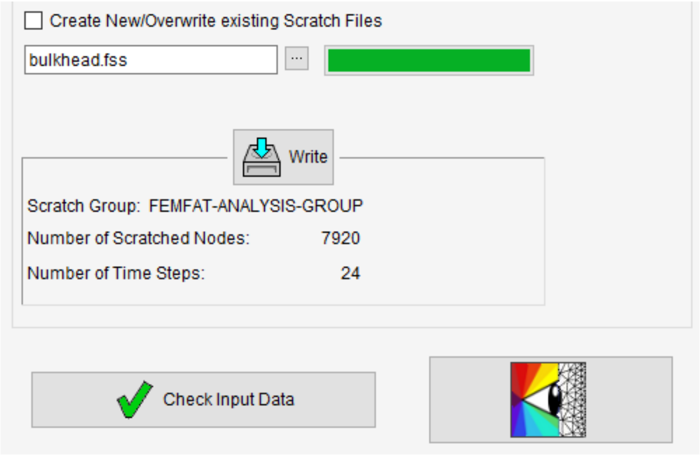
Time Steps

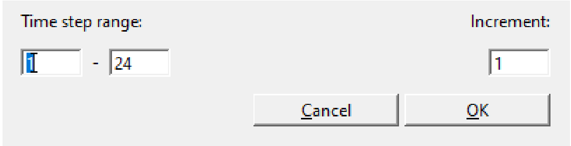


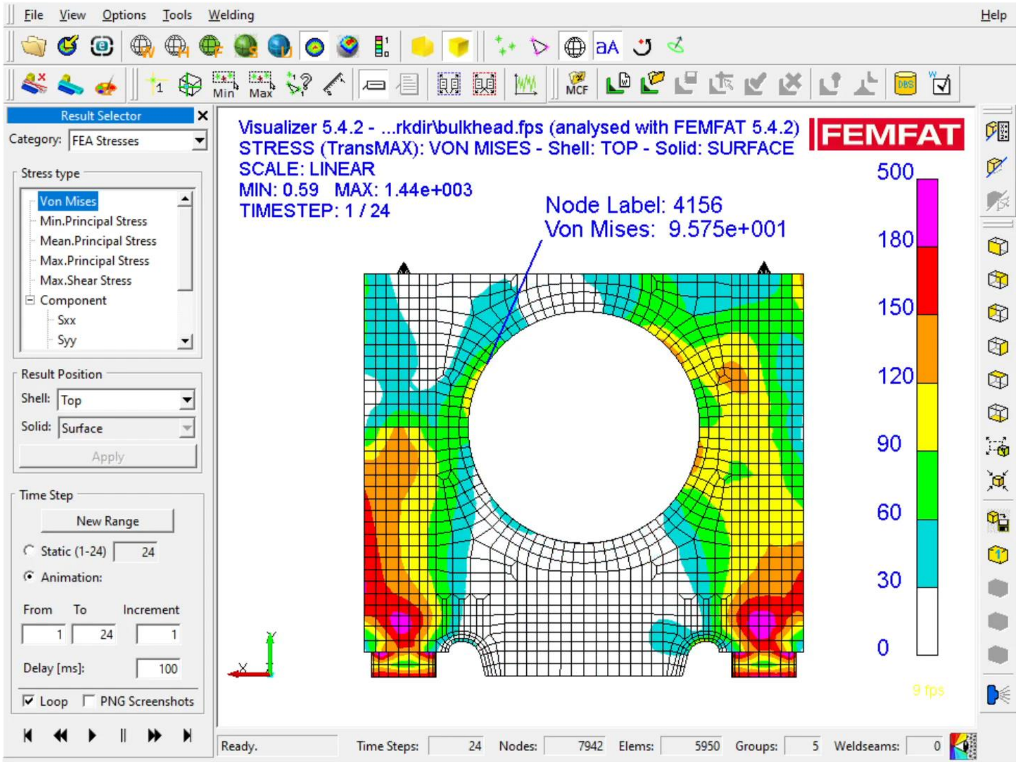
Lbl	File Format	Stress File	Load Case
1	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	1
2	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	2
3	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	3
4	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	4
5	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	5
6	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	6
7	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	7
8	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	8
9	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	9
10	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	10
11	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	11
12	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	12
13	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	13
14	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	14
15	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	15
16	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	16
17	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	17
18	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	18
19	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	19
20	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	20
21	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	21
22	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	22
23	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	23
24	MS-UNV I_DEAS	C:/FEMFAT/workshop/03_transMAX/data/bearing-FE+Result_24-Steps.unv	24

Time Step Definition

Scratch & Animation of Stresses

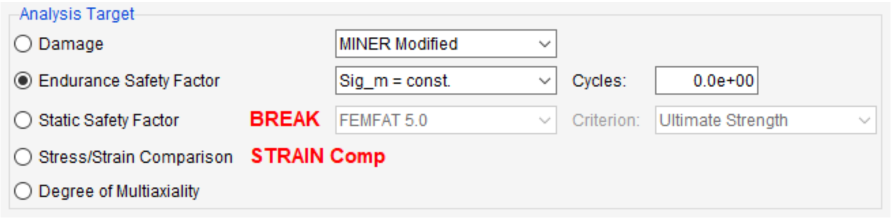


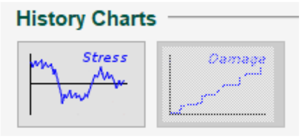


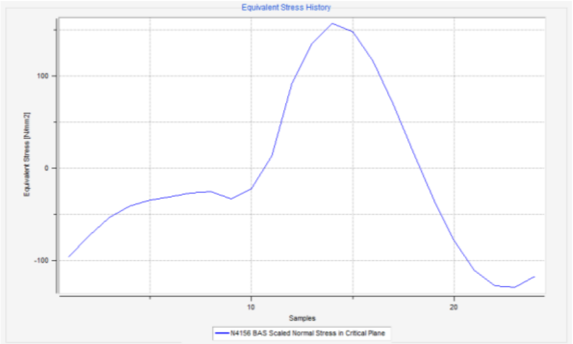


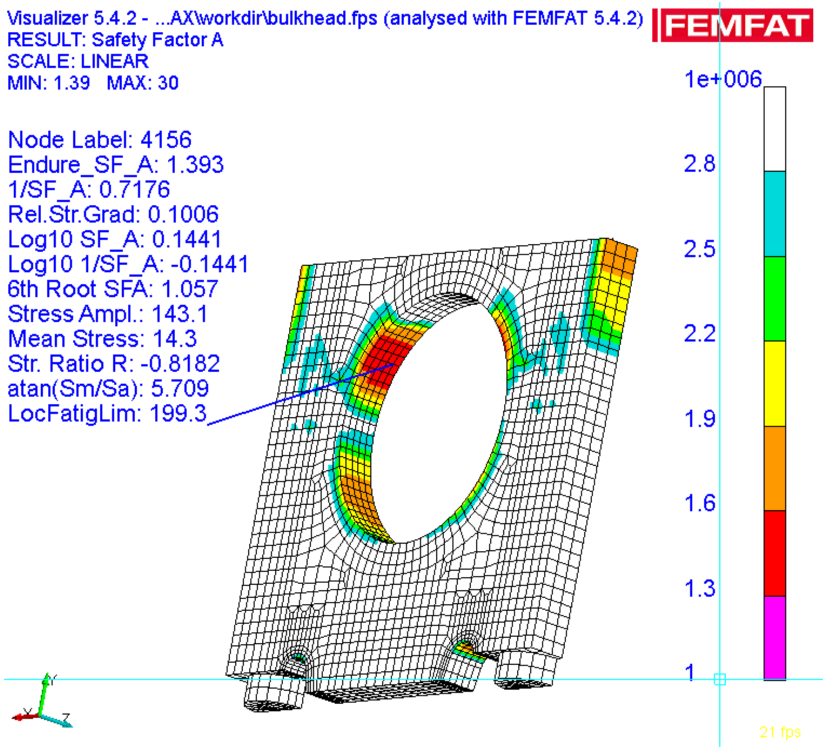
Various charts (S/N curve, Stress History, ...) can also be displayed directly in the FEMFAT GUI in the Visualization menu after the calculation.

Analysis Target, Charts & Visualization









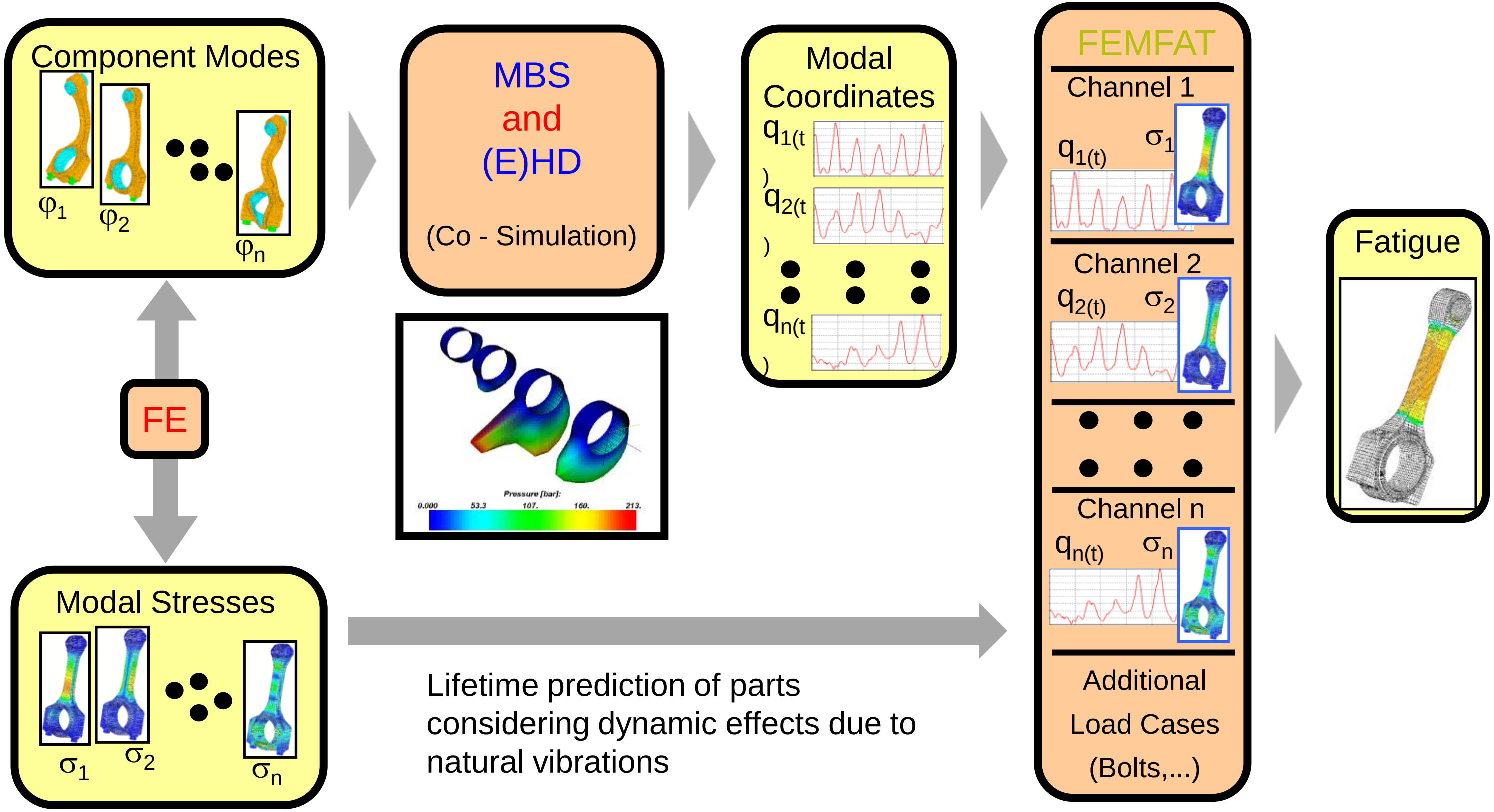
Fatigue Analysis by Modal Stresses

in channelMAX

MBS + channelMAX

FEMFAT ChannelMAX provides the possibility of performing parallel integration of FEM and MBS and operational strength analyses.

Hybrid Models - FE + MBS + channelMAX

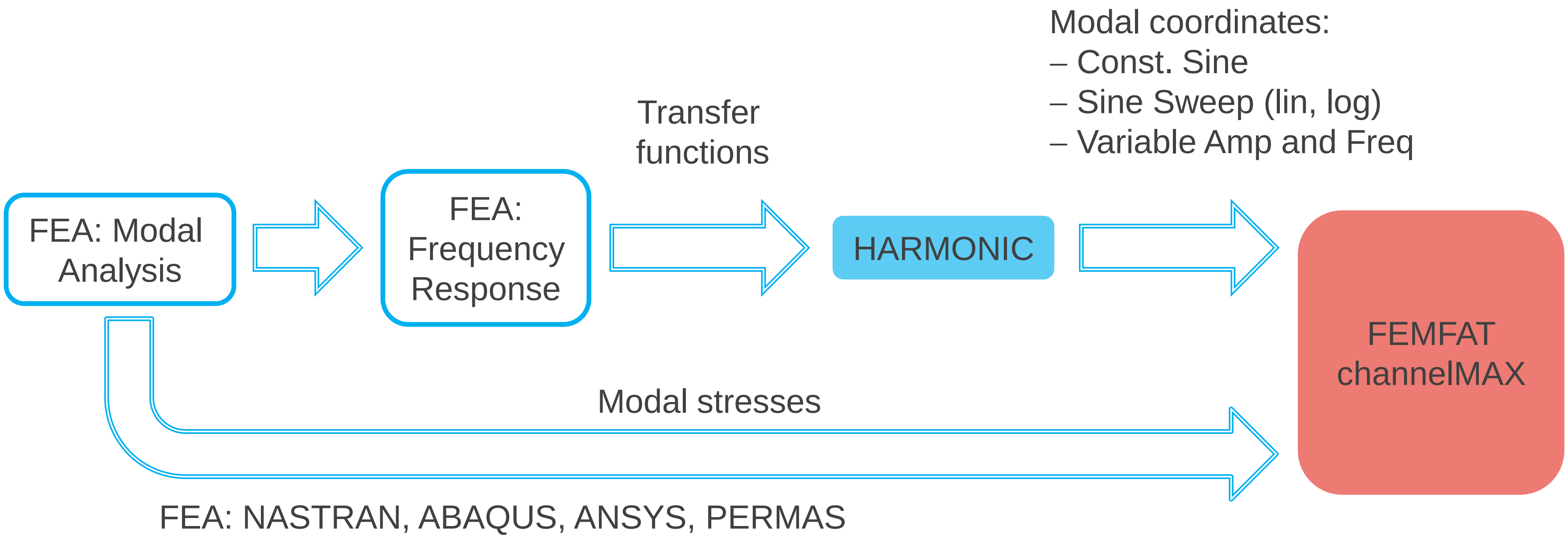


Lifetime prediction of parts considering dynamic effects due to natural vibrations

HARMONIC + channelMAX

The modal coordinates and the modal stresses are the basis for a modal fatigue analysis in channelMAX.

HARMONIC + channelMAX



HARMONIC uses transfer functions to compute a dynamic system's response in terms of modal coordinates.

Analysis Process in HARMONIC

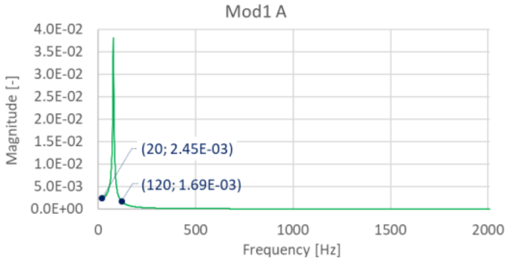
Eigenfrequency Analysis

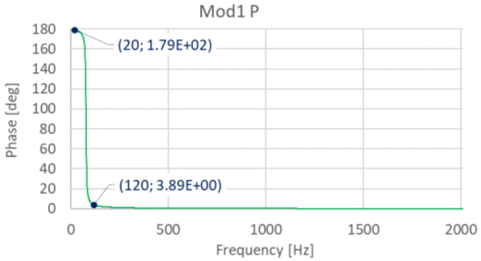
→ Modal Stresses for FEMFAT

↓

Frequency Response Analysis:

(Unit Loadcase with an excitation of 9.81m/s²)





System Behavior:

Transfer Functions from FE File:

$$G(j\omega)$$

Input:

$f_{Start}, f_{End}, f_A, T_{max}, \dots$ from *.hcf File:

$$\omega = 2 \cdot \pi \cdot f$$

$$h(t) = A(\omega) \cdot \sin\big(\omega \cdot t + \varphi(\omega)\big)$$

Optional Input:

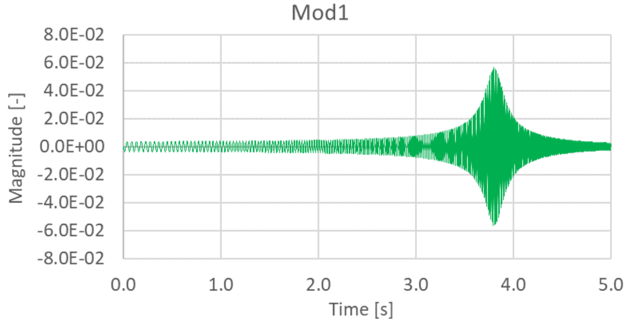
$A(\omega)$

from *.txt File:

```
# freq ampl
END_OF_HEADER
10   10
20   20
30   40
...
```

Output: *… without Optional Input*

→ Modal coordinates in time domain for FEMFAT



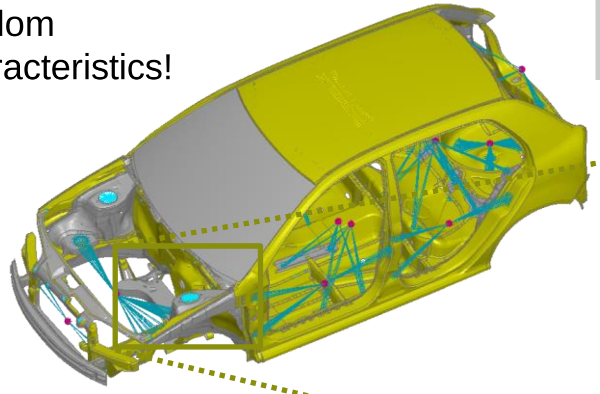
Analyses in Frequency Domain

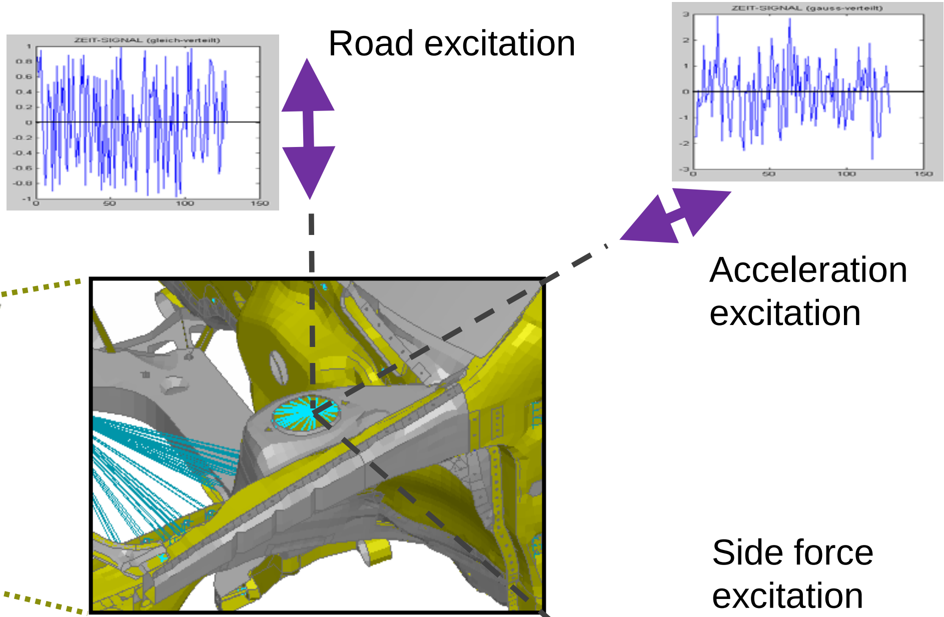
FEMFAT spectral

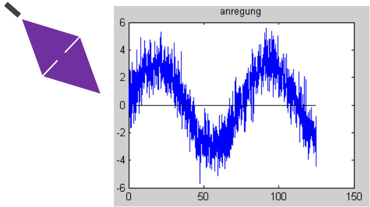
MAGNA

Motivation: Body-In-White

Signals with random characteristics!



