LIFING

Current Release: 3.0

OUTLOOK

- What is LIFING

- LIFING.Life
    - **FEM INTERFACES**
    - **ANALYSIS PROCESS**
    - **IMPLEMENTED ANALYSIS METHODS**
    - **VIRTUAL STRAIN GAUGE**
    - **QUICK 2D FEM**
- LIFING.Growth
    - **ANALYSIS PROCESS**
    - **IMPLEMENTED ANALYSIS METHODS**

- Technical Information
- References

WHAT IS LIFING

- LIFING is a software for performing Fatigue and Damage Tolerance (F-DT) FEM based analysis of components.
- LIFING includes the following modules:

LIFING.Life   Multiaxial Fatigue analysis

- Stress and Strain based analysis methods (multiaxial) are implemented
- PSD based fatigue implemented
- Multiple features for post-processing are included, such as Virtual Strain Gauges, to retrieve at any point in the FEM a stress tensor time history
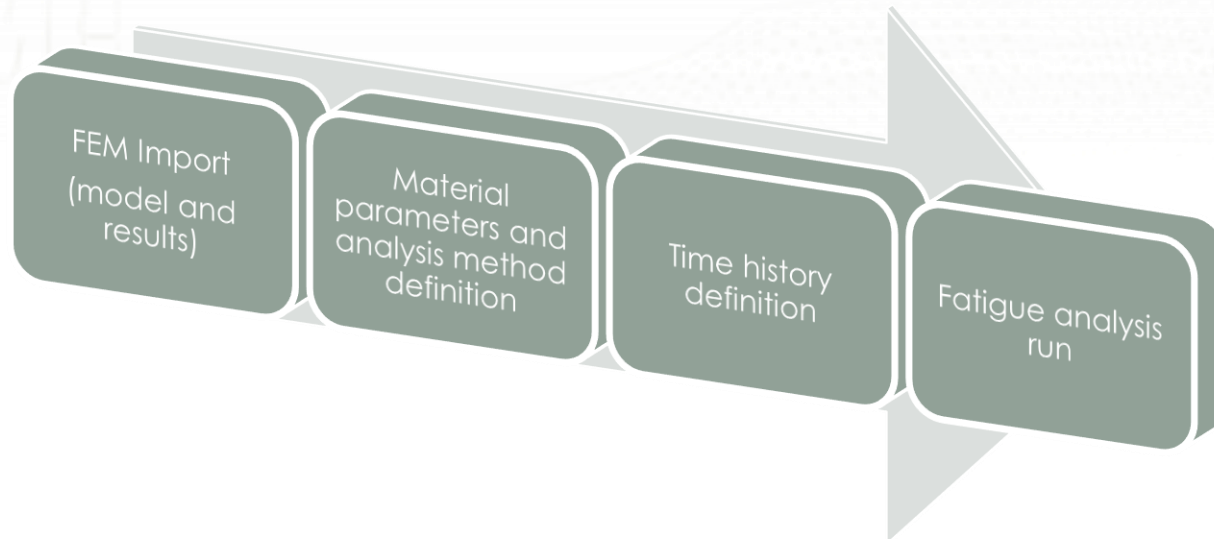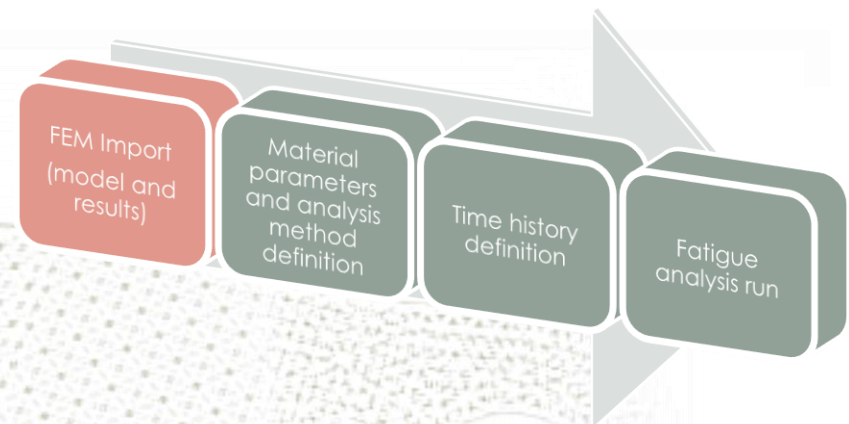
LIFING.Growth   Crack Growth analysis

- 2D FEM can be solved to extract Stress Intensity Factors (internal FEM solver and mesher; J-Integral calculated at crack tips where «quarter point elements» are used
- 3D Solid Mesh Sections can be solved (stress distributions are automatically extracted at section cuts, an initial crack is defined and Stress Intensity Factors are calculated based on Newman-Raju solution corrected with Glinka's Weight Functions
- NASGRO model used for integrating Crack Growth Life curve

LIFING.Life

- LIFING calculates fatigue, in general crack initiation, in a FEM component or assembly
- The analysis can be conducted with multiple methods:
  - **Stress based methods**
 - **Strain based methods**
- The workflow:

FEM Import (model and results) → Material parameters and analysis method definition → Time history definition → Fatigue analysis run
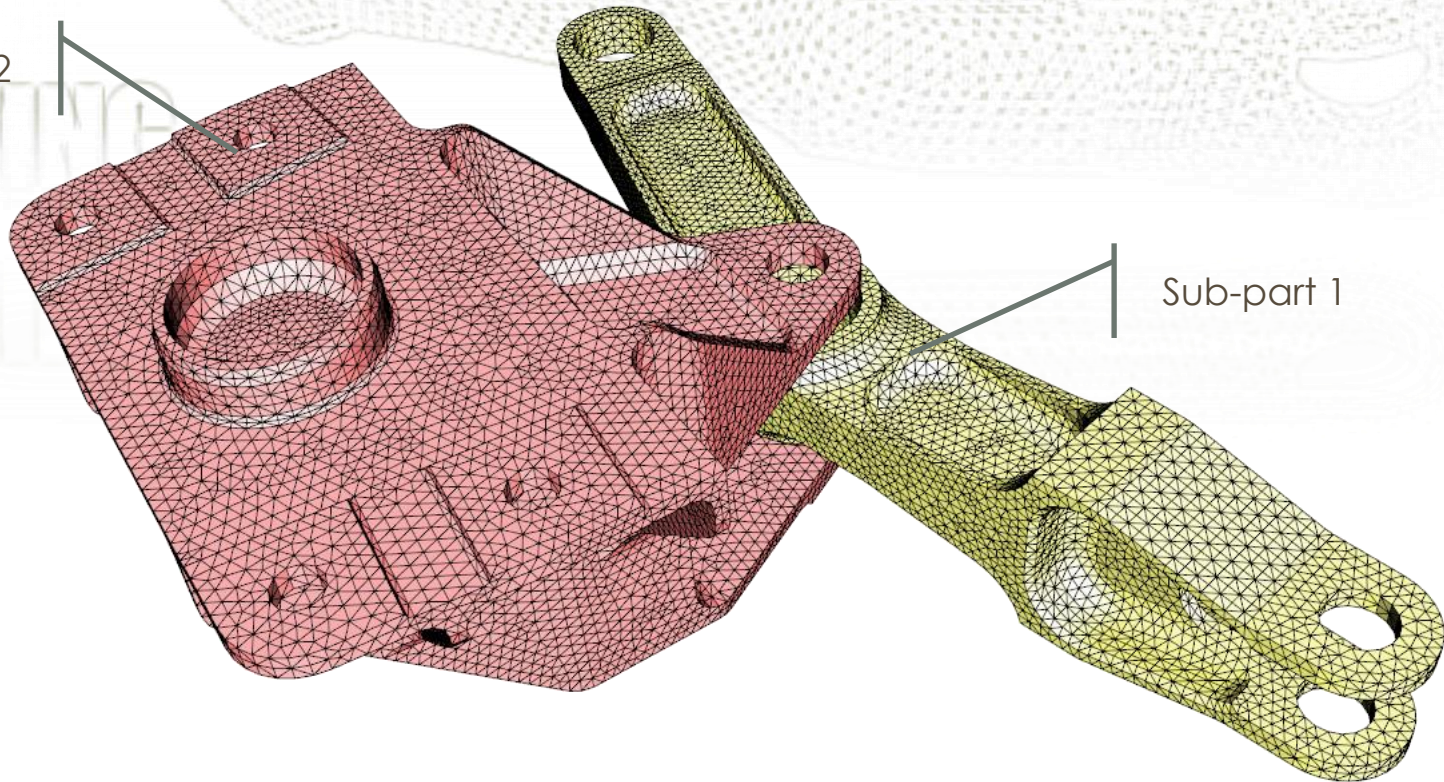
LIFING.Life

