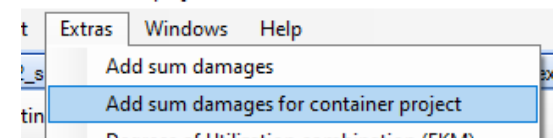
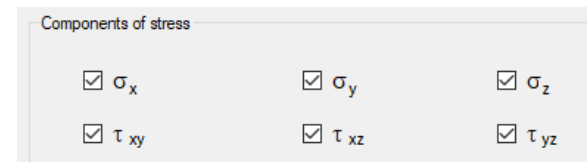
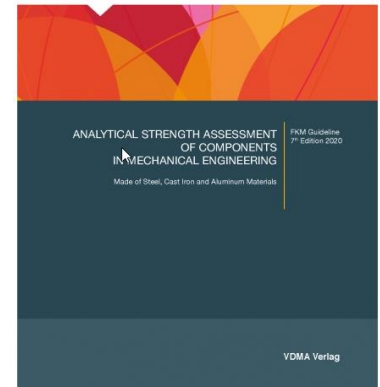




# Overview of innovations in winLIFE 2022

## – User interface

- **Calculation according to FKM guideline, 7th edition (year 2020) implemented**
- **Material database updated according to FKM, 7th edition**
- **Under calculation parameters / rough analysis additional parameters max\_schad.kno for the number of nodes (in the max\_schad.kno file). The size of the max\_schad.kno file is to be defined independently (parameters of the rough analysis)**
- **Container projects: addition of the damage sum for the entire project can be done with one command**
- **More flexible filters are available for selecting an FKM SN-curve or a user SN-curve**
- If load changes occur with an extrapolation > 10e9, the number of cycles is increased accordingly
- Adoption of the node attributes of the container project calculation into the service factor calculation
- With the ZM, the equivalent stress hypothesis 'normal stress with relevant main stress' can only be selected for solid elements when forming the hysteresis loop
- **It is now possible to select individual stress components for the calculation**
- A conventional node selection is again possible via the \* .kno file without having to go through the Viewer





# Overview of innovations in winLIFE 2022

## – Viewer4winLIFE

- Unused node sets can now be edited and exported in the result Viewer
- **Generation of stress gradients possible with imported LST-files, additional stress gradient is implemented at the plate (shell) edges (see image slide 4)**
- The element line setting is now saved in the Viewer after closing
- The command show / hide now also appears when opening the Viewer under result
- Non-linear and linear ADINA files can be read in via the Viewer
- When the Viewer is closed, a message appears asking whether changes should be discarded
- Update of the triad display for welded components

## – FE-Interface

- **Interface Viewer - RecurDyn – Non-linear interface has been optimized.**
- **RecurDyn - Non-linear load steps can be selected individually with start value, end value, step size.**
- PERMAS post-files can be used in the Random Module
- Differentiation between the 2 ANSYS version in the Viewer when reading in deformations. Function changed respectively improved
- ANSYS files with results of arithmetic operations can now be read
- Update of the FEMAP interface for FEMAP 2021.2



# Overview of innovations in winLIFE 2022

- Solver
  - Dynamic modulation with 7 decimal places
  - **Speed optimization when calculating with non-linear LST files**
  
- FKM
  - Several SN-curves can be used in the FKM module
  - In the FKM protocol, the SN-curve used is output with parameters
  - When calculating the FKM (module FKM), you can select whether 2D or 3D stresses are used for the calculation
  - Safety factor settings are saved
  - The input option for over-/under load was removed, as it repeatedly led to confusion
  - In the case of static proof, the internal verification is carried out for both the under stress and the upper stress. After that, the greatest level of utilization is displayed and used.
  - FKM edition 7: Decisive for the degree of utilization is only the comparative degree of utilization for multi-axis, proportional and synchronous stresses. See FKM Section 4.6.3. If edition 6 is used, this is not yet taken into account.
  
- Additional
  - Saving the materials in the library is now possible with the FKM module even without the BASIC module
  - LST-file generation (CDI) optimized
  - Latest Microsoft database (LocalDB 2019)
  - Revision of example 22 of the winLIFE help
  
- **see chapter 10 of the installation instructions**



# Further Functions Stress Gradient

FE Interface - winLIFE LST-File (new format)

LST-files of stresses

Index	FE Stress File	FE Force
1	FE_Lc_1_example_7.lst	100
2	FE_Lc_2_example_7.lst	100

Path : \_\_\_\_\_

Use stress gradient   
  Create stress gradient   
  On edges of PLATEs   
  Create 2D stresses

the related stress gradient is taken into account in the service life calculation

the related stress gradient is determined from the stresses in the LST file

optionally for plates (shells), the stress gradient can be evaluated in the planar direction