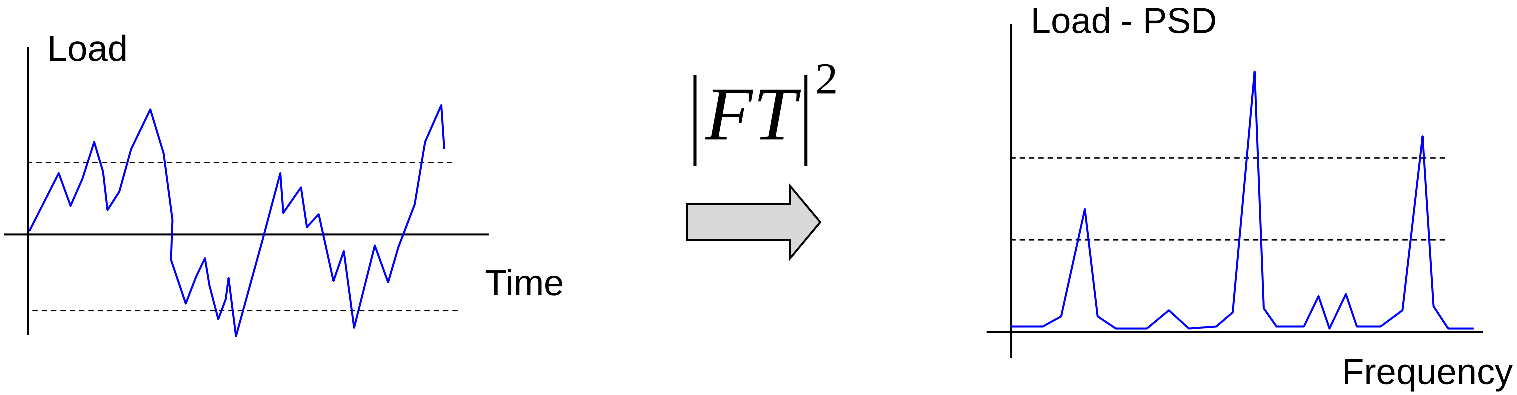




Excitation examples:

- High frequency vibration of firing excitation
- Shaker tests (electronic devices, automotive components)
- Turbulence (aerospace)
- Wind/wave excitation (buildings)
- …

Frequency Domain – Fourier Transformation



$$|FT|^2$$

Power Spectral Density (PSD) =
Square of Fourier Transformed Signal =
Mean Value of “Power of the signal”

Small displacements and linear elastic material behaviour can be approximated with sufficient accuracy by a linear relationship.

- Linear Time-Invariant (LTI) Systems

$f(t)$ → **Linear Time-Invariant System** → $$u(t) = \int_{-\infty}^{\infty} h(t-\tau) f(\tau)\, d\tau$$

Output-Signal = Superposition of time-shifted Impulse-Response-Functions

$$u(t) = h(t) * f(t)$$

$h(t)$ ... Impulse - Response

(System - Property)

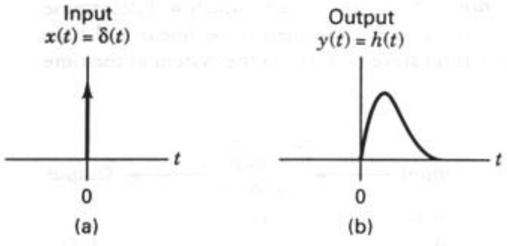


Figure 1.10 (a) Input signal $x(t)$ is a unit impulse function. (b) Output signal $y(t)$ is the system's impulse response $h(t)$.

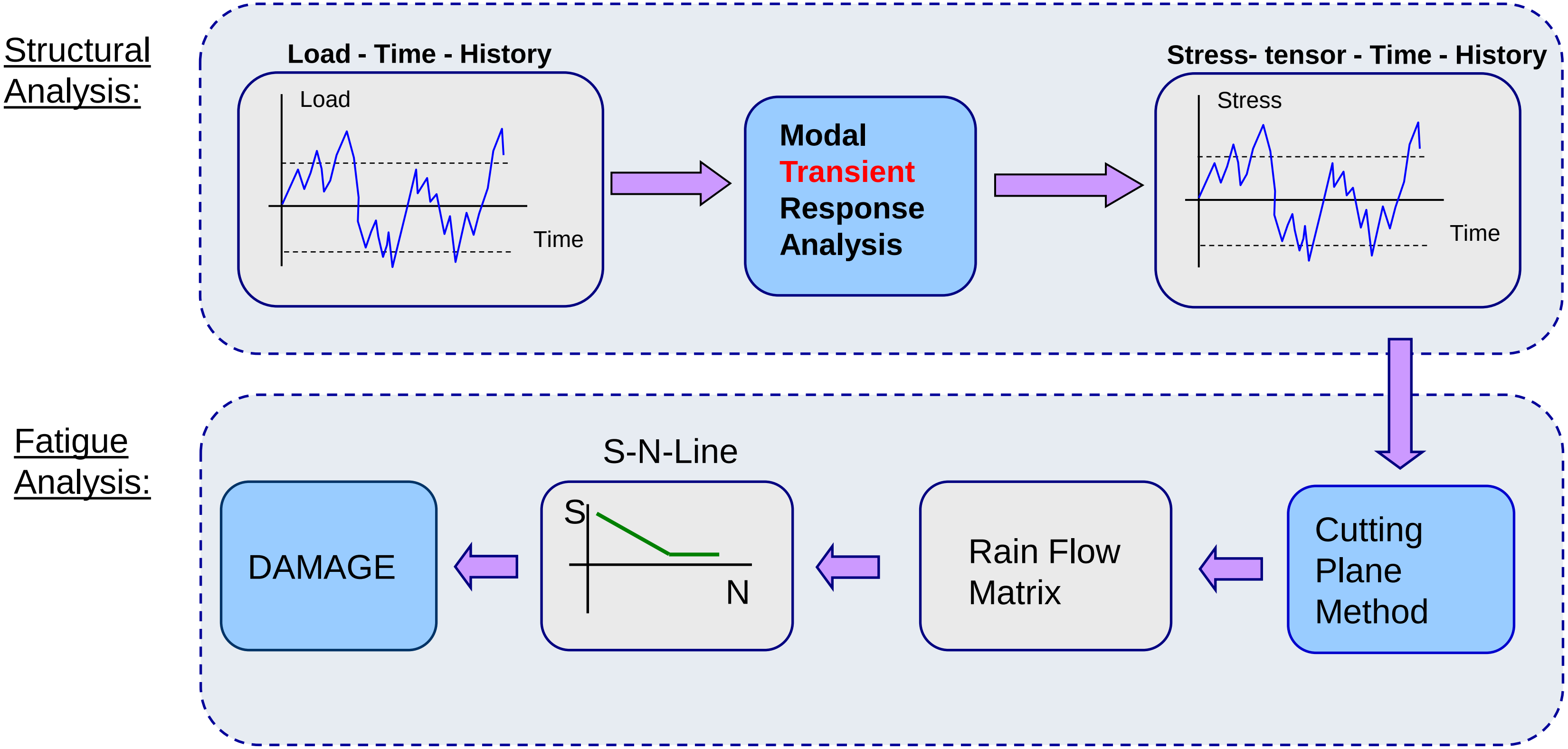
Fourier-transformation:
Convolution in time-domain => Multiplication in frequency-domain

$$U(\omega) = H(\omega) F(\omega)$$

Advantage:
Simple multiplication in frequency-domain

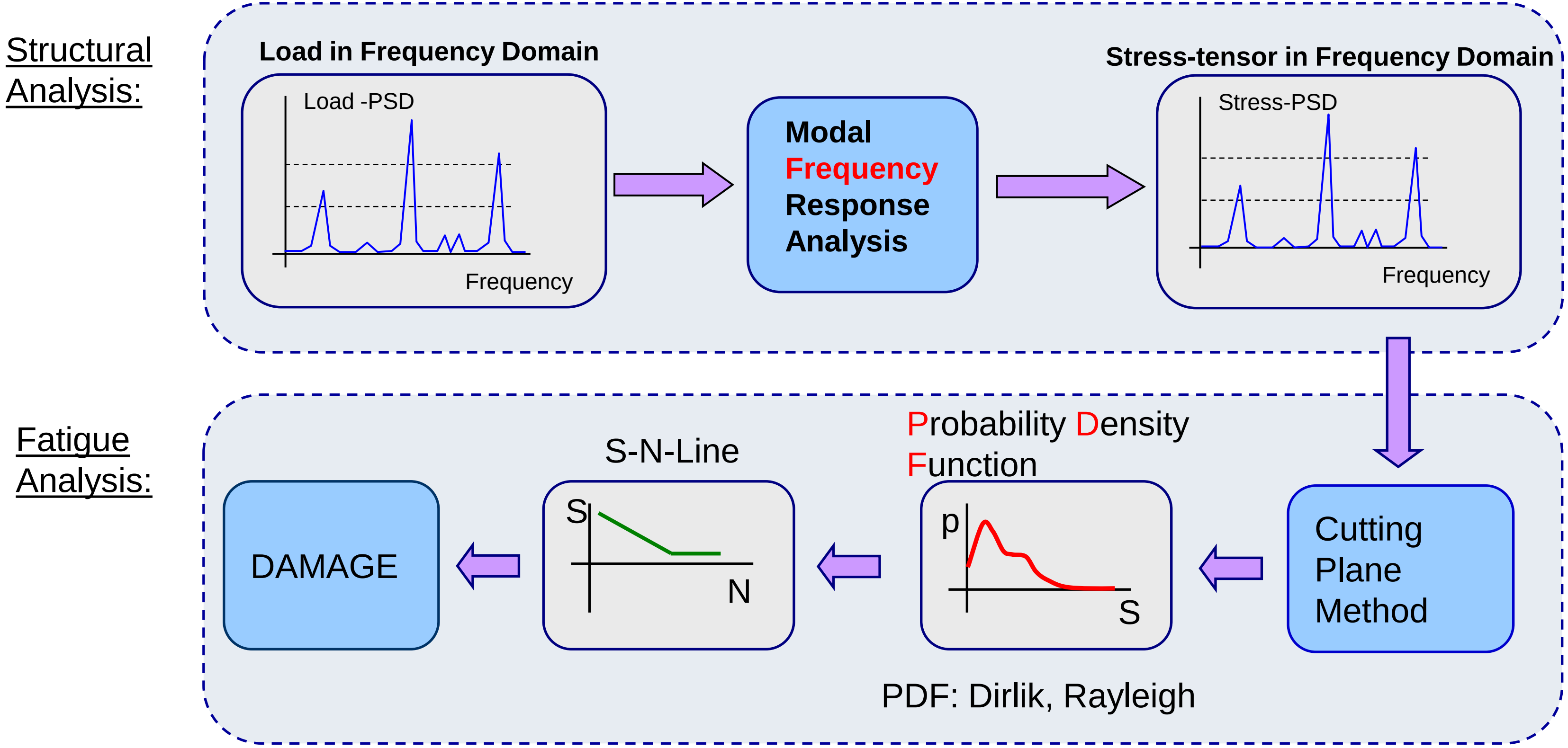
In time domain rainflow counting is done for each cutting plane and S/N curve is displayed for the most damaged rainflow entry on the cutting plane.

Analysis Technique in Time Domain



For each associated equivalent stress PSD (result from application of cutting plane method) a stochastic rainflow matrix is estimated for damage calculation.

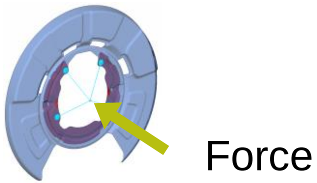
Analysis Technique in Frequency Domain



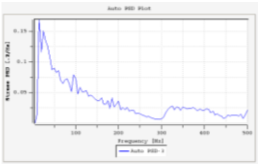
Computation time acceleration compared to time domain analysis is a factor of about 140 !!!

Brake Disc Cover

with courtesy of BMW

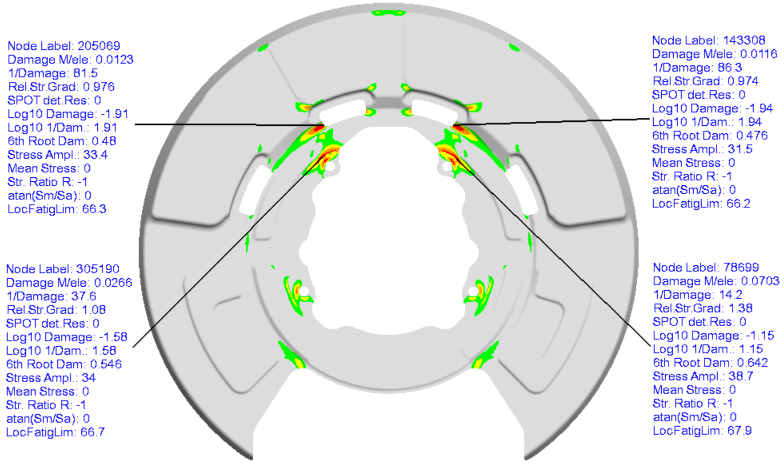


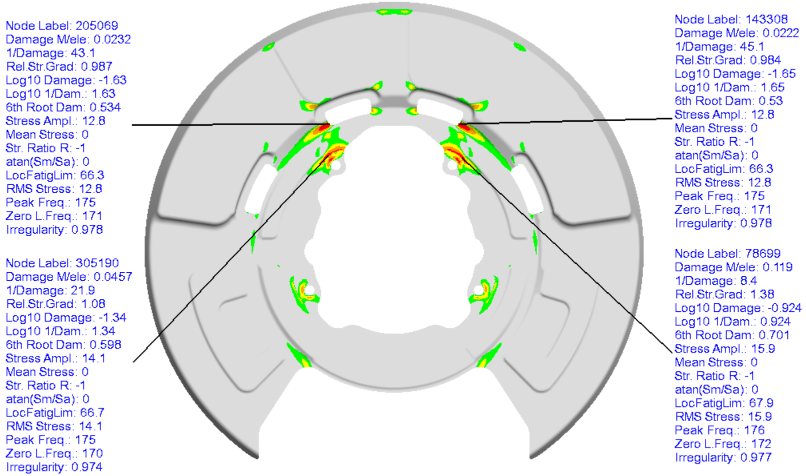
PSD:



Analysis in time domain with channelMAX:
- 1.2 million time points

Analysis in frequency domain with SPECTRAL:
- 2000 spectral lines
- from 0 to 500 Hz





Summary and Outlook

Advantages:

- Fast method (structural analysis, fatigue analysis)
- Load-case superposition in FEMFAT (very flexible)
- Simple combination of different load situations
- Simple simulation chain (no multi-body simulation, reverse FFT not required)

Disadvantages:

- Linear elastic behavior assumed/required (superposition)
- Not suitable for deterministic loads

Conclusion:

- FEMFAT spectral is a reliable and effective tool for damage analysis of multi-axially stochastically loaded systems.

Joint Assessment in FEMFAT

FEMFAT weld

Fatigue Assessment of Welds

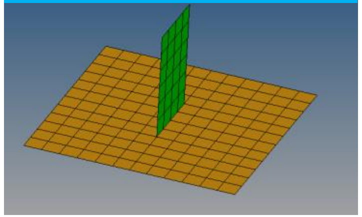
The finite element model needs to be built being conform to the modelling guidelines

FEMFAT modelling guideline for…

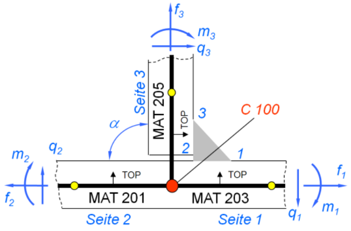
1

Finite Element Modelling acc. to Modelling Guideline

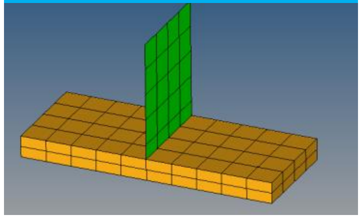
2D shell elements



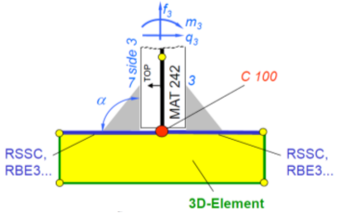
The joint sheets are also simply represented as shell elements, independent of the joint and weld type. The detailed geometry of the welds is not modeled.



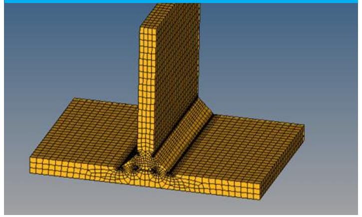
2D & 3D elements



Modeling of welds in which shell elements are welded to solid elements is analogous to pure shell welds. Stresses in the solid element are not considered for analysis.

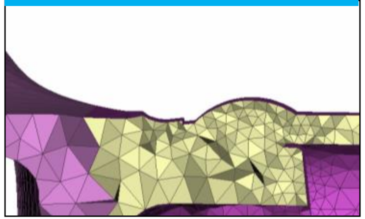


3D elements



In the current FEMFAT weld version there are no specific modeling guidelines for welds made up of solid elements only. In some cases it can be useful to deactivate consideration of the relative stress gradient influence in the notch bases of 3D welds. Node color C200 (or group name C200) can be used to switch off the influence automatically for the respective nodes.