- LIFING includes FEM interfaces for:
  - **NASTRAN, OPTISTRUT**
    - Import of model (ASCII file NASTRAN format) and results files OP2 (bynary) and PCH (ASCII) generated by NASTRAN/OPTISTRUT
  - **ANSYS**
    - Import of model (ASCII file ANSYS format) and results files (ASCII) generated by ANSYS (the CDB bynary file interface is not available)
  - **ABAQUS, CALCULIX**
    - Import of model (ASCII file ABAQUS/CALCULIX format) and results files FIL (ASCII) generated by ABAQUS and FRD (ASCII) generated by CALCULIX
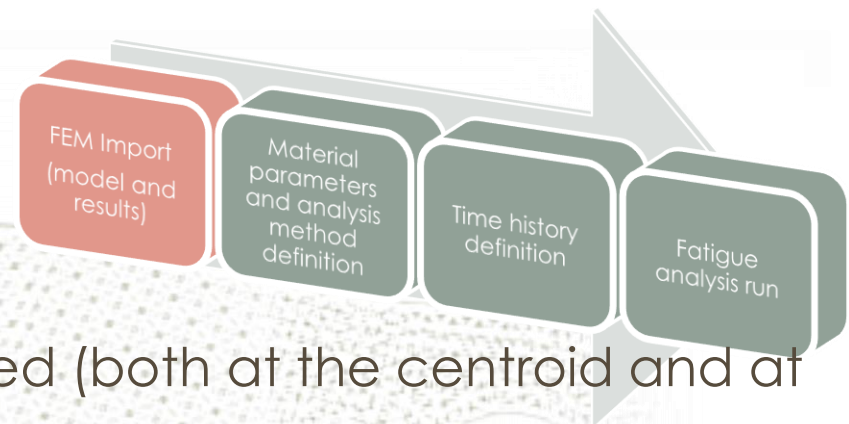
LIFING.Life
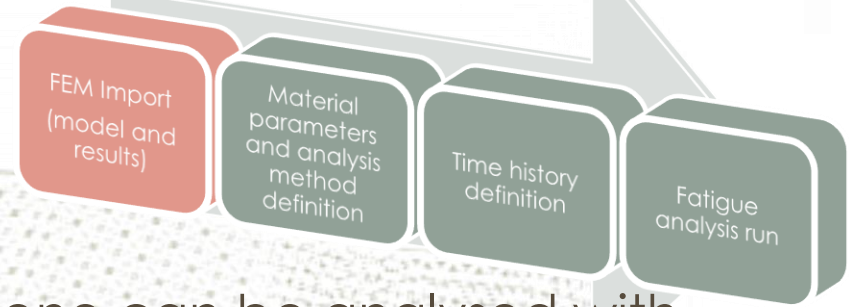
- The FEM can be split in sub-parts (each one can be analysed with different methods and/or parameters, e.g. for accounting local surface treatments)
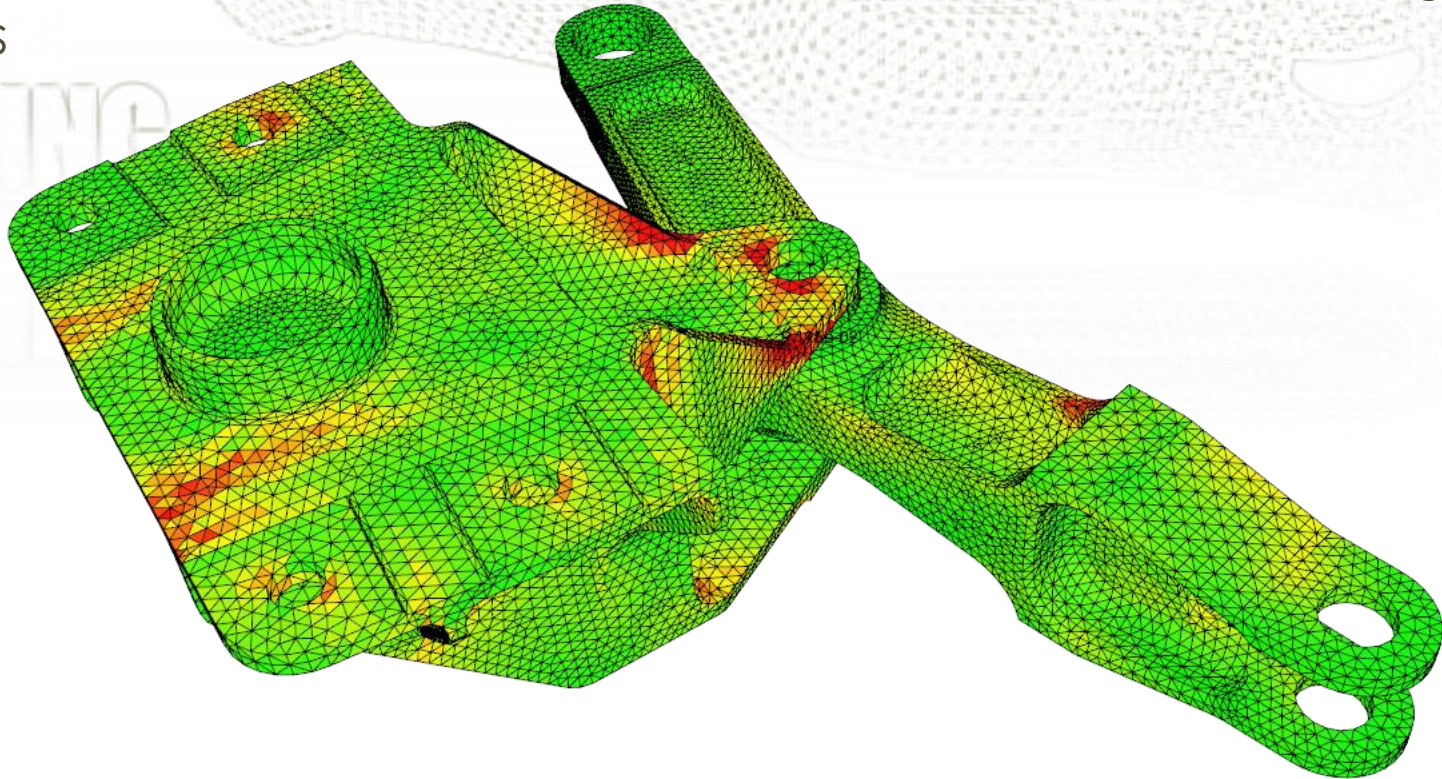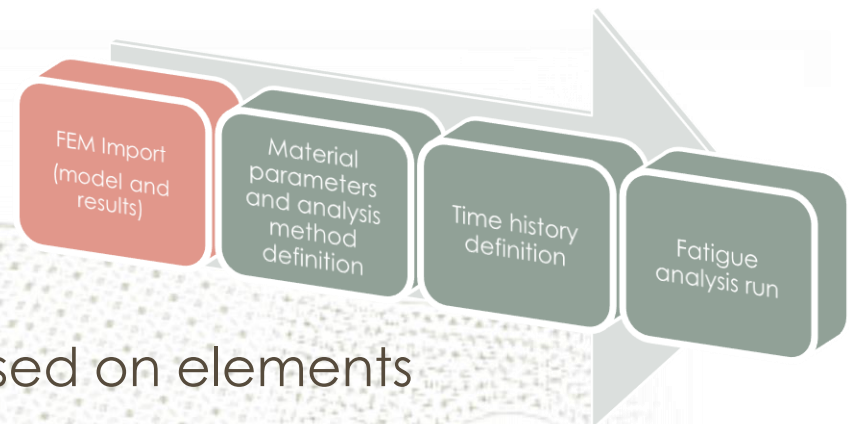
Sub-part 2

Sub-part 1

LIFING.Life

- Elemental stresses can be post-processed (both at the centroid and at the surfaces)
- Only elements and stresses at the surfaces are retained for fatigue analysis

LIFING.Life

- LIFING.LIFE performs fatigue analysis based on elements
- Stresses at facets are:
  - Either taken from centroidal stresses and rotated to local reference system with Z' axis perpendicular to the element facet
  - Either derived from nodal stresses (if available in the result file) as shown in the sketch

Internal node

A

B

Facet Average stresses (from A, B, C)

Facet on the surface

C
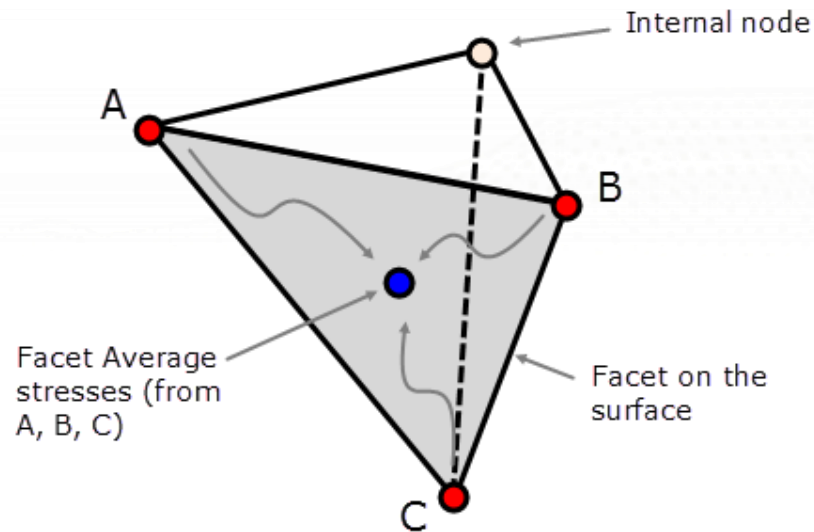
LIFING.Life
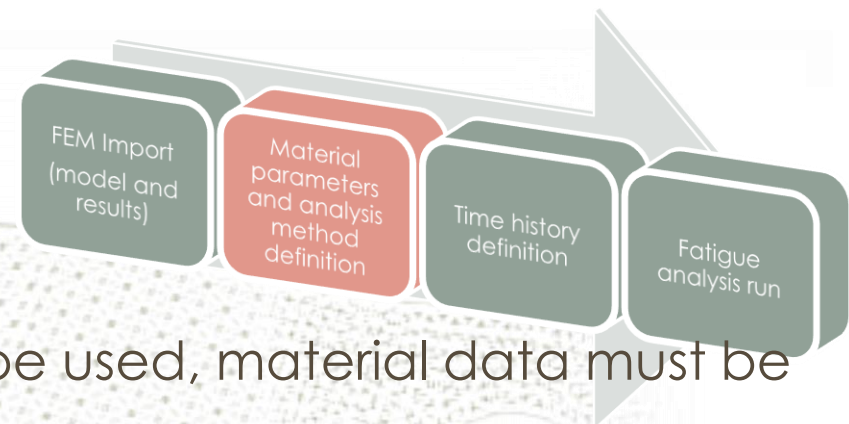
- Depending on the analysis method to be used, material data must be provided
  - S-N curves (generic format or military standard format)
  - Strain-life and cyclic stress-strain curves

- One set of material data and analysis method is assigned to the entire FEM, however different materials and analysis methods can be assigned to the FEM sub-parts
  - If nothing is specified to the Sub-parts, they will be analysed considering the 'global' material data and analysis method, otherwise...
  - ...they will be analysed considering the specified data

- Residual stress conditions can be considered
  - Constant residual stress applied to the entire FEM or differentiated by Sub-Parts
 - Stress offset from a specific static load case

LIFING.Life
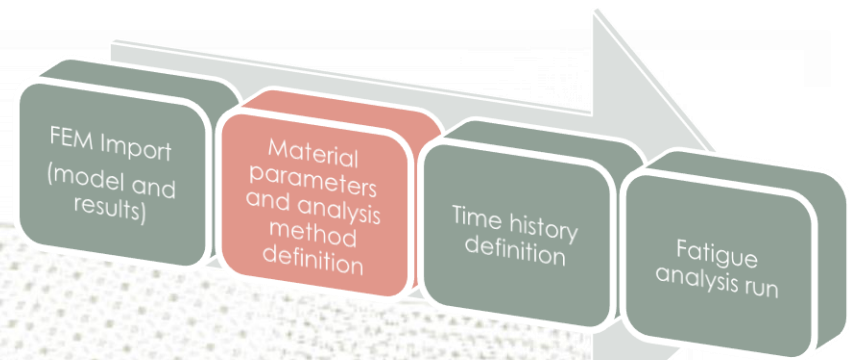
- Stress based methods
  - Uniaxial reduction (*) analysis based on S-N curves with Goodman, Gerber, Soderberg, Walker, Morrow, Smith-Watson-Topper, Haigh methods for mean stress levels accounting (it can also be decided not to consider mean stress corrections)
  - Uniaxial reduction (*) analysis based on military standard S-N curves
  - Dang-Van, McDiarmid multiaxial stress based analysis
  - Reserve Factor calculation based on Goodman-Smith diagram
  - PSD based fatigue analysis can be carried out, based on the Dirlik or Narrow Band methods