Solid Weld

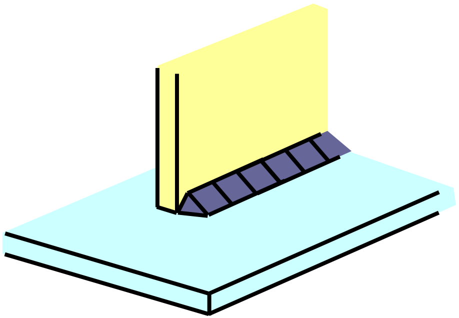


With this method, which is currently only available for ChannelMAX, the weld roots and toes are analyzed based on a relatively coarse volume mesh with no rounding radiuses by means of stress interpolation method. Stresses are compared to a master S/N curve which is based on many tests.

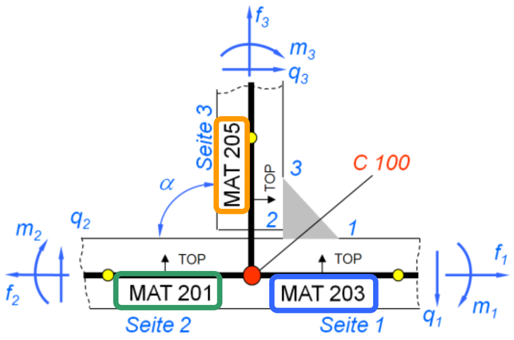
WELD

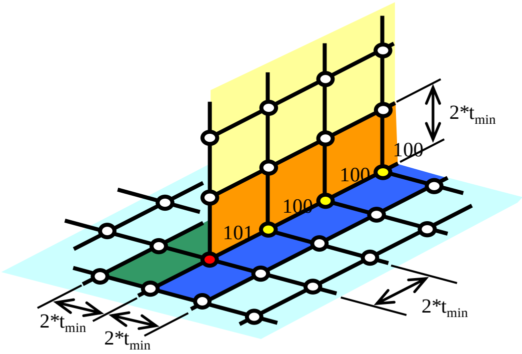
Creating a weld seam mesh with the help of the Modelling Guideline and your pre-processor

Modelling of FEM weld seams



Based on the geometry a mesh is generated. The element size of the mesh should be double the minimum thickness of the welded plates.





The finished definition includes all parameter (for every node and element) for the weld seam.

The Modelling Guideline includes all necessary information about the weld seam which are needed to define the weld for the FEMFAT analysis with the Pre-processor. FEMFAT visualizer uses this information automatically when defining the weld seams. There are different guidelines and weld databases provided for common standards as EUROCODE, BS,….

FEMFAT weld - Methods overview

MAGNA

Choice of 2 Methods

Stress based

Advantages

- Support of many welding seam types
- Special notch factors for welding seam start/end nodes available
- Consideration of normal stress perpendicular to welding seam, normal stress parallel to welding seam and shear stress

Disadvantages

- No weld geometry parameters considered
- Continuous transition between T-joint and overlap joint must be divided

Force based

Advantages

- Less sensitive to mesh quality
- Takes into account weld geometry parameters
- Continuous transition between T-joint and overlap joint

Disadvantages

- Support of 4 welding seam types
- Welding seam start/end nodes are treated like weld middle nodes
- Consideration only of normal stress perpendicular to welding seam

Analysis in FEMFAT weld – stress based

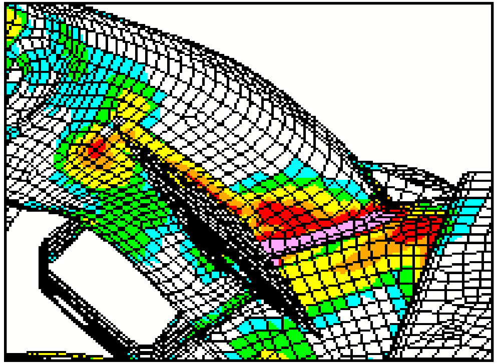
Structural Stresses • Notch Factors = Notch Stress

Structural Stresses out of shell-meshed FE models

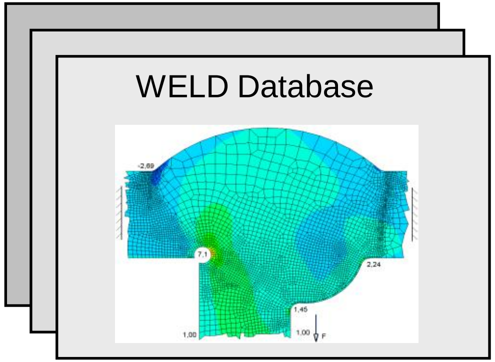
Details are stored in open FEMFAT WELD database:

- Notch factors
- Weld Haigh-Diagram
- Weld S/N Curve

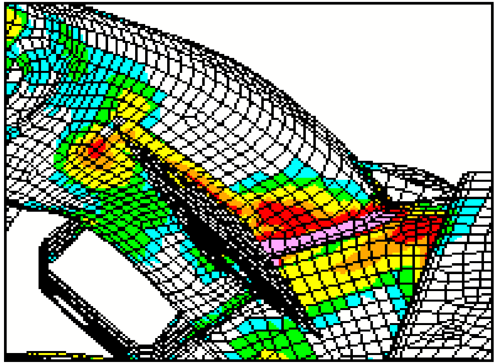
Notch Stresses are Structural Stresses multiplied by Notch factors



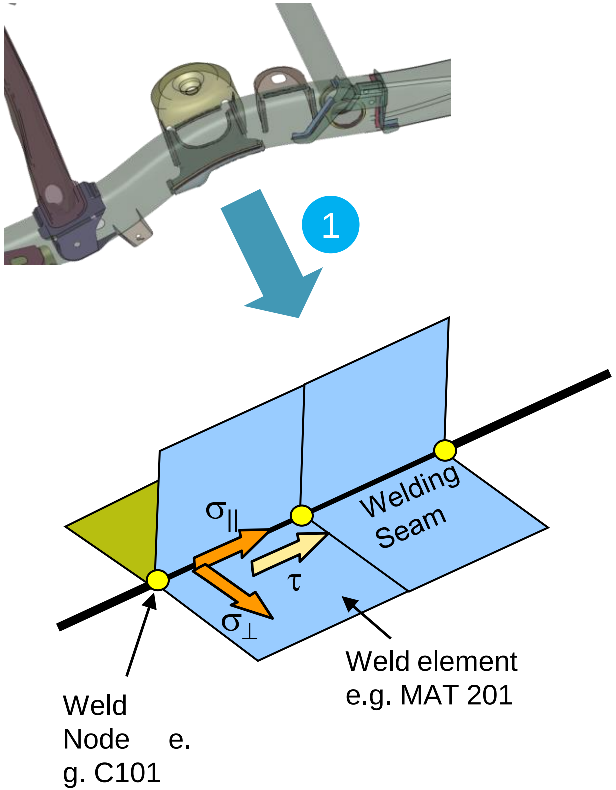
•



=



Analysis in FEMFAT weld – stress based



1. Transformation of the stresses to a local coordinate system
2. Calculation of notch stresses in all directions

Principal Points

- Anisotropic strength characteristics for welding seams
- Stresses perpendicular and parallel to seam are critical
- Transformation to local coordinate system

Normal stress perpendicular direction in notch (**in principle**):

$$\sigma_{\perp,notch} = \sigma_{\perp,nominal} \cdot \beta_{\perp}$$

Normal stress longitudinal direction in notch:

$$\sigma_{\parallel,notch} = \sigma_{\parallel,nominal} \cdot \beta_{\parallel} \cong \sigma_{\parallel,nominal} + \nu \cdot (\sigma_{\perp,notch} - \sigma_{\perp,nominal})$$

Shear stress in notch with $\alpha_{\tau} = (\alpha_{\perp} + 1)/2$:

$$\tau_{notch} = \tau_{nominal} \cdot \beta_{\tau} \cong \tau_{nominal} \cdot (\beta_{\perp} + 1)/2$$

$\beta_{\perp}$ …derived notch factor perpend. to the weld seam ← from the database
$\beta_{\parallel}$ …notch factor parallel to the weld seam ← from the database or estimated
β_{τ} …shear notch factor ← from the database or estimated.

Analysis in FEMFAT weld – stress based
Equivalent Safety Factor / Equivalent Damage

Based on the stresses a damage or safety factor is calculated for every direction. Then a total result is generated on the base of the formula which is based on the DVS 1608. The weld start- and end- results are calculated additionally.

- Equivalent Strain Energy Hypothesis using DVS 1608 (standard)

$$a_v = \sqrt{a_\perp^2 + a_\parallel^2 + f \cdot a_\perp \cdot a_\parallel + a_\tau^2} \quad \longrightarrow \quad D_v = {a_v}^{k_{eff}}$$

$$\frac{1}{S} = \sqrt{\left(\frac{1}{S_\perp}\right)^2 + \left(\frac{1}{S_\parallel}\right)^2 + f \cdot \left(\frac{1}{S_\perp \cdot S_\parallel}\right) + \left(\frac{1}{S_\tau}\right)^2}$$

a......Utilization degree
D_v ... Equivalent damage
k_{eff} ....weighted averaged slope of S/N curve
S.......Safety factor
f ....weighting factor for multiaxility ($-1 \leq f \leq +1$)

- Weld Start and End Nodes using HAIBACH (assessment of stress components individually)

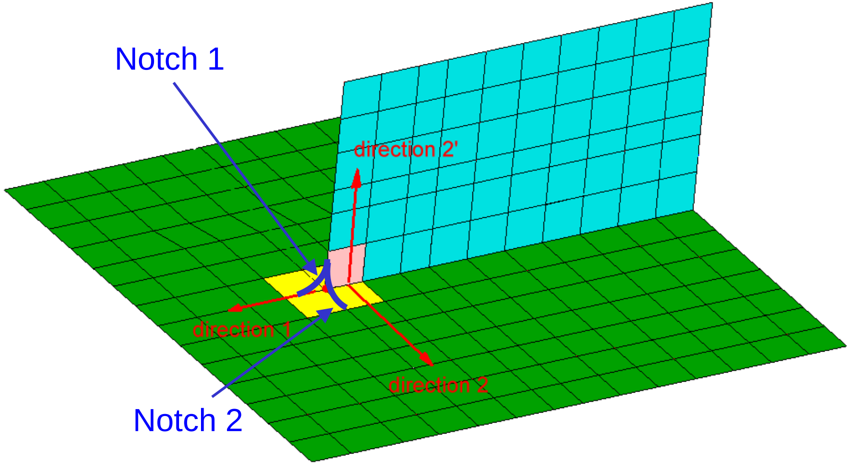
$$\frac{\sigma_{a,Equiv.}}{\sigma_{A,Equiv.}} = MAX\left(\frac{\sigma_{a,\perp}}{\sigma_{A,\perp}}; \frac{\sigma_{a,||}}{\sigma_{A,||}}; \frac{\tau_a}{\tau_A}\right)$$

$$\frac{1}{S} = MAX\left(\frac{1}{S_\perp}; \frac{1}{S_\parallel}; \frac{1}{S_\tau}\right)$$

σ_a...Stress amplitude of the load
σ_A...Admissible stress amplitude (from S/N curve)
S... Safety factor

Stress correction and analysis at weld start/end node is done in two directions

Weld start/end is analyzed twice:



Position code (analysis output)

EWUTOP	Weld start/end notch 1, root, shell top
EWUBOT	Weld start/end notch 1, root, shell bottom
EUETOP	Weld start/end notch 1, toe, shell top
EUEBOT	Weld start/end notch 1, toe, shell top
NWUTOP	Weld start/end notch 2, root, shell top
NWUBOT	Weld start/end notch 2, root, shell bottom
NUETOP	Weld start/end notch 2, toe, shell top
NUEBOT	Weld start/end notch 2, toe, shell bottom

1.) Analysis as weld middle node

- Automatic stress correction in direction 2 and 2‘ resp.
- Notch factors and SN curves for middle nodes from weld DB are used for analysis

2.) Analysis as weld start/end node

- Automatic stress correction in direction 1
- Notch factors and SN curves for start/end nodes from weld DB are used for analysis

$$\sigma_{\perp,notch} = \sigma_{\perp} \cdot \beta_{||}$$

$$\sigma_{||,notch} = \sigma_{||} \cdot \beta_{\perp}$$

FEMFAT weld database content - overview

FEMFAT WELD database

Overview of supported joint types

Polygon lines for influences on the weld fatigue limit:
- Sheet thickness
- Base material
- Temperature

Haigh Diagrams for normal and shear stress

Strength data for SolidWELD assesent:
- S/N curves for root, toe, start, end
- Interpolation distances
- Sheet thickness influence
- Base material influence
- Temperature influence
- Haigh Diagram

Notch Factors and S/N Curves
- For weld start, end and middle nodes
- For about 50 joint types

Table for substitute element allocation

Data for force based assessment (SSZ/MSZ Method)
- Geometry parameters
- Master S/N Curve

Table with SID-ranges of the joint types

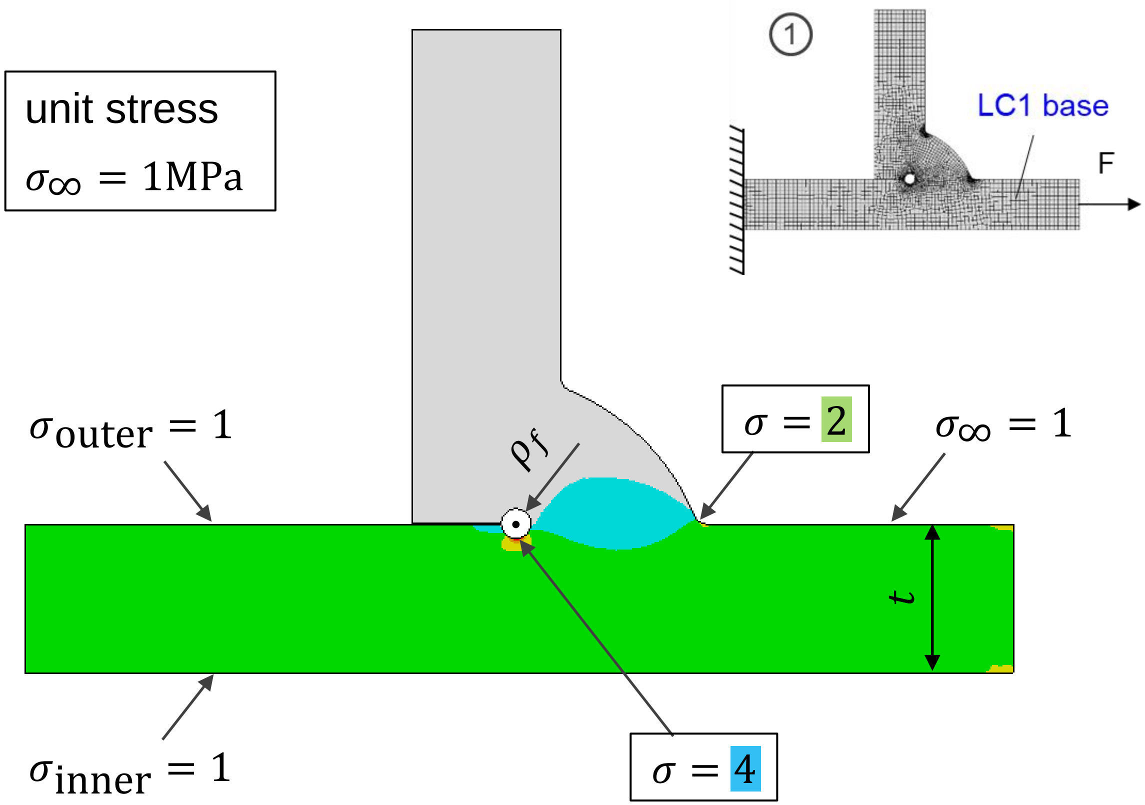
Parameters for detailed weld geometry display in VISUALIZER

Must be modifed when introducing a new joint into the database

Notch factors are determined for the relevant sheet and load-case for unity stress. The undercut from the radius of Radaj modelling leads to higher root stresses. Therefore, the notch factors must be corrected.