- (*) Uniaxial Reduction consists in a fatigue analysis carried out considering one stress contribution: the elemental stress tensor time history is projected at different angles and for each one the fatigue Life is calculated. The minimum Life is retained as element Life
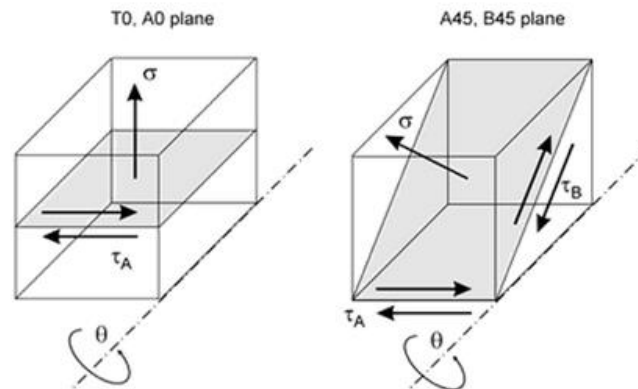
LIFING.Life
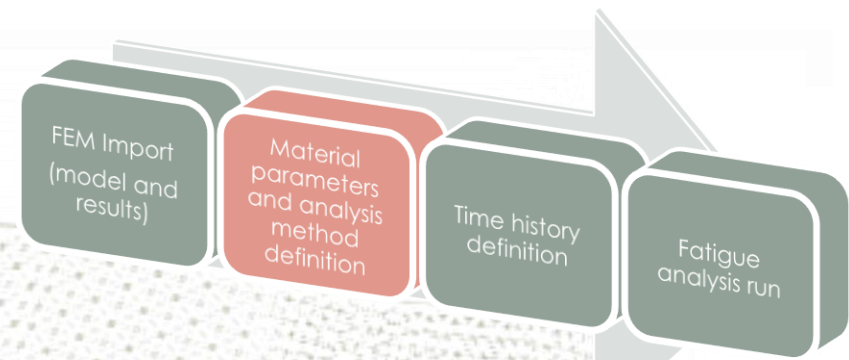
- Strain based methods
  - Uniaxial reduction analysis based on e-N curves with Morrow, Smith-Watson-Topper, Manson-Halford methods for mean stress levels accounting (it can also be decided not to consider mean stress corrections)
    - Plastic stress-strains derived with Neuber or Equivalent Strain Energy Density (Glinka) methods
  - Multiaxial strain based analysis can be performed with
    - **Hoffman-Seeger** or **Dowling** methods in case of Proportional Loading situations
    - Köttegen-Barkey-Socie **Pseudo-Material Approach** in case of Non-Proportional Loading situations
      - Elastic-Plastic stress/strain time histories are derived with the Mroz-Garud cyclic plasticity model
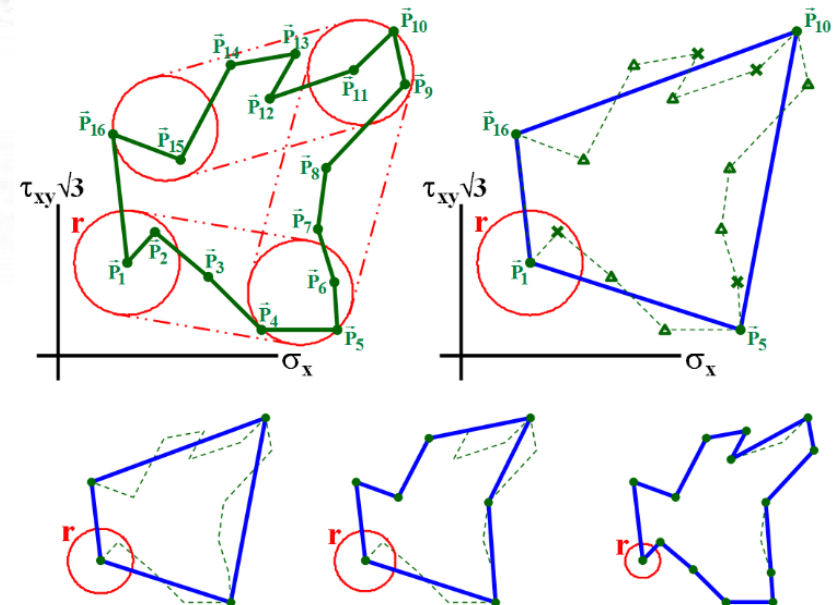
LIFING.Life
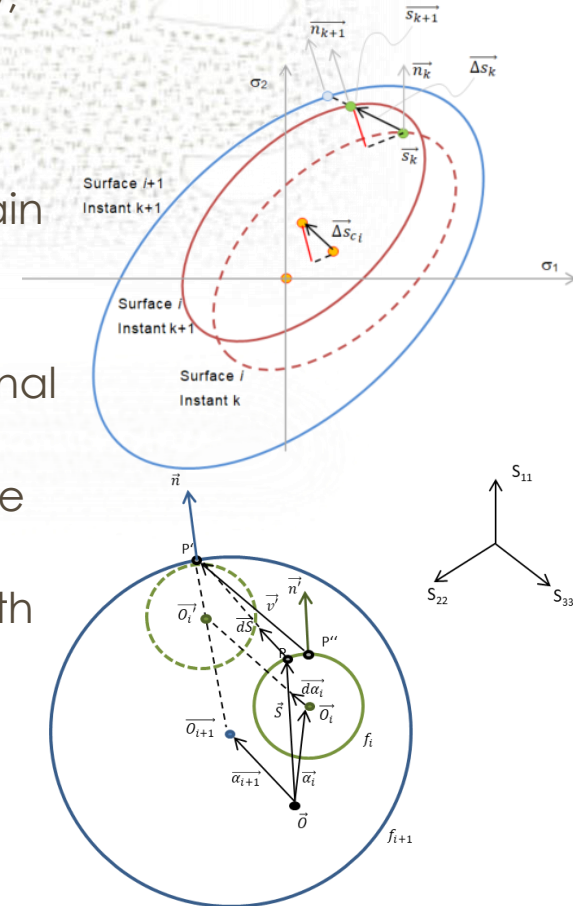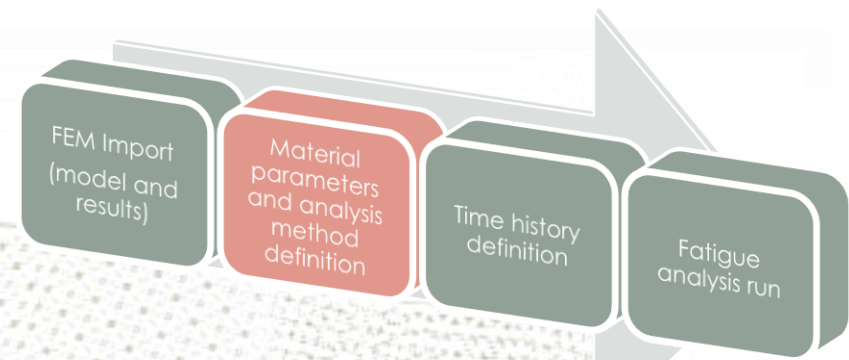
- Strain based methods
  - The user decides which method to use (it is possible for example to run the analysis of Non-Proportional Loading cases with Hoffman-Seeger or Dowling methods: in this case 'average'principal angles and biaxiality ratios are calculated and the analysis is carried out considering the stress time history projected on this plane and accounting the calculated biaxiality ratio

  - Implemented Fatigue parameters in case of Multiaxial Fatigue analysis:
    - Smith-Watson-Topper (plane T0)
    - Fatemi-Socie (planes A0, A45, B45)
    - Brown-Miller (with and without mean stress corrections) (planes A0, A45, B45)
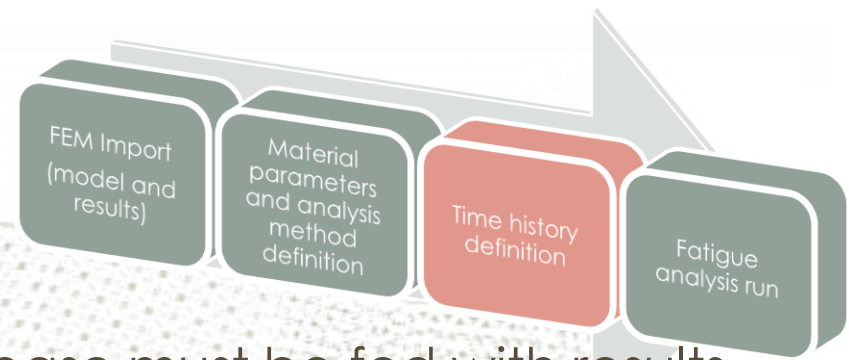

T0, A0 plane   A45, B45 plane

LIFING.Life
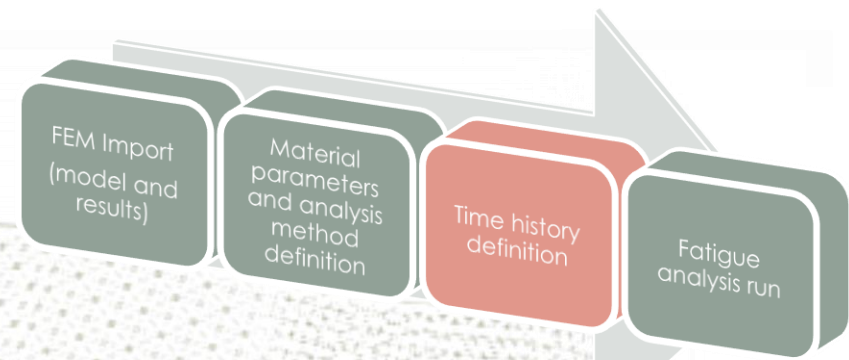
- The time history is defined on the basis of the imported result load cases
  - One sequence can be defined for each Load Case or ...
  - ... It can be defined a sequence of Load Cases combination

- The assembled element tensor time history can be filtered with a Racetrack Filter (in case of full multiaxial fatigue analysis with Pseudo-Material a multiaxial Racetrack filter is used)

LIFING.Life

FEM Import (model and results) → Material parameters and analysis method definition → Time history definition → Fatigue analysis run

- If the analysis is based on PSD the database must be fed with results from Frequency Response analysis (complex stress tensors).

- LIFING calculates the corresponding Von Mises stress, from the complex tensor, for each frequency.

- An input PSD is provided, which acts as multiplicator for the Frequency Response analysis results.

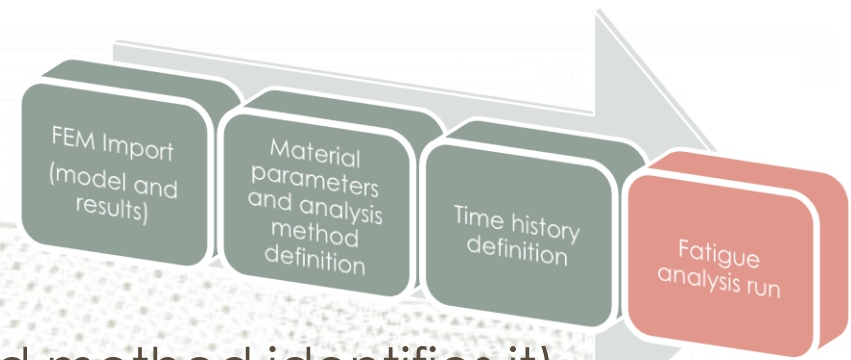- Cycles are calculated with Dirlik or Narrow Band methods

LIFING.Life
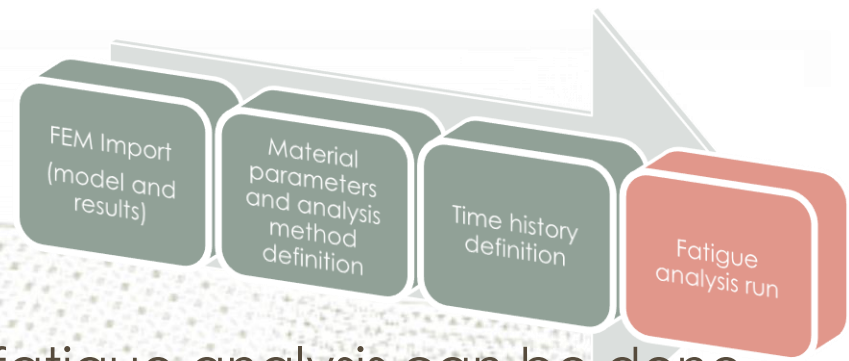
- Once the previous steps are done, the fatigue analysis can be done
  - On the entire FEM
  - On a defined set of Sub-Parts
  - On a defined Element
- Based on the defined analysis method, material properties, time history, etc.
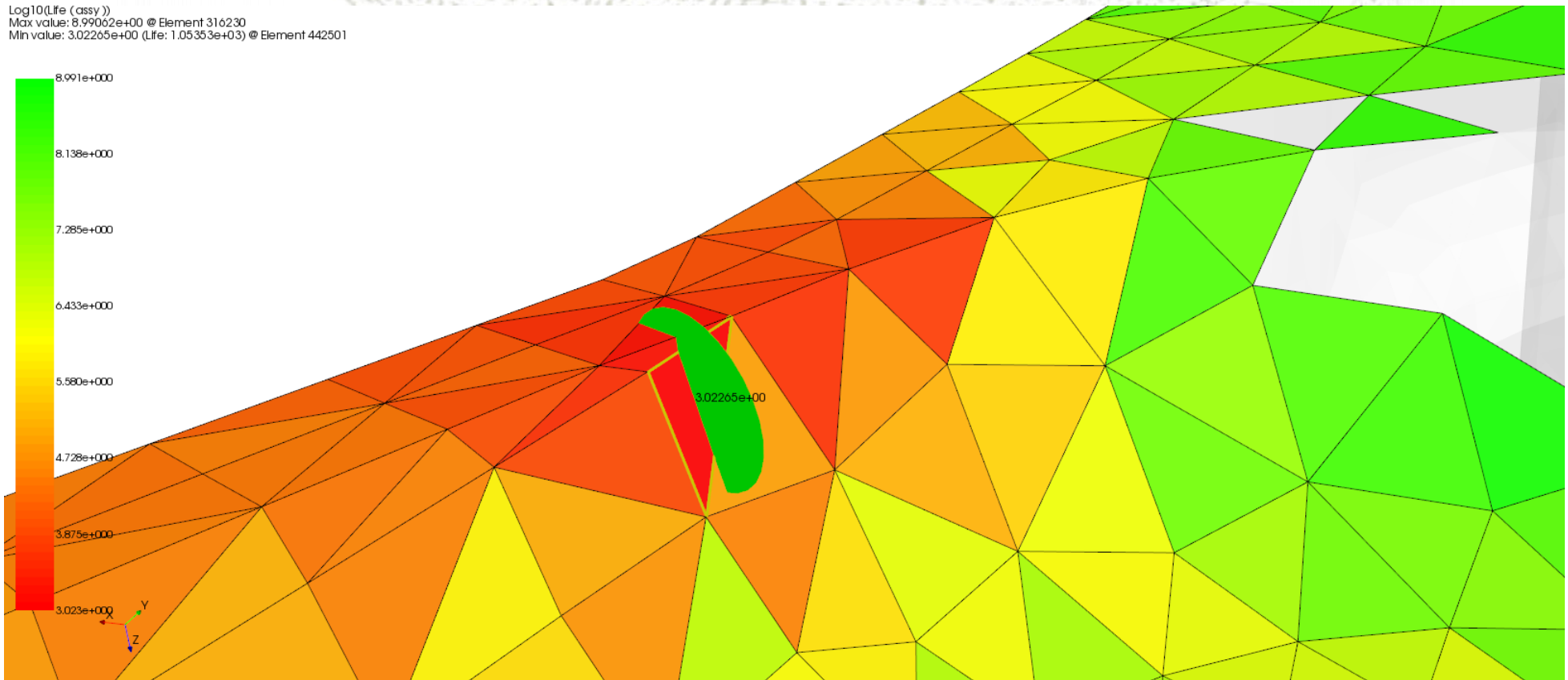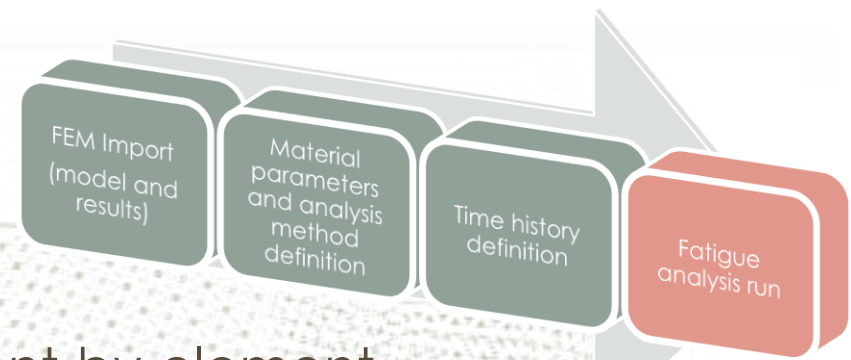- Fatigue Life is shown in the post-processor
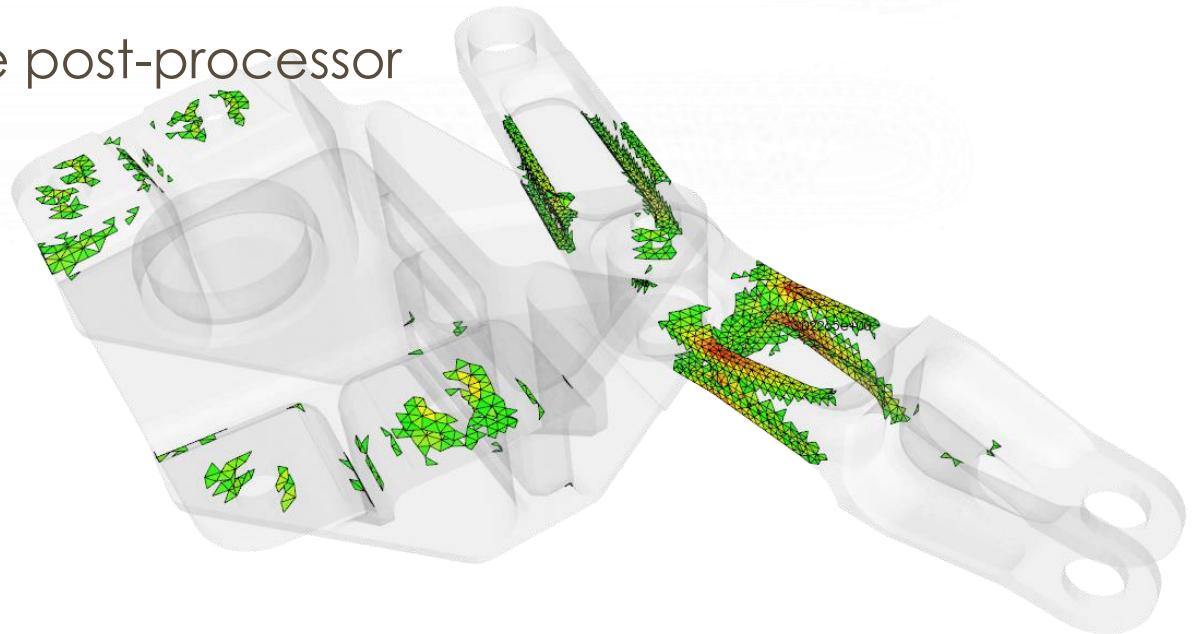
LIFING.Life