4

Determine notch factors



… in roots:

Correction of Notch Factors

$$NFR = \frac{4}{1} \cdot KF = \mathbf{2,7}$$

$$KF = \frac{(1-\rho^*)^2}{1+\rho^*(1+\sigma^*)} = 0{,}675$$

$$\rho^* = \frac{\rho_f}{t} = \frac{0{,}6}{6} = 0{,}1$$

$$\sigma^* = \frac{\sigma_{outer}}{\sigma_{inner}} = \frac{1}{1} = 1$$

$$NFT = \frac{\sigma}{\sigma_\infty} = \frac{2}{1} = \mathbf{2,0}$$

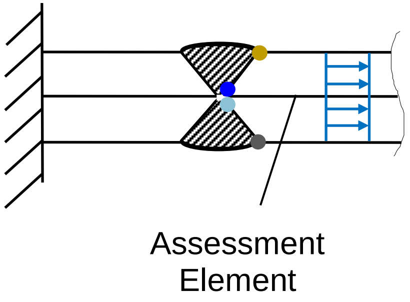
NFR …notch factor root
NFT …notch factor toe

Notch factors are assigned to different material labels acc. to modelling guideline

5

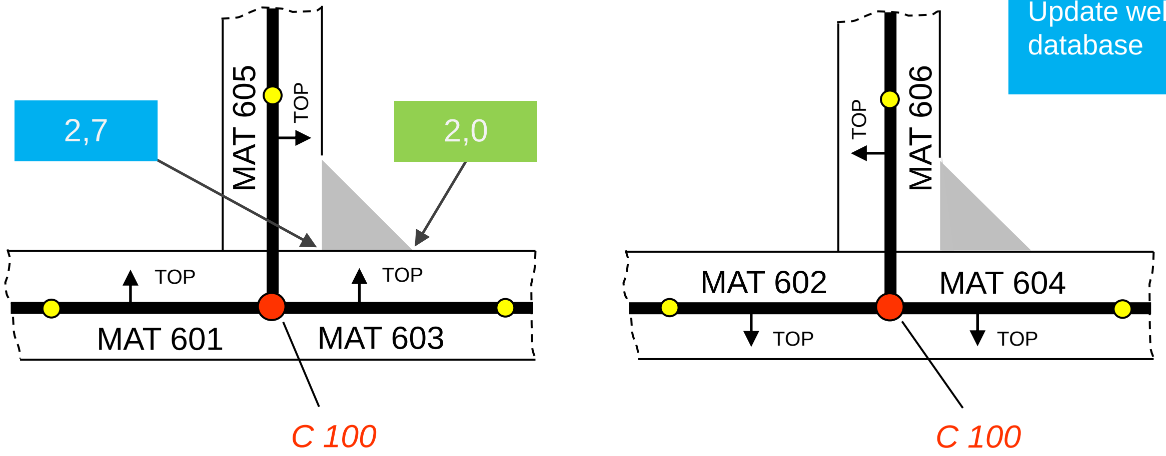
Update weld database

Assessment Points



- UT: Toe TOP
- WT: Root TOP
- WB: Root BOTTOM
- UB: Toe BOTTOM

… up to 4 assessment points per element



MAT	UTL1	UTL2	UTL3	UBL1	UBL2	UBL3	WTL1	WTL2	WTL3	WBL1	WBL2	WBL3
601							2,7					
602										2,7		
603	2,0											
604				2,0								
605												
606												

WELD – Sensitivity Analysis

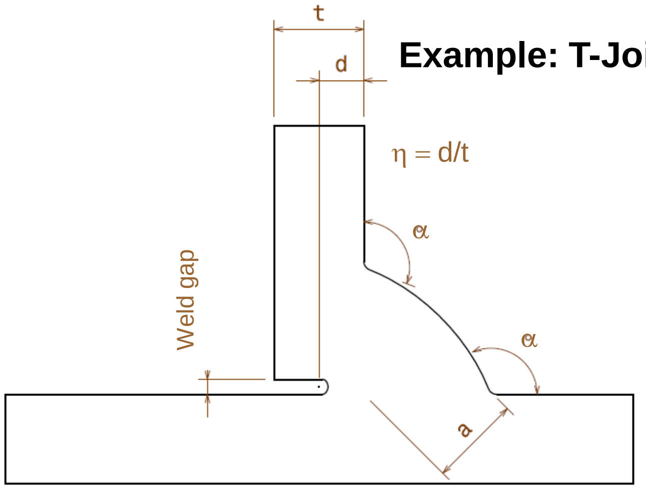
FEMFAT weld Sensitivity Analysis: Weld Geometry Definition

different quality classes for geometry parameters

different fatigue results

Computation of Sensitivity Factor

Example: T-Joint 90°



Parameter – T-Joint 90°	Better Quality (Quality class B)	Standard Quality (Quality class C)	Worse Quality (Quality class D)
Degree of weld penetration - **η**	100%	50%	0%
Seam thickness - **a**	1.5 t	t	0.7 t
Seam inclination angle **- α**	110°	100°	90°
Weld gap (at 3mm thickness)	0mm	0.5mm	1.5mm

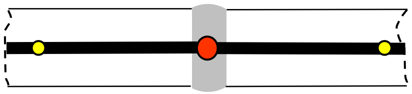
→ Determination of notch factors for weld database (total 9 databases)

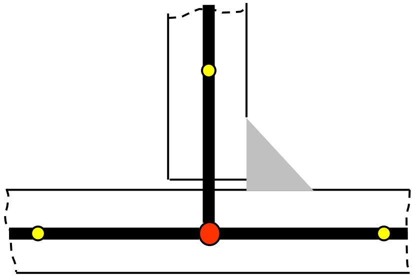
FEMFAT Weld Sensitivity Analysis: Supported Weld Types

MAGNA

- Butt Joint
- Lap Joint
- T – joint 90°
- T – joint 45°

Only one sided joints for steel. Double sided joints or aluminum welds are not supported.

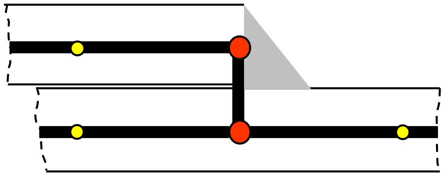


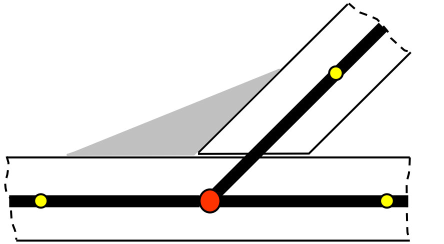


Sensitivity

S

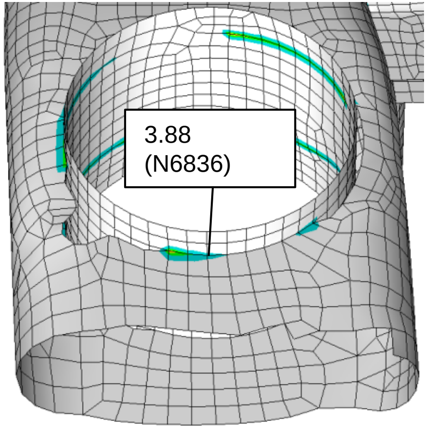
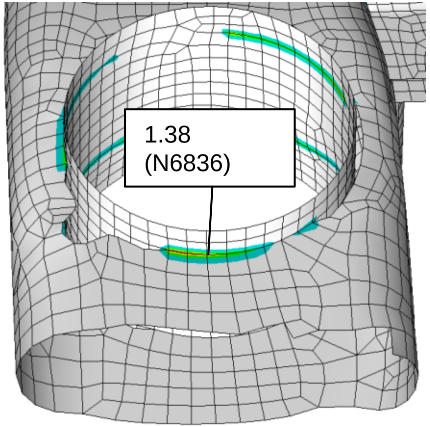
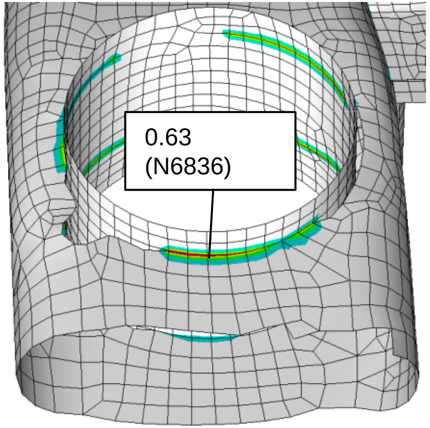
Analysis





FEMFAT Weld Sensitivity Analysis: Example

Results for varying geometry parameter ‘Degree of Weld Penetration‘

Endurance Safety [-]		
Better quality	Standard quality	Worse quality
		

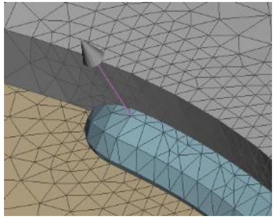
$$F_{sens} = \frac{S_{big} - S_{small}}{S_{big}} = \frac{3.88 - 0.63}{3.88} = 0.84$$

High Sensitivity to variation in degree of weld penetration

SolidWELD

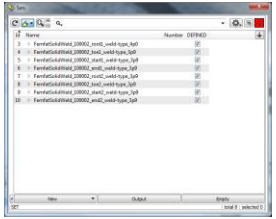
Workflow (at least a FEMFAT 5.2 MAX license is presumed)

1 – Volume mesh creation



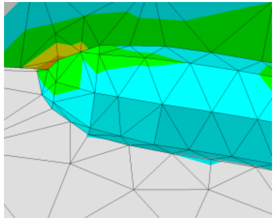
Creation of a volume meshed FE model of the weld joint. The toes and roots are not rounded in this case, but are instead angular. The FE mesh here must be refined locally in a general fashion in the vicinity of the weld toes and roots (at least three elements over the sheet thickness).

2 – Save nodes of toes and roots in sets/groups



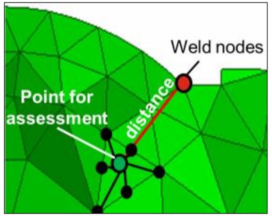
The nodes on the edges for toes and roots must be saved in groups (sets) using certain naming conventions. These sets are used in FEMFAT to identify the solid weld seam nodes that are to be analysed.

3 – Finite Element Simulation



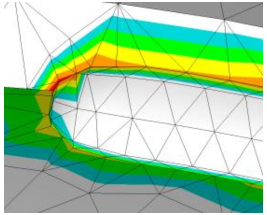
Finite Element Simulation can be done with each solver that is supported by FEMFAT. It is essential to have the groups to further process the data in FEMFAT.

4 – Stress interpolation method using Master S/N curve



The analysis is performed using the critical distance method, which has been modified for coarse FE meshes (element length < t/3). Analysis data, such as a master S/N curve, sheet thickness influence, assessment distances, etc., are stored in the weld database.

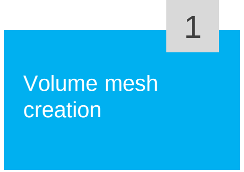
5 – Calculation of the safety factor or the damage

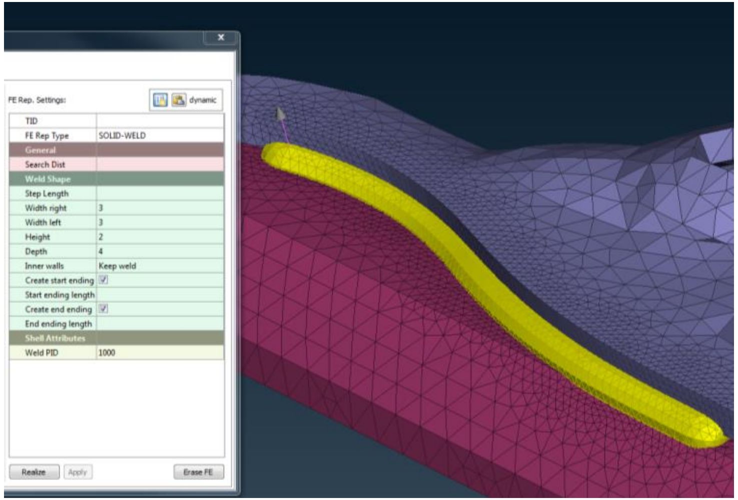


Creation of the MAX scratch files (*.fms) for a node group, which also contains enough nodes in the vicinity of the assessment depth for root and toe. Enable the WELD module switch. Calculation of the safety factor or the damage. It is possible to perform both base material and shell-based WELD and SPOT analyses at the same time using SOLID WELD assessment.

In cooperation with ANSA and SimLAB an automated process of modelling a welding seam was developed.

1. Define the connection lines between the sheets.
2. Define the geometry of the cross section of the seam.
3. The geometry of the sheets and the seam is combined to a meshable volume.
4. The mesh process is fully automated including the correct element size and group information for FEMFAT.





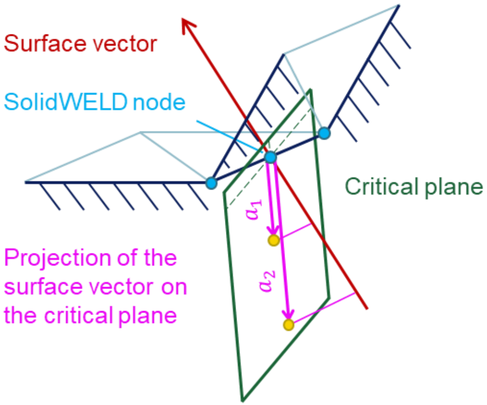
The nodes of the SolidWeld groups are analyzed twice.

4

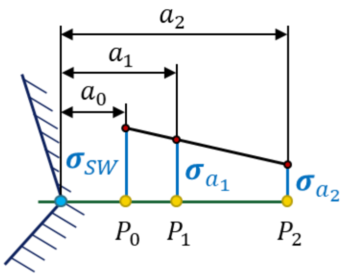
Stress interpolation method using Master S/N curve

The **first analysis** is required to determine the direction of the critical cutting plane. Here, the node is analyzed as a conventional solid node with the default material data assigned to the node, however, the results are not saved.

In the **second analysis** as a SOLID WELD node, the stress is determined at a certain depth in the direction of the critical cutting plane which was determined in the first analysis.



Default values: a_0 = 0.0 mm (extrapolation onto the surface).
a_1 = 0.1 mm
a_2 = 0.5 mm
If a_2=0 or a_2=a_1, then $\sigma_{SW} = \sigma_{a1}$ (no extrapolation).



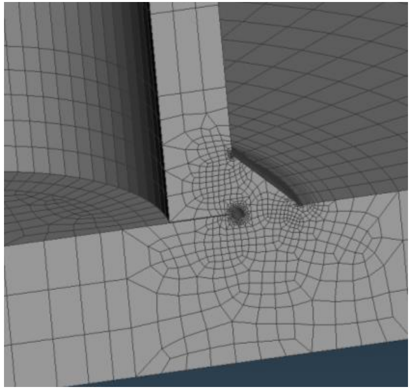
Welds with reference radius

Workflow of a FEMFAT weld assessment using solid model and reference radius

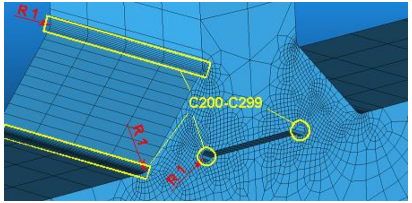
1	2	3
Finite Element Modelling using Reference Radii for root and toe	Material assignment and definition of groups named C200 for nodes in the radius	Fatigue assessment using FEMFAT basic, max or spectral

Example

Finite element model with rounding radius at the root and toe notch.

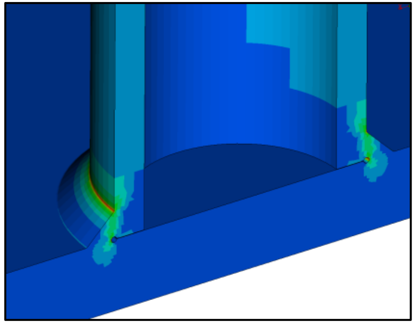


Assign appropriate material to the nodes in the radius.



Definition of groups C200 – C299 to switch off the gradient influence etc.

FEMFAT results in root and toe



Assign material for nodes in radius

2

Material assignment and definition of groups named C200 for nodes in the radius

Reference radius of **r = 1 mm** or **r = t/10** :

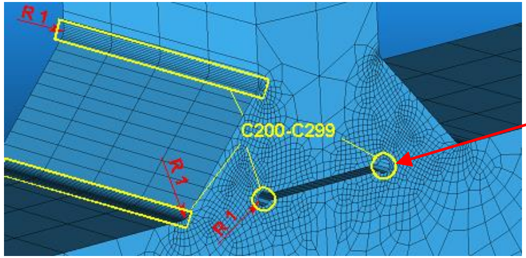
…material_database/iron/General_Structural_Carbon_Steel/

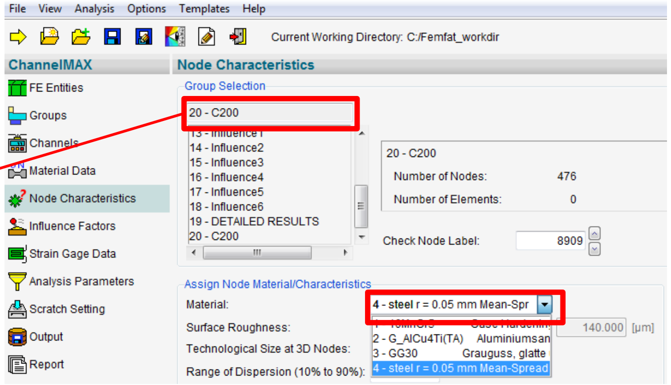
ASTM-50_r1ms_root_TZS_userdef_Haigh.ffd
ASTM-50_r1ms_toe_TZS_userdef_Haigh.ffd

Reference radius of **r = 0.05 mm** :

…material_database/iron/General_Structural_Carbon_Steel/

steel_r=0_05 mm_TZS.ffd.ffd

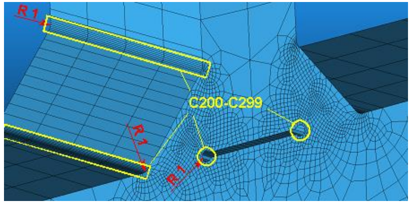




2

Material assignment and definition of groups named C200 for nodes in the radius

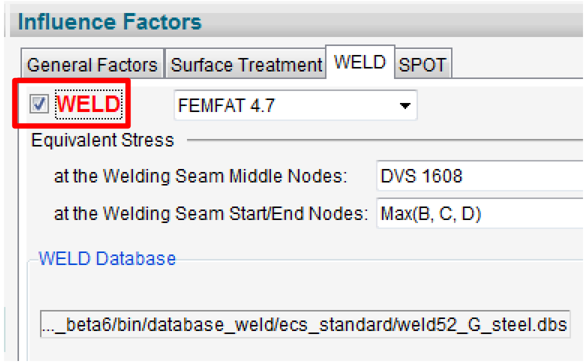
Put nodes of reference radius in groups named ‚C200‘ - ‚C299‘.



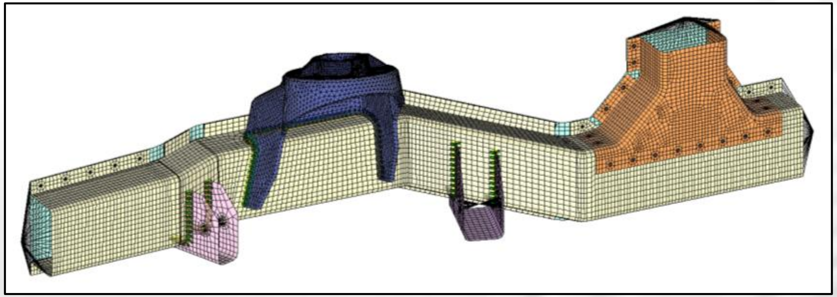
Switch on FEMFAT weld.

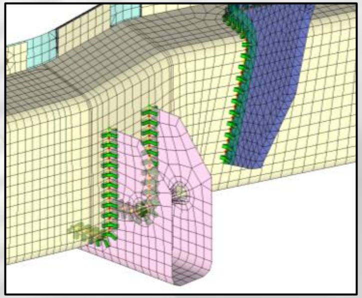
During analysis, for the nodes in the group C200-299 following influences are ignored (if WELD is ON).

- Gradient influence
- Mean stress influence on slope
- Mean stress influence on cycle limit
- Surface roughness
- Mean stress rearrangement (PLAST)



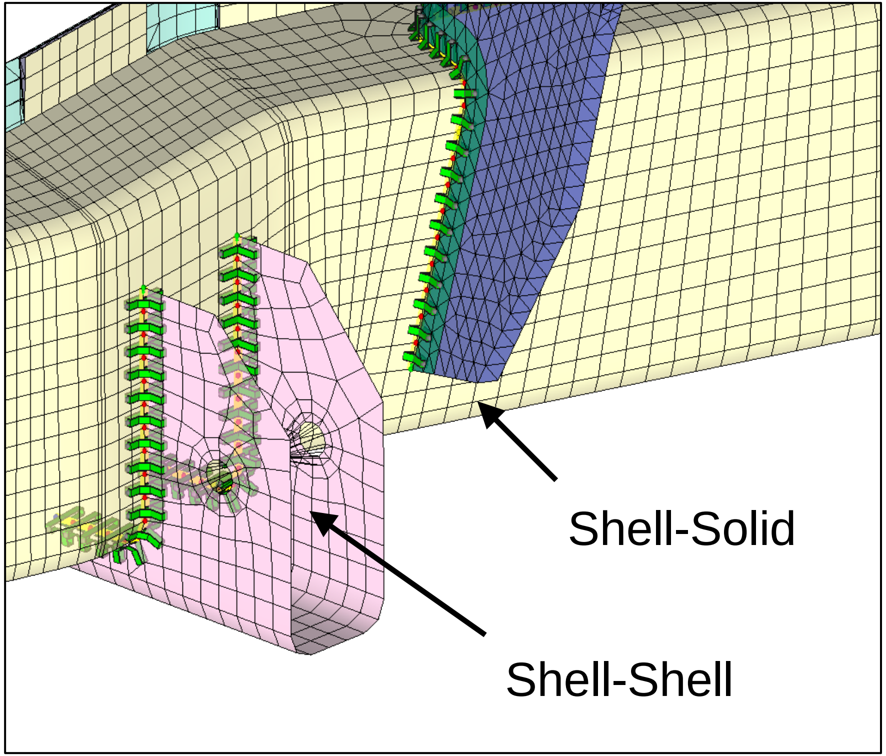
WELD – Example





With shell-solid couplings, only the shell side is analyzed, which typically has the smaller sheet thicknesses and is therefore critical.