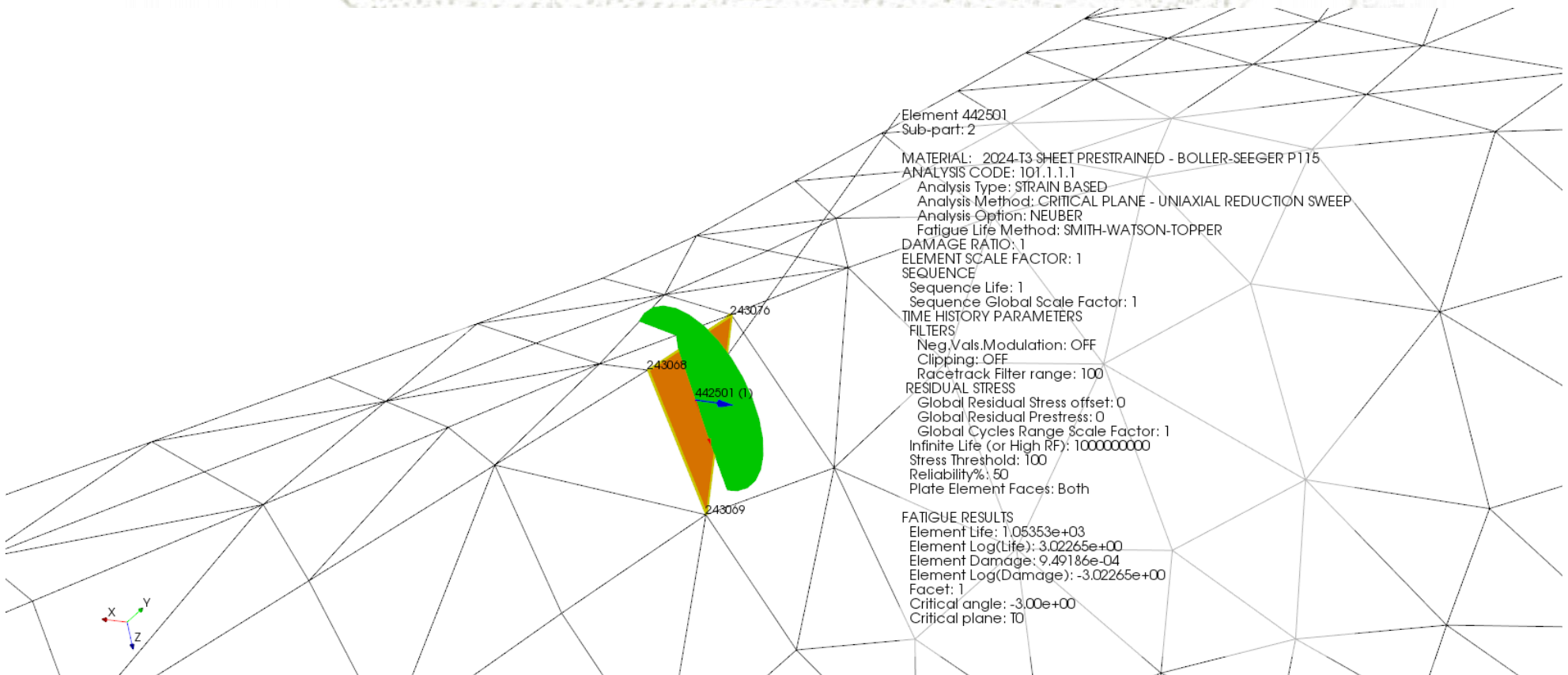- The critical plane is shown (if the defined method identifies it)

Log10(Life (assy))
Max value: 8.99062e+00 @ Element 316230
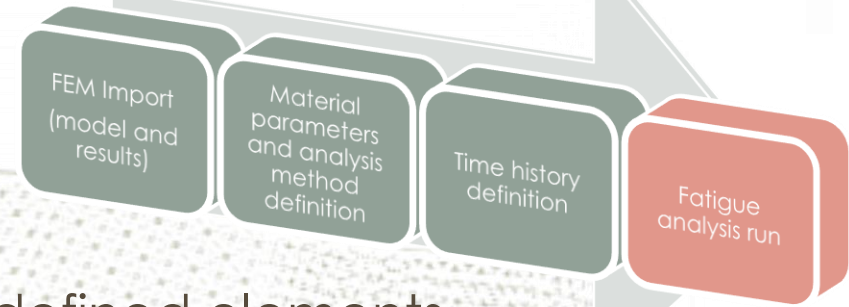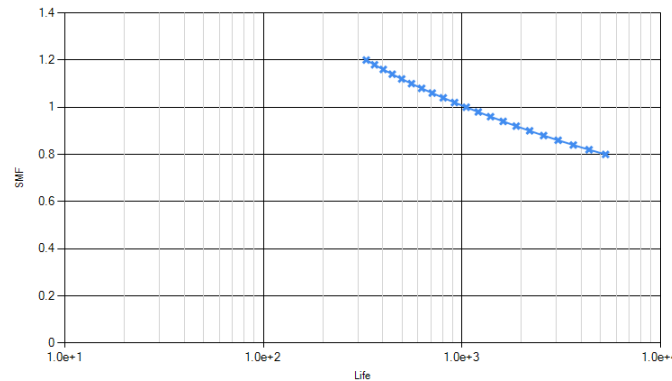Min value: 3.02265e+00 (Life: 1.05353e+03) @ Element 442501

LIFING.Life

- The solution can be interrogated element-by-element

Element 442501
Sub-part: 2

MATERIAL: 2024-T3 SHEET PRESTRAINED - BOLLER-SEEGER P115
ANALYSIS CODE: 101.1.1.1
  Analysis Type: STRAIN BASED
  Analysis Method: CRITICAL PLANE - UNIAXIAL REDUCTION SWEEP
  Analysis Option: NEUBER
  Fatigue Life Method: SMITH-WATSON-TOPPER
DAMAGE RATIO: 1
ELEMENT SCALE FACTOR: 1
SEQUENCE
  Sequence Life: 1
  Sequence Global Scale Factor: 1
TIME HISTORY PARAMETERS
  FILTERS
    Neg.Vals.Modulation: OFF
    Clipping: OFF
    Racetrack Filter range: 100
  RESIDUAL STRESS
    Global Residual Stress offset: 0
    Global Residual Prestress: 0
    Global Cycles Range Scale Factor: 1
  Infinite Life (or High RF): 1000000000
  Stress Threshold: 100
  Reliability%: 50
  Plate Element Faces: Both

FATIGUE RESULTS
  Element Life: 1.05353e+03
  Element Log(Life): 3.02265e+00
  Element Damage: 9.49186e-04
  Element Log(Damage): -3.02265e+00
  Facet: 1
  Critical angle: -3.00e+00
  Critical plane: T0

LIFING.Life

- The user can run 'sensitivity analysis' at defined elements

- LIFING provides in a ASCII file the breakdown of the damage calculation (useful for debugging the analysis or for further investigations)

```
Damage Calculation Breakdown
----------------------------

SMF: 1

Critical Plane Angle: -3.00000e+00

S_Max        S_Min        !d_S!        !d_sigma!    sigma_Max    eps_Max      !d_eps!      !d_eps_eq!   Ncycles      Occurrences   Part. damage   Cum. damage

303.42325    -390.59899   694.02224    693.89420    311.86615    3.958e-03    9.317e-03    4.416e-03    1.427e+04    1             7.006e-05      7.006e-05
312.56065    -395.43145   707.99210    707.78199    318.80406    4.11e-03     9.506e-03    4.51e-03     1.209e+04    1             8.271e-05      1.528e-04
249.91932    -346.42997   596.34929    596.34639    258.10593    3.243e-03    8.005e-03    3.724e-03    5.833e+04    1             1.714e-05      1.699e-04
364.88808    -425.44311   790.33119    787.32429    370.13271    4.826e-03    1.065e-02    5.143e-03    4.543e+03    1             2.201e-04      3.9e-04
448.27794    -433.26981   881.54774    856.88763    432.62125    6.235e-03    1.217e-02    5.945e-03    1.788e+03    1             5.592e-04      9.492e-04


Life =         1.0535e+03
Life*D =       1.0535e+03
N            =             1.0535e+03
N*LifeSeq    =             1.0535e+03
Life = N*LifeSeq*D =       1.0535e+03
```
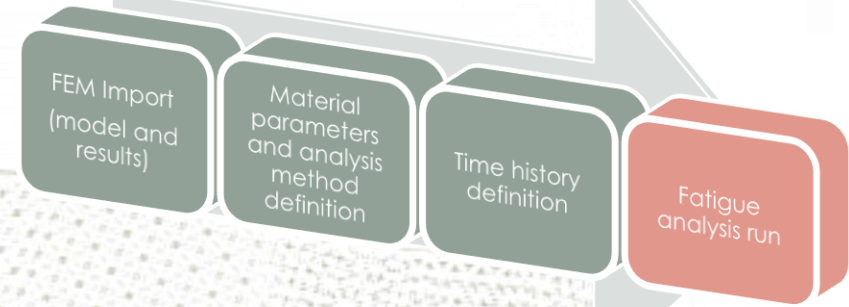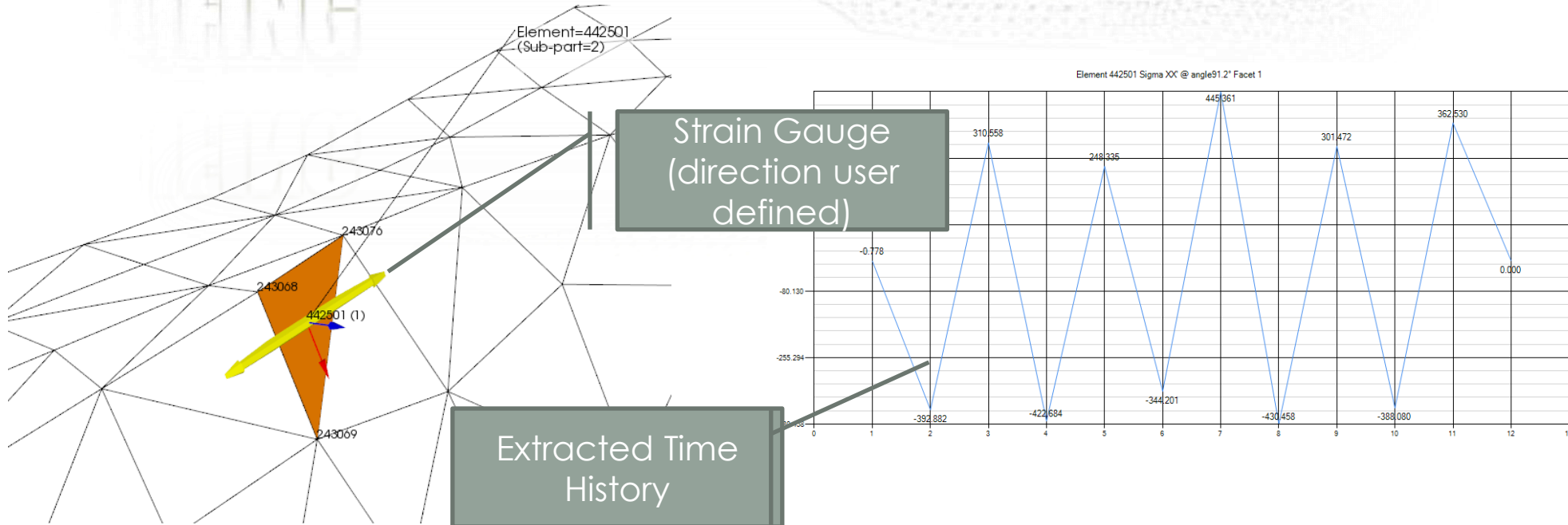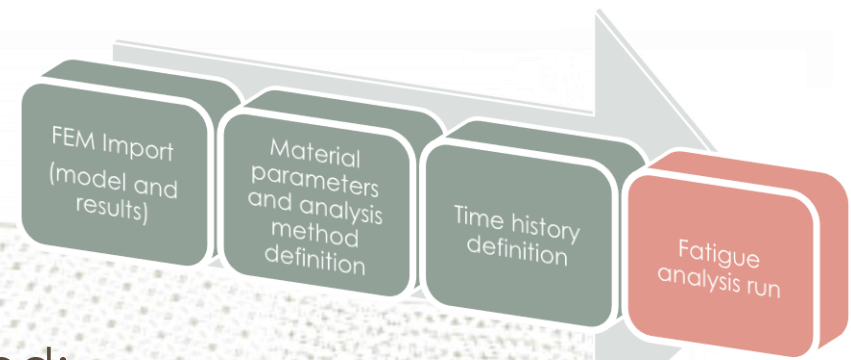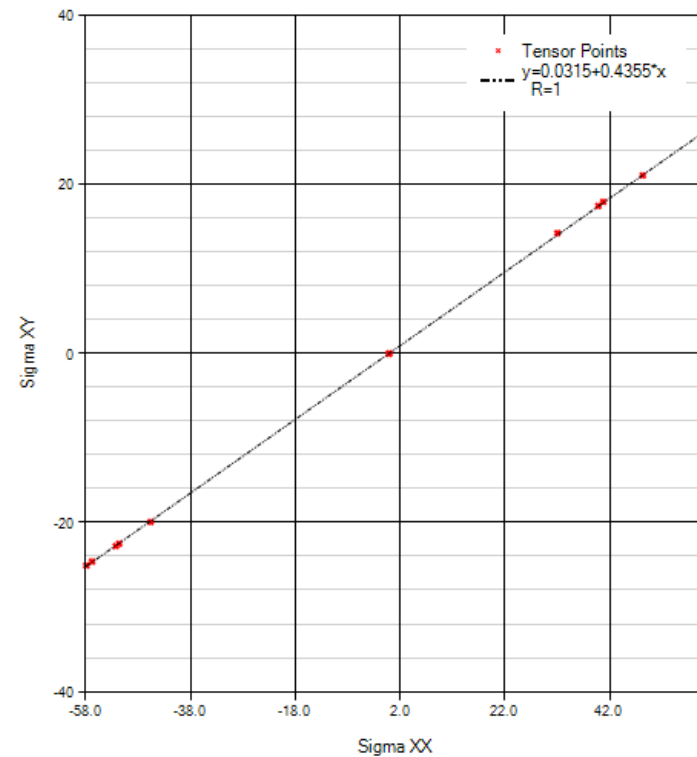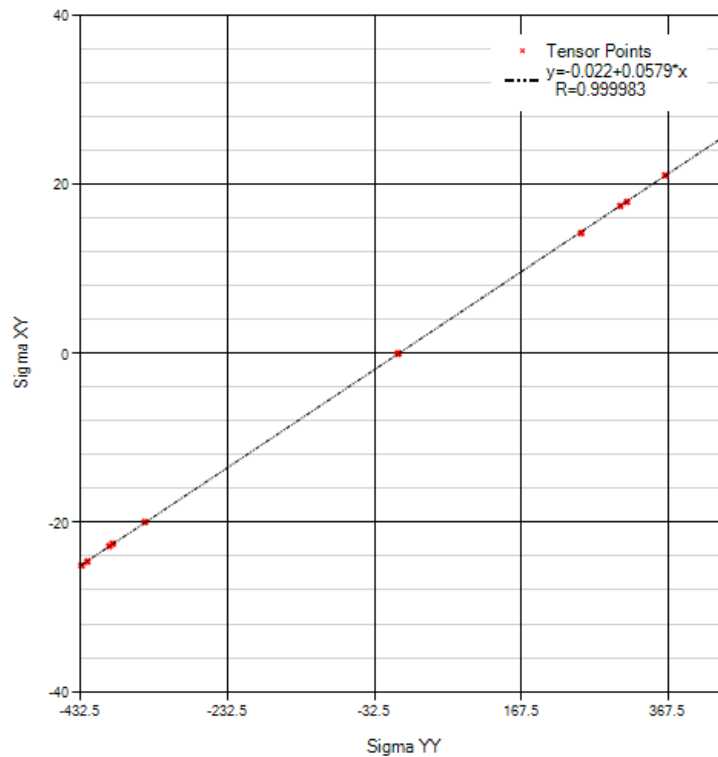
LIFING.Life