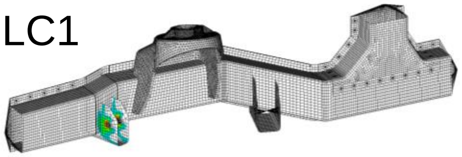
Longitudinal Member: WELD assessment under multiaxial loading

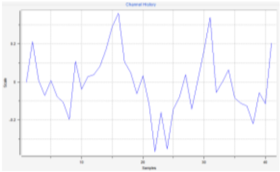


Stresses from Unit Loadcases:

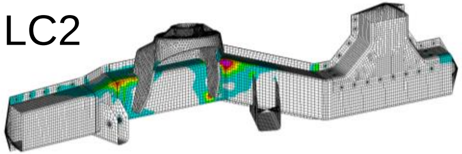
Load-Time Histories:

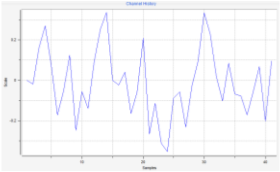
LC1





LC2

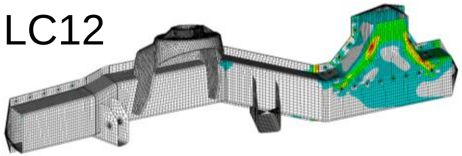


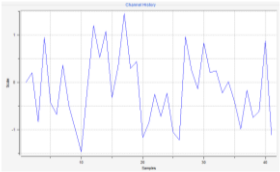


…

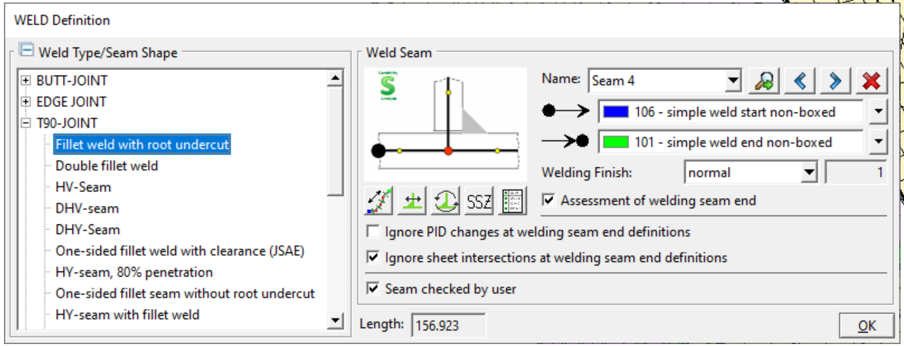
…

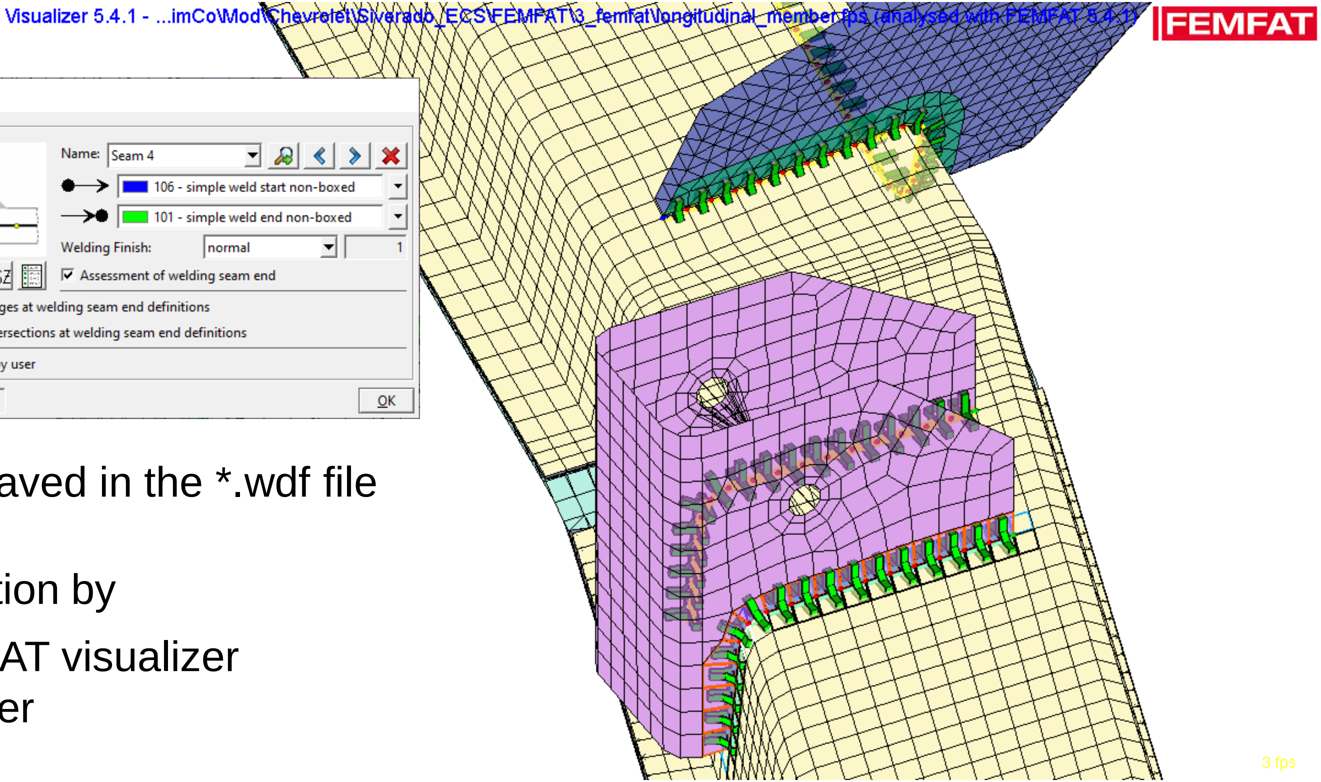
LC12





Definition of weld seams for the fatigue analysis





Weld definitions are saved in the *.wdf file

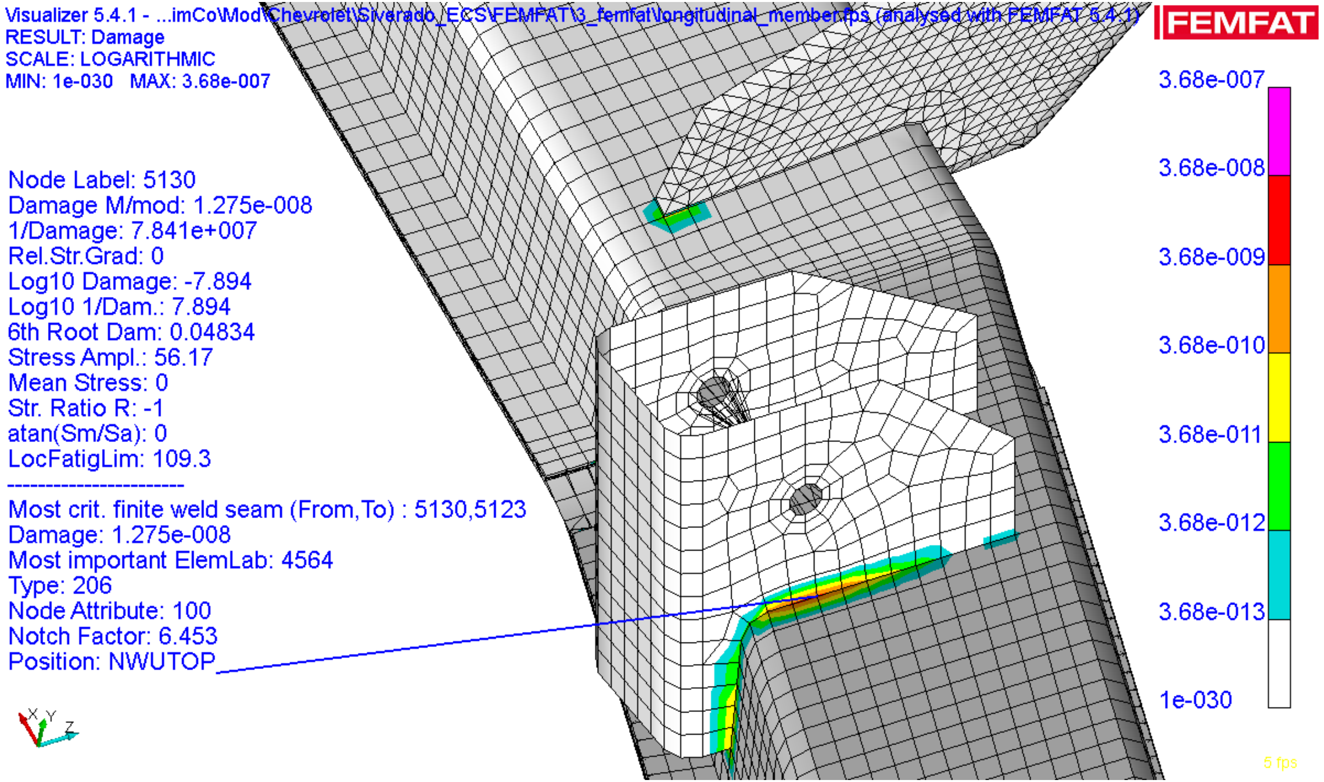
Automatic weld definition by

- xMCF file in FEMFAT visualizer
- WELDseamScanner

Longitudinal Member: WELD assessment under multiaxial loading

Additional Information for WELD nodes.

The detailed output can be found in the protocol file (*.pro)



FEMFAT spot

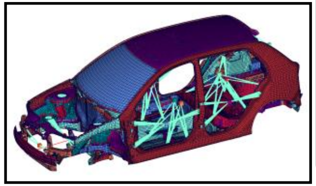
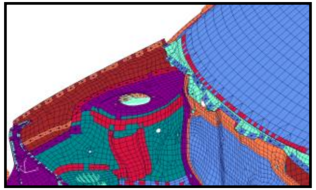
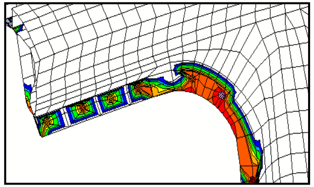
Fatigue Assessment of Spot Welds, Rivets, Nails

Workflow for spot-weld analysis

1. Initial FE-Mesh without weld spot definition
2. Implementation of the FEMFAT „SPOT element“
3. FE Analysis using a FE-solver
4. Fatigue assessment using BASIC or MAX

In contradiction to other FEMFAT modules FEMFAT spot requires an implementation of the connecting SPOT element. Depending on the method a desired connecting element has to be implemented. Then a FEA has to be performed before the fatigue analysis.

Example

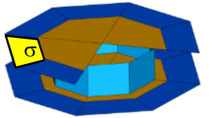
Original Body In White Model	Implementation of the Spot-Joints	FE-solver	FEMFAT Damage Results
		compatible FE-solver	

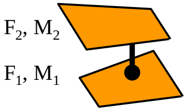
Mind the rules for connecting elements when the force based concept is used in order to obtain suitable results.

The choice of analysis method is dependent on the existing joining technique

Choice of 2 Methods

Detailed model





Connector element

Structural stress of sheets

- **Advantages**
  - Accurate
  - Fast fatigue analysis
  - Easy postprocessing
  - Spot and rivet assessment
- **Disadvantages**
  - Higher meshing effort

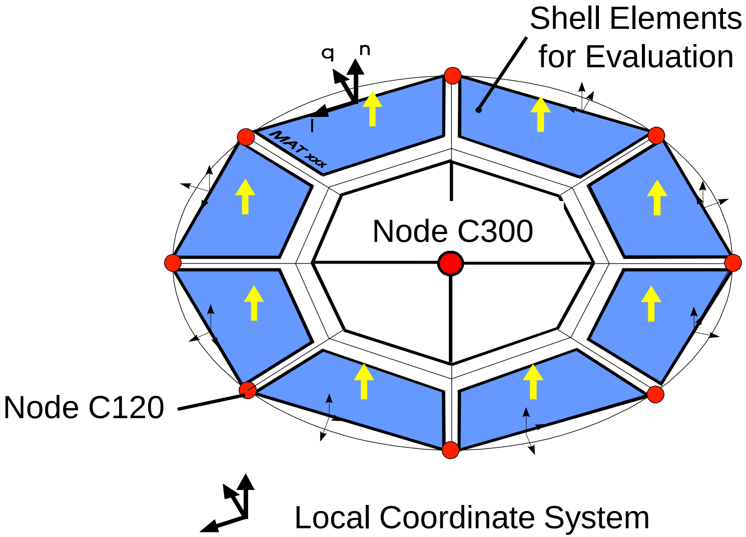
Force between sheets

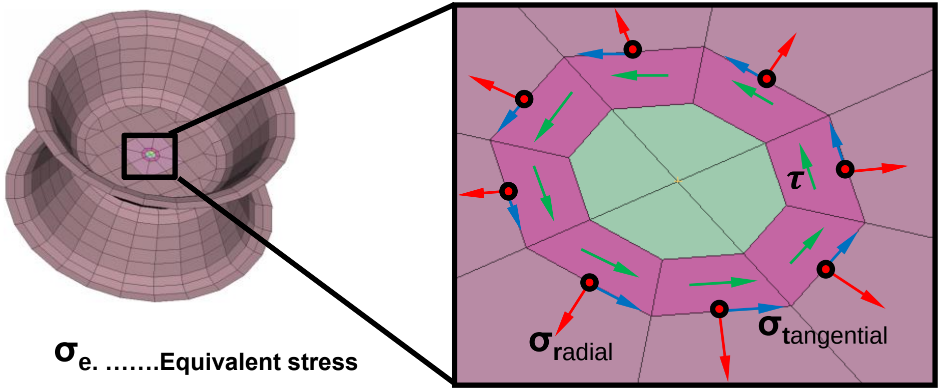
- **Advantages**
  - Easy preprocessing
  - Easy repositioning
  - Various connection models
- **Disadvantages**
  - Problems with certain loading conditions
  - Mesh dependent
  - No rivet assessment

Creation of SPOT nuggets can be generated by FEMFAT Spot Remesher or Preprocessor (e.g. ANSA). In FEMFAT different stress selections are available.

Stress based option

Minimum 2 inner elements are needed!
Maximum 64 outer elements can be used!





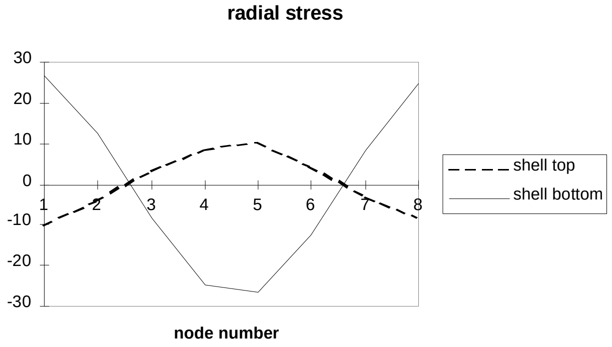
- $\sigma_e = sign(\sigma_r) \cdot \sigma_{Mises}$ **Signed v. Mises Stress**

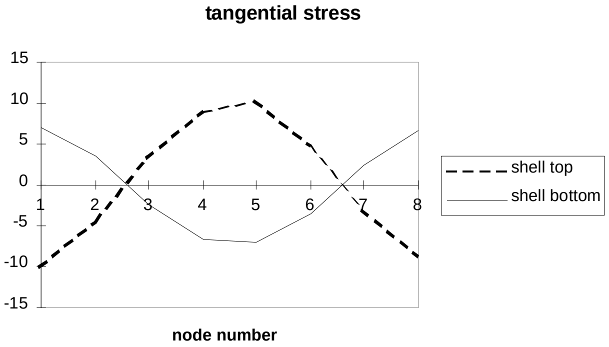
$$\sigma_{Mises} = \sqrt{\sigma_r^2 + \sigma_t^2 - (\sigma_r \cdot \sigma_t) \cdot 3\tau^2}$$

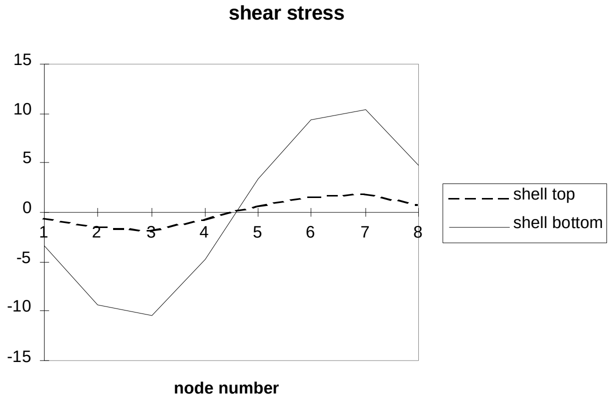
- $\sigma_e = \sigma_r$ **Normal Stress Radial**

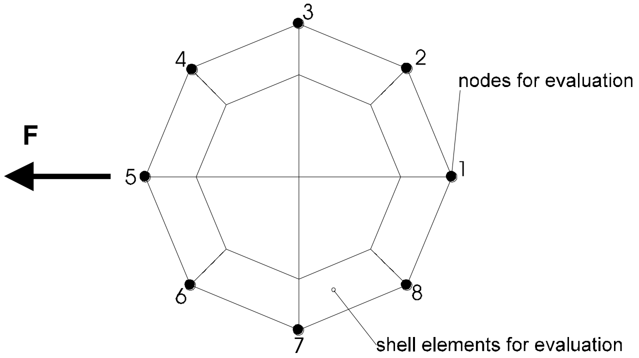
FEMFAT uses the stress components at the top and bottom of the shells to identify the loading type – here an example for pure shear.

Extreme load case: Pure shear load is applied



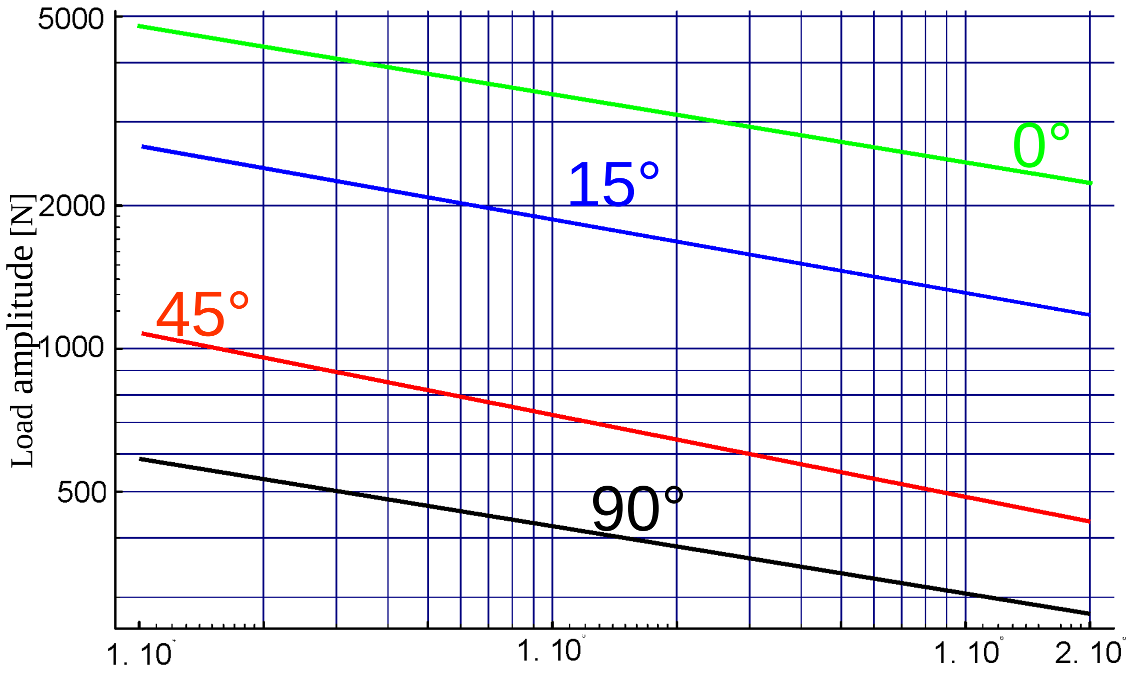


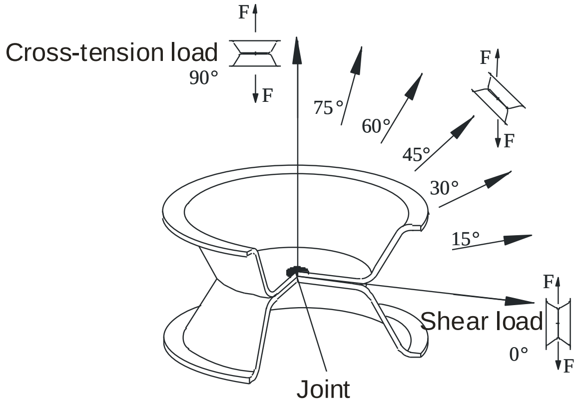




Depending on the loading type different S/N curves are used for analysis.

Selection of the appropriate S/N curve



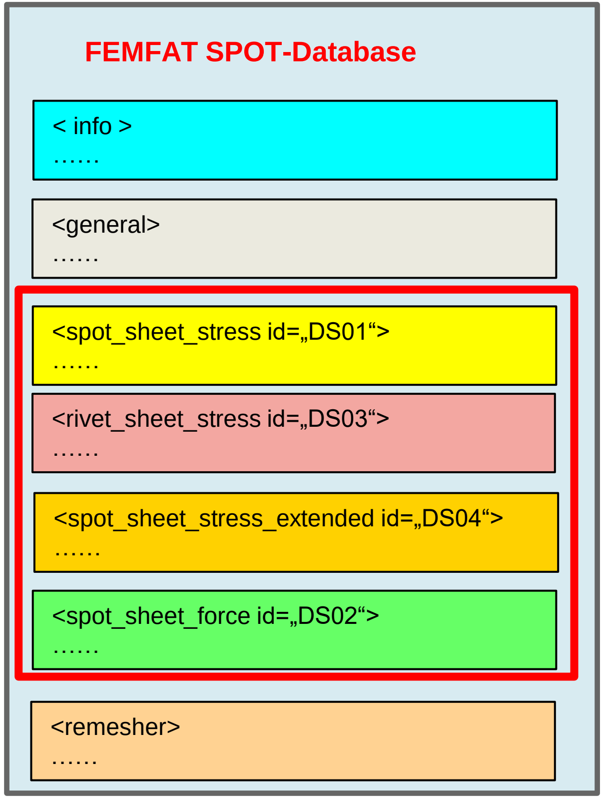


Material :St05

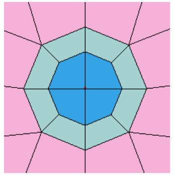
R = 0.1 s1 = 0.75 mm

d = 5 mm s2 = 1.5 mm

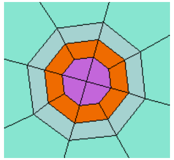
Concept of FEMFAT spot database



- **SPOT stress based assessment:**
  The FEMFAT analysis is based on the stresses of the outer nugget elements.

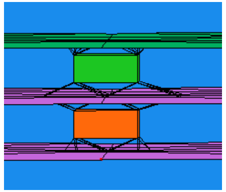


- standard spot nugget **<spot_sheet_stress>**
- standard rivet nugget **<rivet_sheet_stress>**



- extended spot nugget **<spot_sheet_stress_extended>**

- **SPOT force based assessment:**
  The FEMFAT analysis is based on the forces of connection elements.



- spot elements for force based assessment (CBAR,CBEAM, CHEXA, CWELD, soon: Abaqus Fastener) **<spot_sheet_force>**

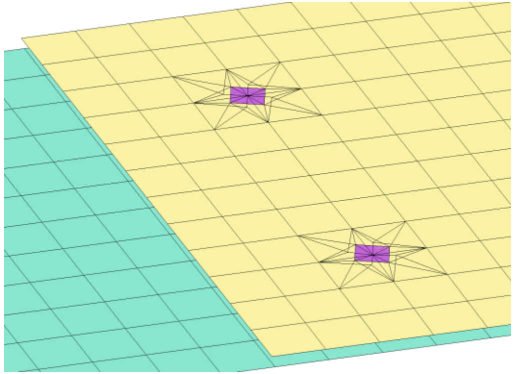
SPOT – Super Elements

Detailed FE-model leads to accurate stiffness and stress results. Assessment of stresses with one master S-N curve independent from load type and direction.

MAGNA

Workflow: Usage of Super Elements

BIW sheet structure with hexa elements connected by RBE3 to sheets

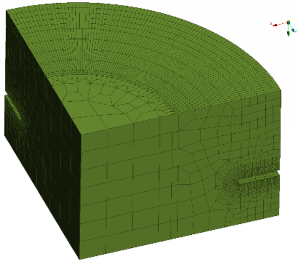


Applicable to all types of spot joints (spot welds, rivets, nails, …)

Pre-Proc

Database with super elements and stiffness matrices

Replace hexas by super elements connected by 4 nodes/sheet via RBE3



NASTRAN

Op2-file

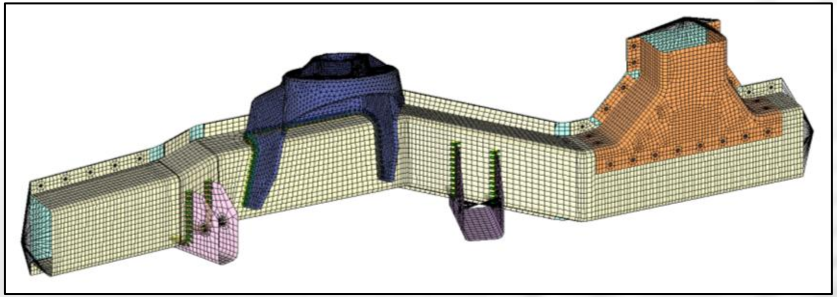
FEMFAT

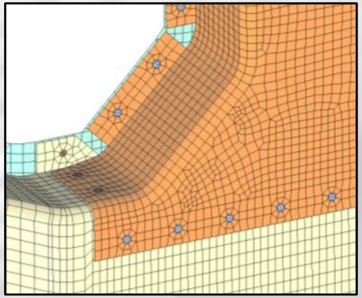
Op2-file

Post-processing

Post-Proc

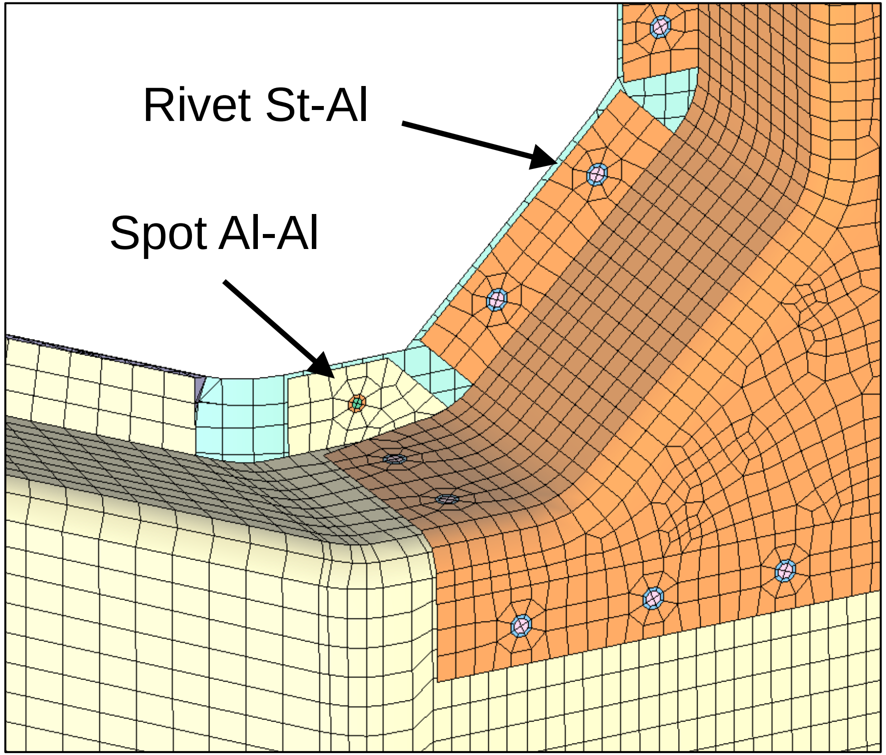
SPOT – Example

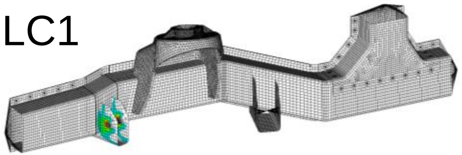
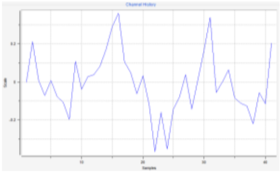
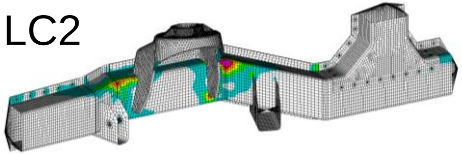
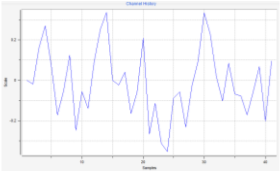
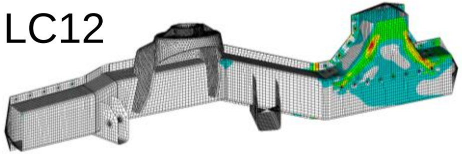
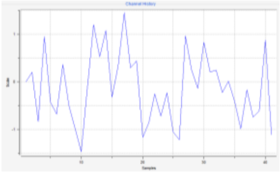




Rivets can also be evaluated with different material pairings. The new Rivet Advanced method even offers the possibility to evaluate for punch rivet failure.

Longitudinal Member: SPOT assessment under multiaxial loading

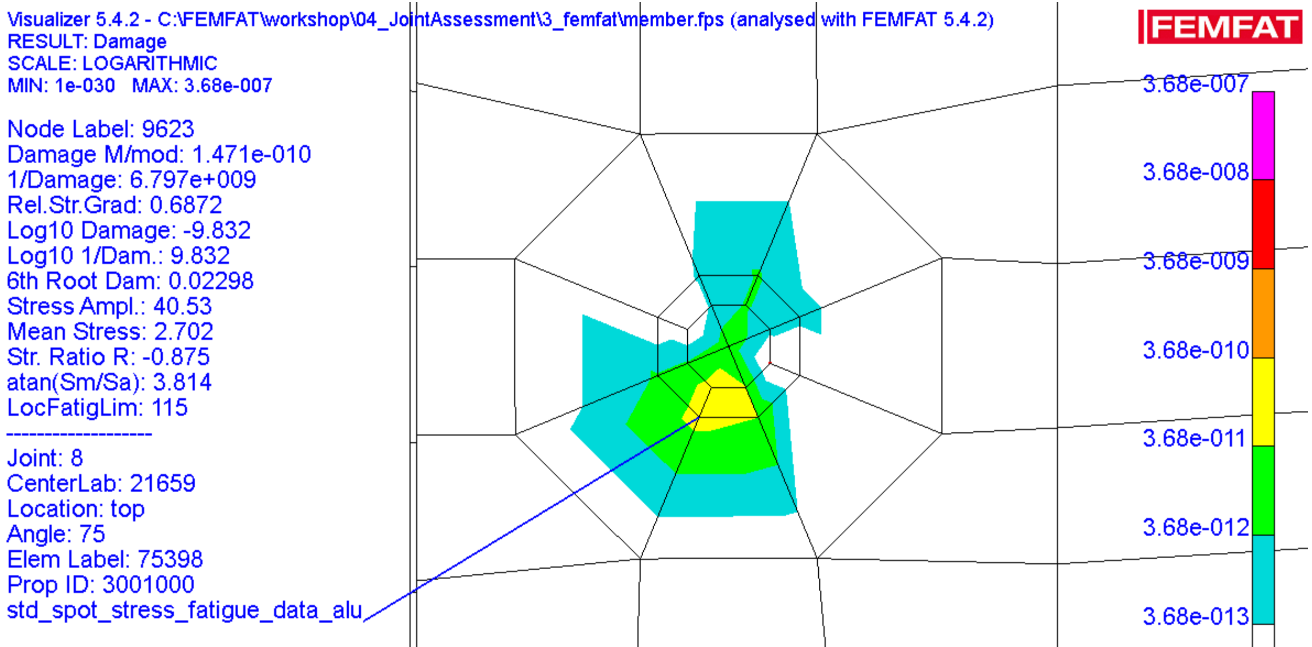


Stresses from Unit Loadcases:	Load-Time Histories:
LC1 	
LC2 	
…	…
LC12 	

Longitudinal Member: SPOT assessment under multiaxial loading

Additional Information for SPOT nodes.

The detailed output can be found in the protocol file (*.pro):




```
==========================================================================================================
Joint 8
Connector Element(s): 75374
Stress based Assessment
    NodeLab   CenterLab       Damage      Loc.   Angle  Amplitude   MeanStress     ElemLab      PropID   FatigueID
       9623       21659   1.471e-010       top    75.0  4.053e+001  2.702e+000       75398     3001000   std_spot_stress_fatigue_data_alu
==========================================================================================================
```

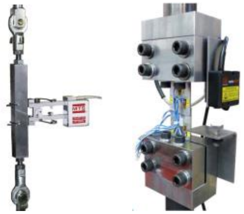
FEMFAT + ClaRP

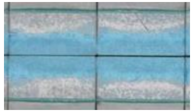
Fatigue Assessment of Adhesives

Several calibration loops were required to create the new material file for adhesives based on a master S/N curve.

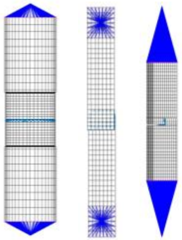
Way to Fatigue Assessment of Bonded Joints

1. Analysis Concept & Strength Data

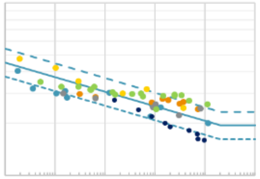




FOSTA [1] [2] [3] [4]

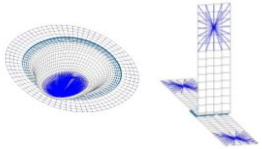


Master S-N curve:

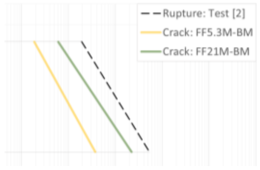


2. Validation

FOSTA [1] [2] [3] [4]



Validation S-N curves:



Tests have shown that the fatigue analysis leads to conservative results in locally stressed areas. Therefore, a new method for damage assessment was developed.

MAGNA

Workflow for Fatigue Analysis and Damage Assessment

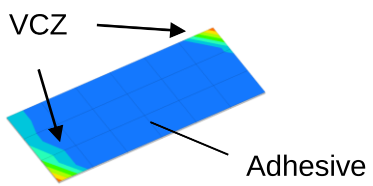
FE Model — e.g. BIW → 1310 Adhesives in BIW

↓

FE Analysis — linear, static

↓

Fatigue Analysis — FEMFAT → 180 Virtual Crack Zones (VCZ) at 133 Adhesives with $D_{max} > 1$



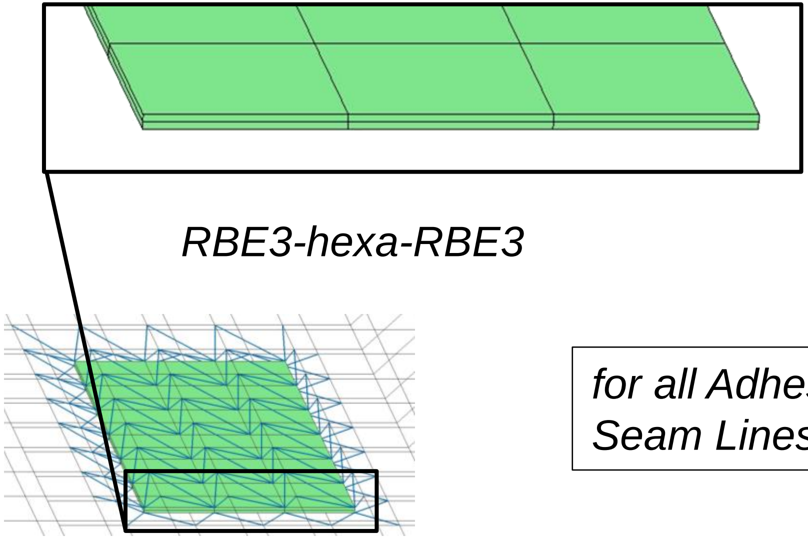
↓

Damage Assessment — *ClaRP* → **20 critical VCZ with $a_V > 1$**

The groups shown here are used in FEMFAT. After fatigue analysis, adhesive layers must be defined for ClaRP using element sets.

FE Mesh & Groups

Meshing: *by ANSA Connection Manager, HyperMesh Connector, …*



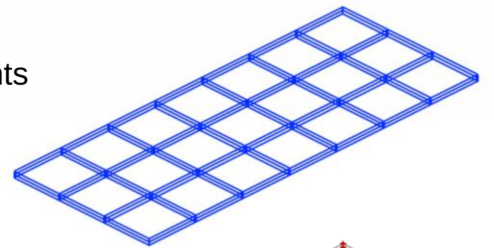
RBE3-hexa-RBE3

for all Adhesive Seam Lines

Generate the Adhesive Sets:

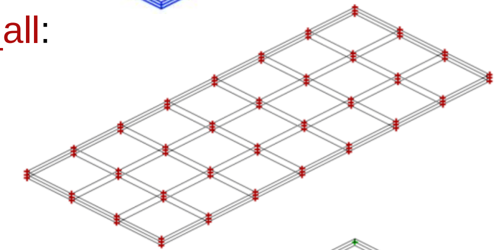
Adh_ELE:

Hexa elements of adhesive



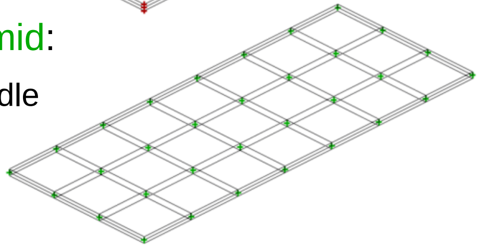
Adh_GRI_all:

All nodes of adhesive



Adh_GRI_mid:

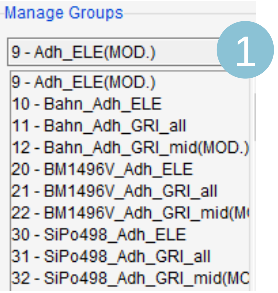
Nodes in middle plane of adhesive



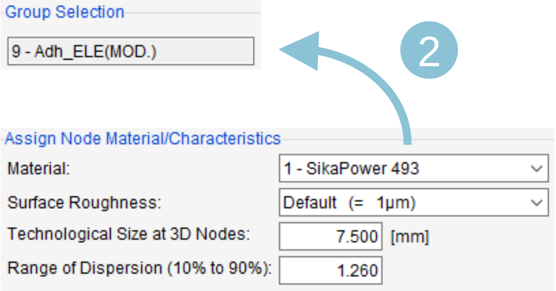
Adhesives are analyzed in the middle plane of the layer. Furthermore, specific settings and material files must be used.

FEMFAT – Minimum Input & Analysis Run

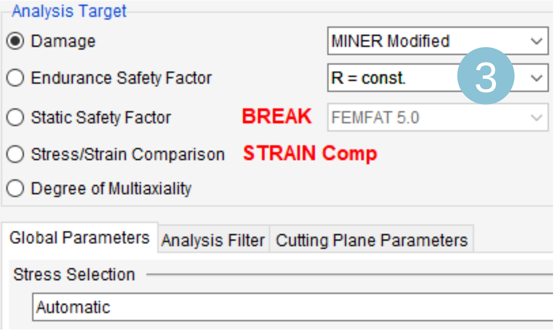
Groups:

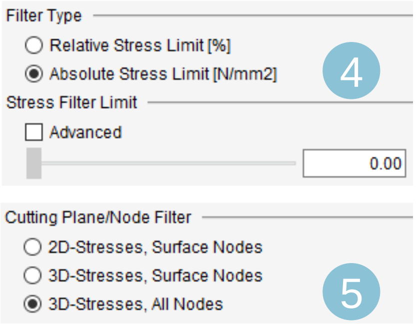


Node Characteristics:



Analysis Parameters:





1. Add Nodes of *_Adh_GRI_mid to Elements of Adh_ELE to create the Analysis Group

   *Add *_Adh_ELE to *_Adh_GRI_mid for separate analysis of the adhesives*

2. Assign the available Material SP493_FEMFAT54_ClaRP211.ffd to the Analysis Group

3. Select R = const

4. Set Abs. Str. Limit to 0 N/mm2

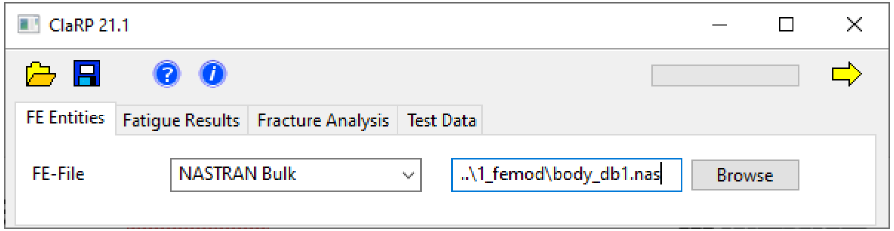
5. Select 3D-Stresses, All Nodes (for MAX only)

ClaRP has a GUI mode for creating new jobs and performing the analysis. If job files are available, batch mode can be used for process automation.

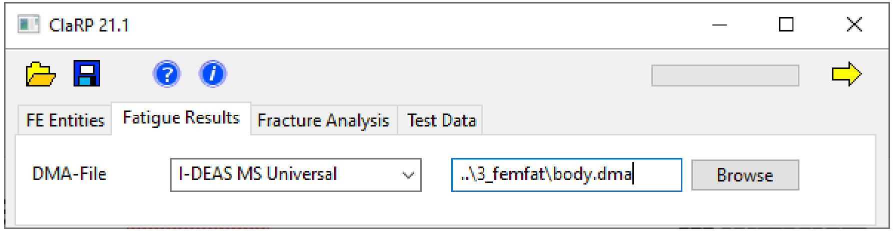
ClaRP – Minimum Input & Analysis Run

GUI mode:

1. *Select FE Entities:*

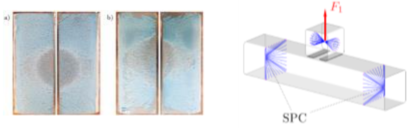


2. *Select Fatigue Results:*



3. *Save & Run the job*

Material parameters from test are saved in the ClaRP database:



Batch mode:

Job File (.crp):*

```
Version: ClaRP 21.1

# FE
FE_File: ..\..\1_femod\example.nas          # file path

# FF
FF_File: ..\..\3_femfat\max\example.dma          # file path
```

Run the job (.bat, *.que, …):*

```
rem  -----------------------------------------------------
set CP=CALL "C:\path2\software\clarp\win\ClaRP.exe"
rem  -----------------------------------------------------
%CP% --crp C:\path2\projects\project1\job1.crp
%CP% --crp C:\path2\projects\project2\job1.crp
```

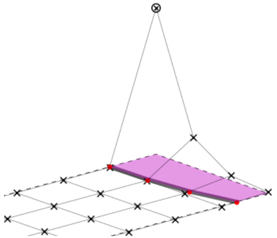
The ClaRP report file gives a clear overview of the critical areas. The output can be sorted by different results.

Assessment result in the report file (*.rrp)

Computation time: 14s

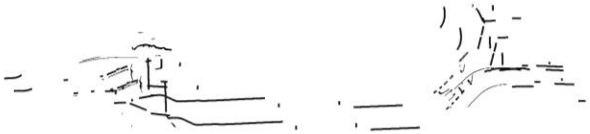
```
-----------------------------
A N A L Y S I S   R E P O R T
-----------------------------

Program                 : ClaRP
Version                 : 21.1
Date                    : Di Apr 13 16:31:22 2021
Analysis Content        : Adhesive Layer Assessment on FE-Structures
                              ... based on Fatigue Analysis Results from FEMFAT
Notice                  :
Comment                 :


General Input Data
==================

ClaRP Input
  C:\FEMFAT\body\4_clarp\body.crp

ClaRP Database
  C:\FEMFAT\body\4_clarp\body.drp

ClaRP Allocation Table
  C:\FEMFAT\body\4_clarp\body.drpa

FE Input
  C:\FEMFAT\body\1_femod\body_db1.nas

FEMFAT Result
  C:\FEMFAT\body\3_femfat\body.dma

Status
  Analysed Adhesive Layer Planes
    Fatigue Analysis ............................. Middle Plane
    Fracture Assessment .......................... Middle Plane
  Scaling .......................................... Linear
  Report
    Zones per Layer ..............................      All
    Sorting ...................................... Utili. Deg.
  Number of Load Cycles ..........................        1

Limits
  Damage from Crack Initiation ...................      1.0 [-]
```

```
Virtual Cracked Zones Data
==========================

Virtual Cracked Zones Types      ... combined types are possible
  ci ... D > 1 at i corner nodes
  ei ... D > 1 at i edge nodes
  mi ... D > 1 at i middle nodes
```



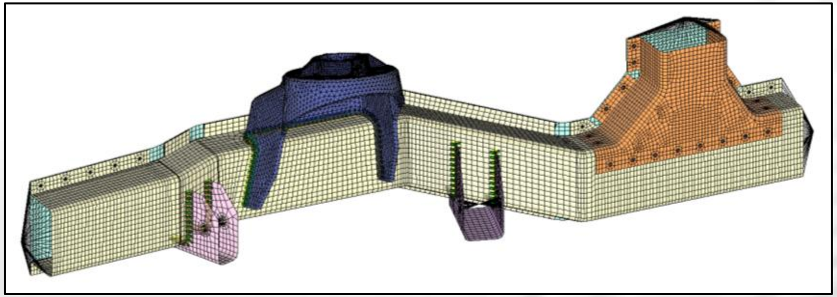
In Scope

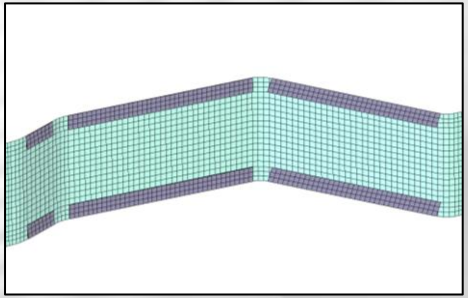
Adh. ID	Zone ID	Zone Type	Dmax	Node ID(Dmax)	Asse. Mat.	Equiv. Len.	Utili. Deg.	ClaRP Res.
85	3	c2e3	1.578E+04	4260471	SP493	3.53003E+00	1.66476E+00	not OK
117	3	c2e3	1.057E+04	4363734	SP493	3.51498E+00	1.65767E+00	not OK
2	1	c2e2	2.378E+02	4398927	SP493	3.33234E+00	1.57153E+00	not OK
121	1	c2e2	4.228E+02	4474239	SP493	3.22838E+00	1.52250E+00	not OK
86	1	c2e2	2.752E+01	5477385	SP493	3.21192E+00	1.51474E+00	not OK
12	1	c2e3	4.935E+03	5572658	SP493	3.14312E+00	1.48229E+00	not OK
42	1	c2e3	3.153E+03	5570711	SP493	3.13976E+00	1.48071E+00	not OK
89	3	c2e2	2.231E+02	4328595	SP493	3.01723E+00	1.42293E+00	not OK
118	1	c2e2	8.232E+01	4435071	SP493	2.90110E+00	1.36816E+00	not OK
40	1	c2e2	1.911E+01	5616679	SP493	2.89957E+00	1.36744E+00	not OK
116	1	c2e2	1.083E+01	4356714	SP493	2.77718E+00	1.30972E+00	not OK
33	1	c2e2	1.967E+01	5643278	SP493	2.76410E+00	1.30355E+00	not OK
7	1	c2e5	4.102E+02	5641541	SP493	2.75993E+00	1.30159E+00	not OK
46	1	c2e5	1.902E+02	5643488	SP493	2.42202E+00	1.14222E+00	not OK
15	3	c1e2	3.098E+02	4329963	SP493	2.36398E+00	1.11485E+00	not OK
49	1	c2e2	4.111E+01	5570918	SP493	2.36396E+00	1.11484E+00	not OK
124	3	c2e2	7.393E+02	4388406	SP493	2.35127E+00	1.10886E+00	not OK
50	5	c1e2	3.005E+01	4440378	SP493	2.26355E+00	1.06749E+00	not OK
45	2	c1e4	7.585E+01	5643932	SP493	2.18922E+00	1.03244E+00	not OK
6	1	c2e6	9.527E+01	5642828	SP493	2.18041E+00	1.02828E+00	not OK
50	3	e4	1.598E+02	4440198	SP493	2.05837E+00	9.70725E-01	OK



20 critical Zones with $a_V > 1$

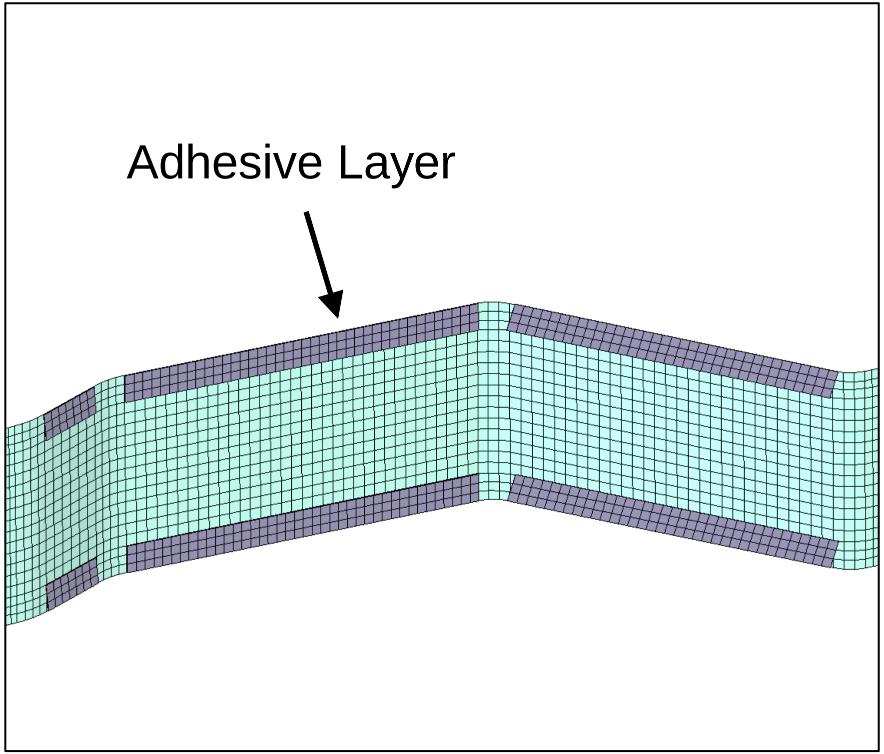
Adhesives – Example





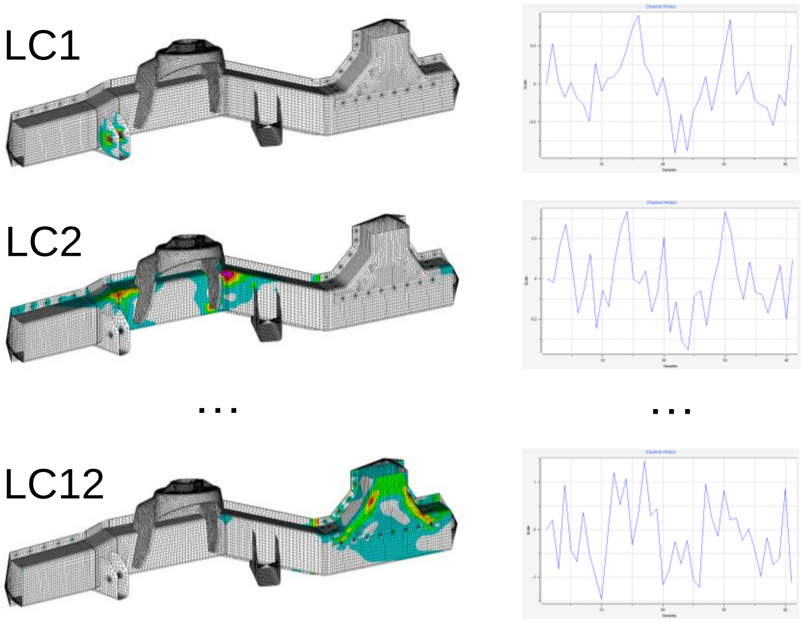
Some adhesives have very different material properties. The linear static values used in the FE analysis should be consistent with FEMFAT.

Longitudinal Member: Adhesive assessment under multiaxial loading



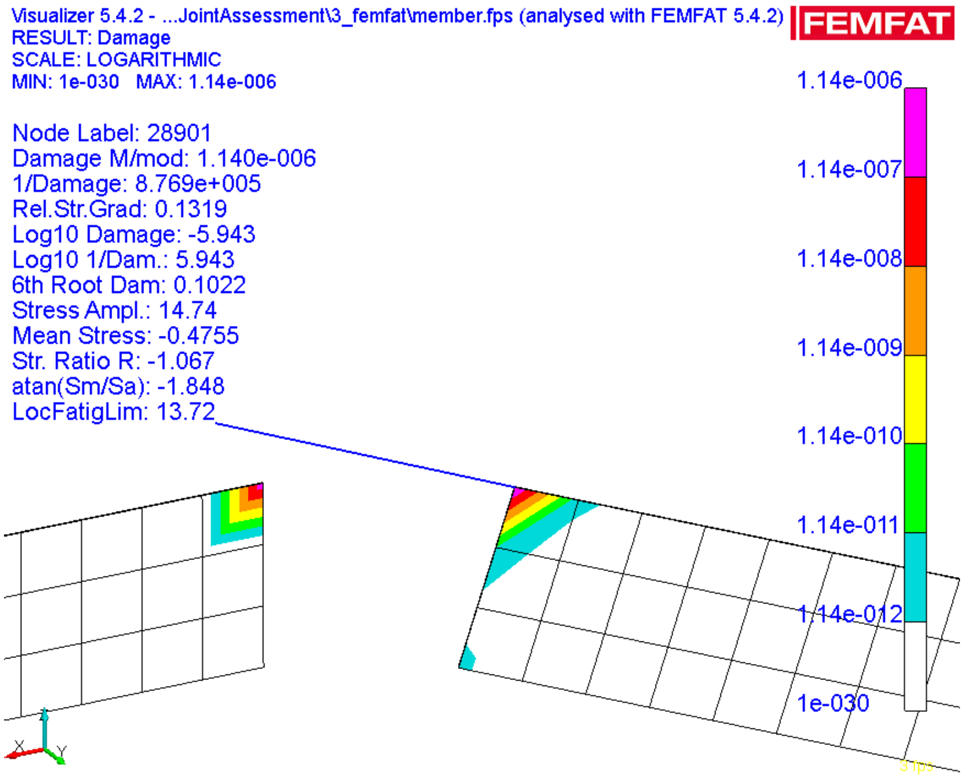
Stresses from Unit Loadcases:

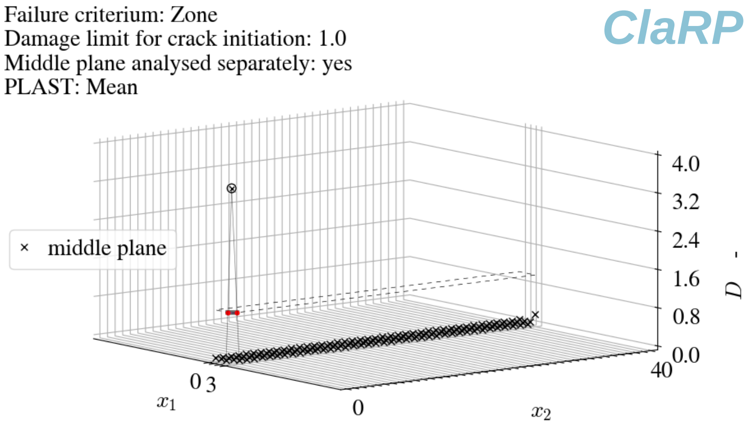
Load-Time Histories:

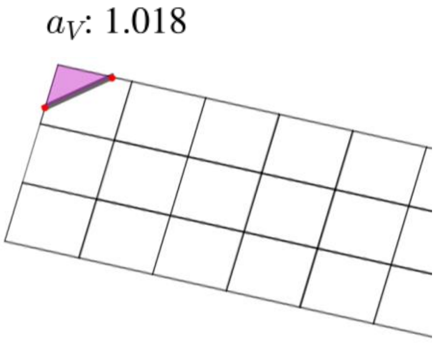


For the evaluation of adhesive layer nodes, detailed results can be requested in FEMFAT. Repetition Factors can be considered in ClaRP.

Visualization & Detailed Investigation







Degree of Utilization

$a_V > 1$

$(f_{rep} = 5e6)$

→ critical!

Non-metal Fatigue

Assessment of short fiber reinforced plastics

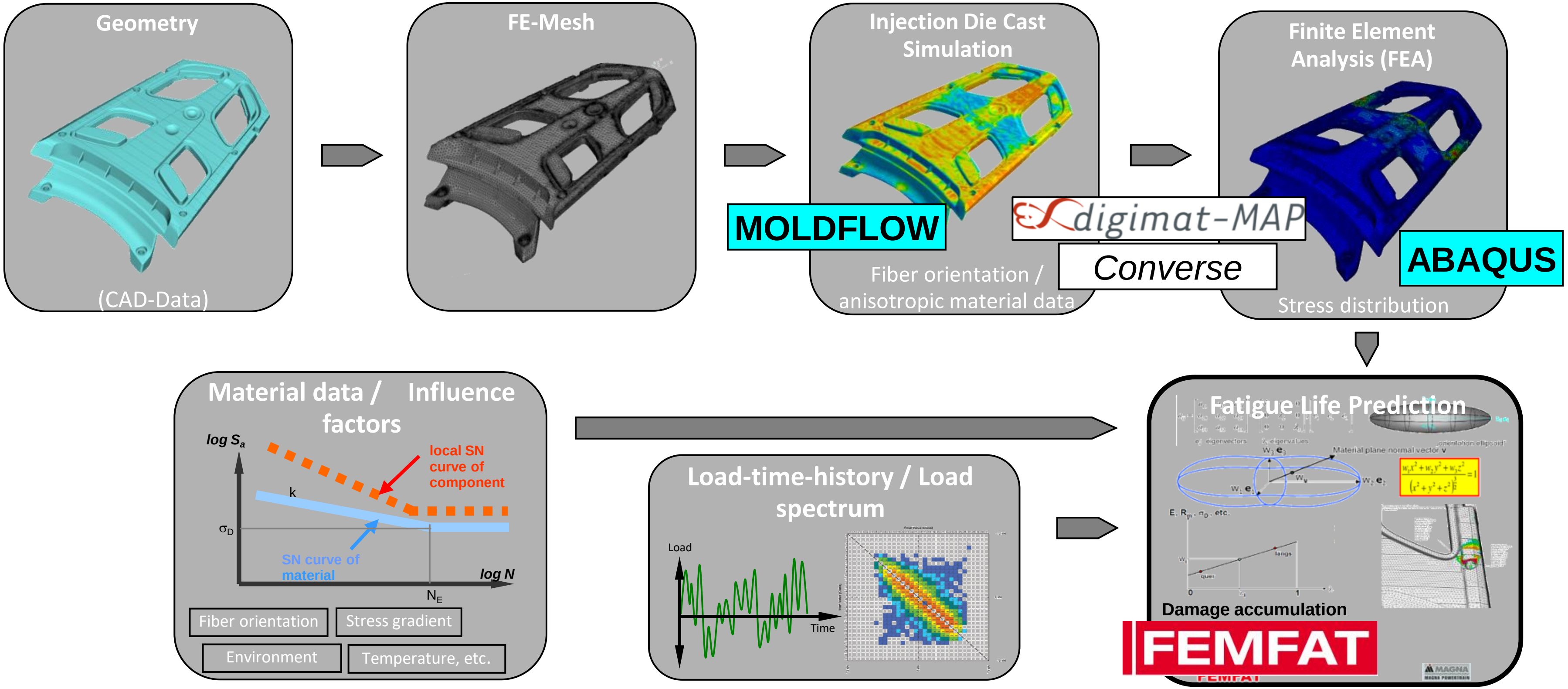
in MAX

in SPECTRAL (from 5.4.1)

Since a separate mesh is used for the injection molding simulation, the fiber orientations must be mapped onto the FE mesh before the FE analysis.

Motorcycle Luggage Rack – Workflow

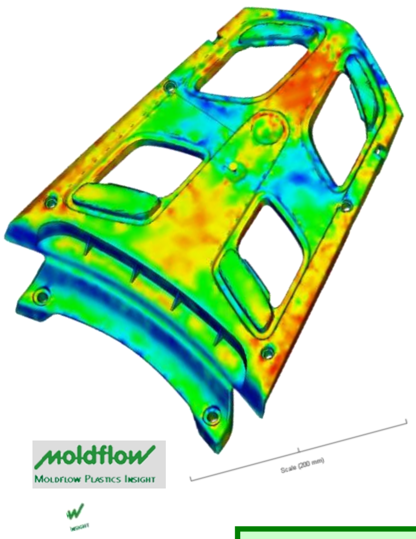
with courtesy of BMW

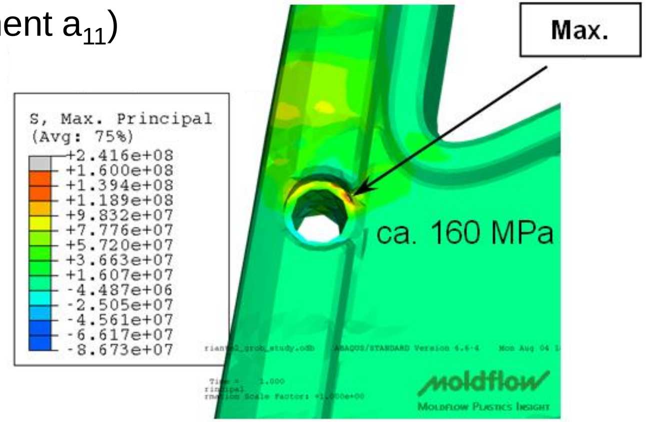


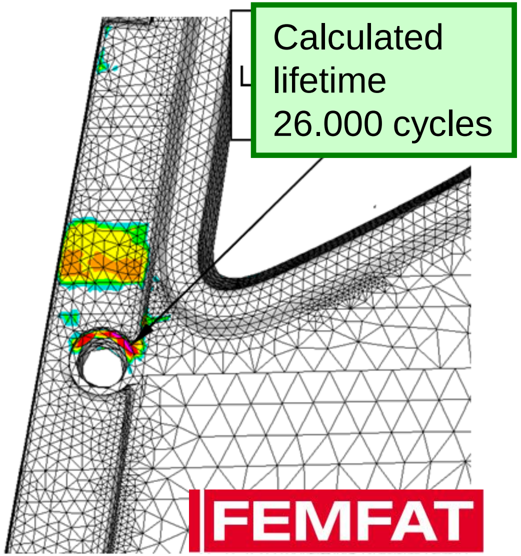
Motorcycle Luggage Rack – Results

- FE-Model for fill simulation: ~ **1,3 Mio. Elements**
- FE-Model with mapped data: ~ **400.000 Elements**

Fiber Orientation Tensor (Component a_{11})







Test of five components yielded load cycles between 46.000 und 96.000 until crack initiation.

Analysis without considering anisotropy delivers 2.000.000 cycles ⇒ 30 times too optimistic!

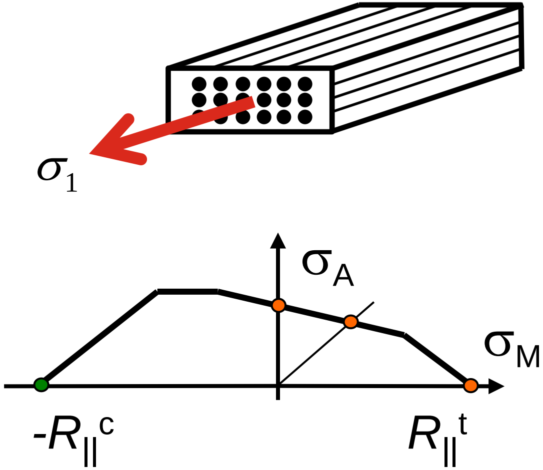
FEMFAT laminate

in channelMAX

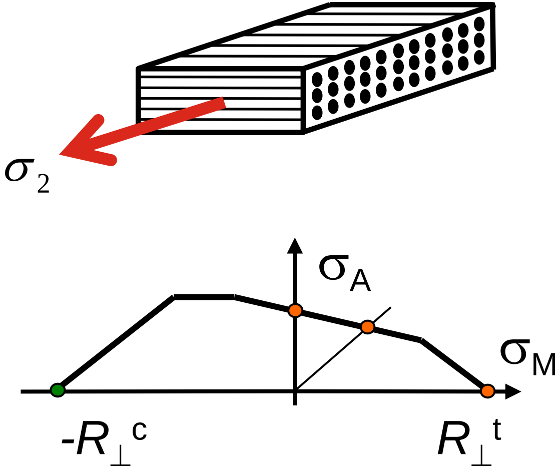
The fatigue assessment need to be done in several directions and therefore all this material data must be known from test.

Fatigue assessment of different load directions

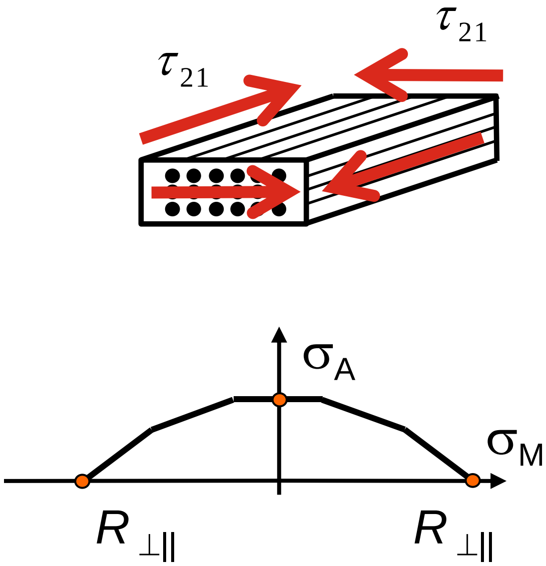
Fatigue Assessment for Fiber Fracture (FF)



Fatigue Assessment of Normal Stress for Inter Fiber Fracture (IFF)

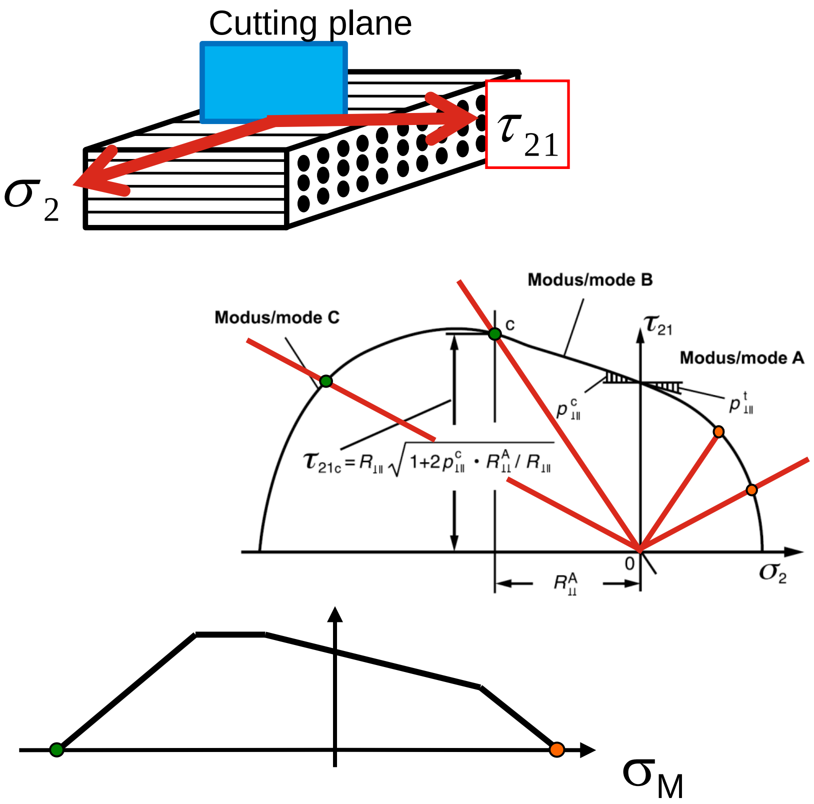


Fatigue Assessment of Shear Stress for Inter Fiber Fracture (IFF)



LAMINATE can only be used just in the combination of Abaqus (SHELL/ SOLID SECTION, COMPOSITE) and channelMAX.

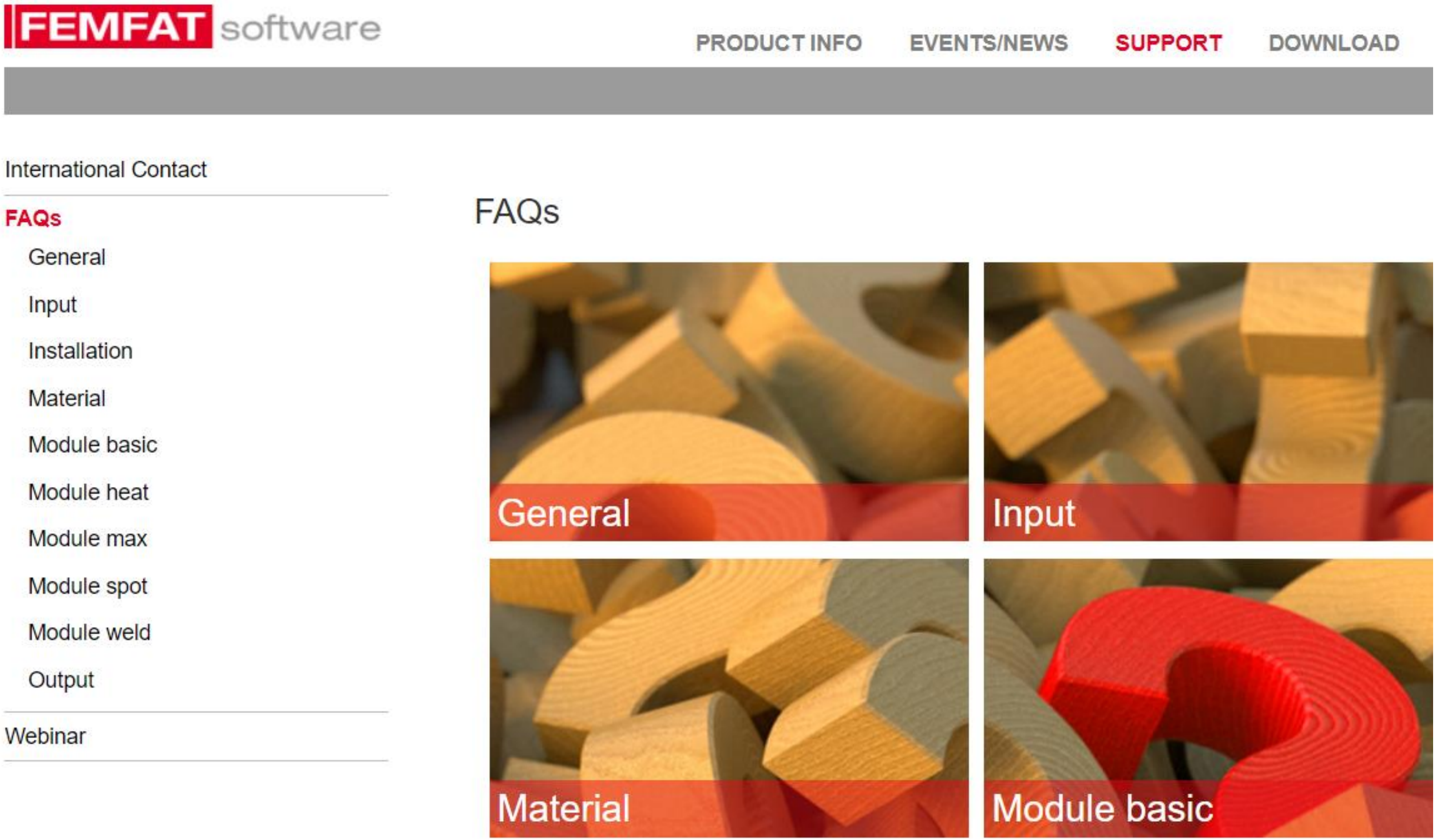
Fatigue assessment of different load directions in plane



- Necessary material data for fatigue analysis:
  - S-N curve is interpolated between normal and shear
  - Static strength depend on load direction and are taken from Puck's curve
  - Haigh-diagram is interpolated between normal and shear
- Input number of load directions
- Rainflow counting of stress vector projected on each load direction (red lines)
- Linear damage accumulation for each load direction.
- Additional parameters $p_{\perp||}^{t}$ and $p_{\perp||}^{c}$ have to be specified, default values for CFK acc. VDI 2014:
  - $p_{\perp||}^{t} = 0.35$
  - $p_{\perp||}^{c} = 0.3$

Additional Information

FEMFAT Homepage www.femfat.com



FEMFAT Homepage www.femfat.com

MAGNA

FEMFAT software

PRODUCT INFO | EVENTS/NEWS | SUPPORT | DOWNLOAD

FEMFAT
- Software & Release Notes
- Documentation
- **Getting Started**

FEMFAT LAB

Papers

Flyers

Getting Started

BASIC, HEAT, SPECTRAL
- BASIC
- HEAT
- SPECTRAL

MAX
- MAX Transient
- MAX channel

MAGNA

DRIVING **EXCELLENCE.**
INSPIRING **INNOVATION.**