- VIRTUAL STRAIN GAUGE
  - The user can interrogate the model by putting a 'virtual strain gauge' at a given element, orient it at a given angle, and the related stress time history can be directly extracted

Element=442501 (Sub-part=2)

243076

243068

442501 (1)

243069

Strain Gauge (direction user defined)

Extracted Time History

Element 442501 Sigma XX @ angle91.2° Facet 1

445.361

362.530

310.558

301.472

248.335

-0.778

0.000

-80.130

-255.294

-344.201

-392.882

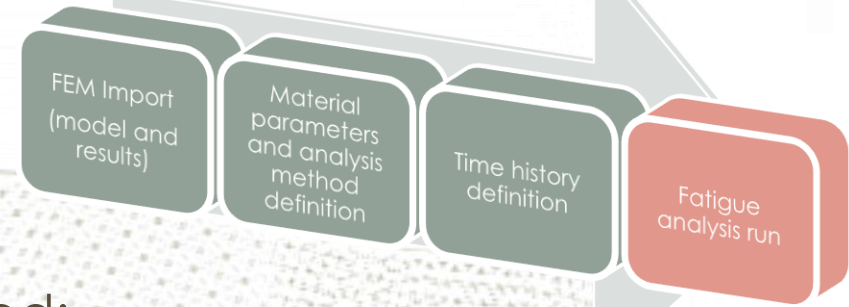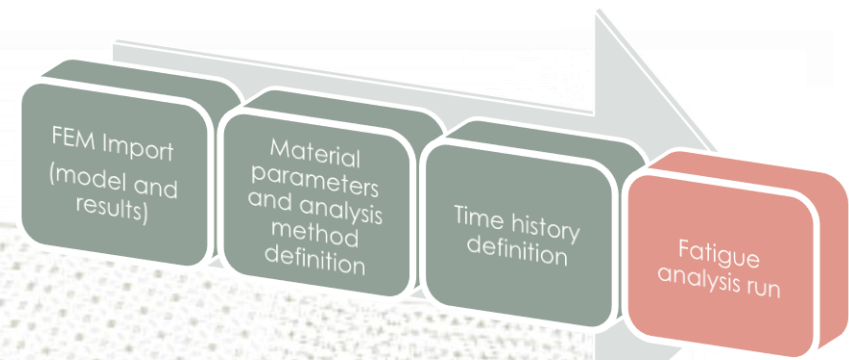-422.684

-430.458

-388.080

LIFING.Life

- Additionally the following can be derived:
  - Biaxiality plots and histograms
  - Mohr circles at given instants of the time history
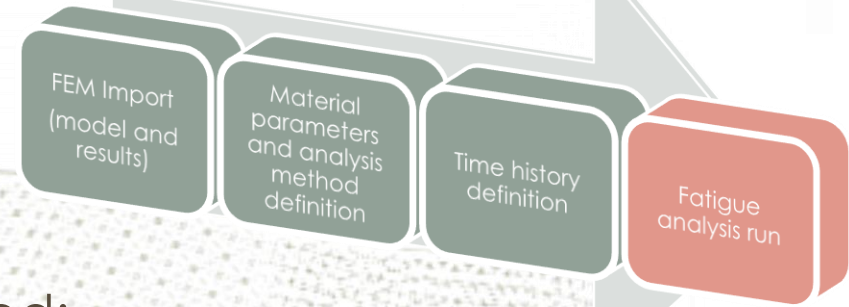
LIFING.Life

- Additionally the following can be derived:
  - Biaxiality plots and histograms
  - Mohr circles at given instants of the time history
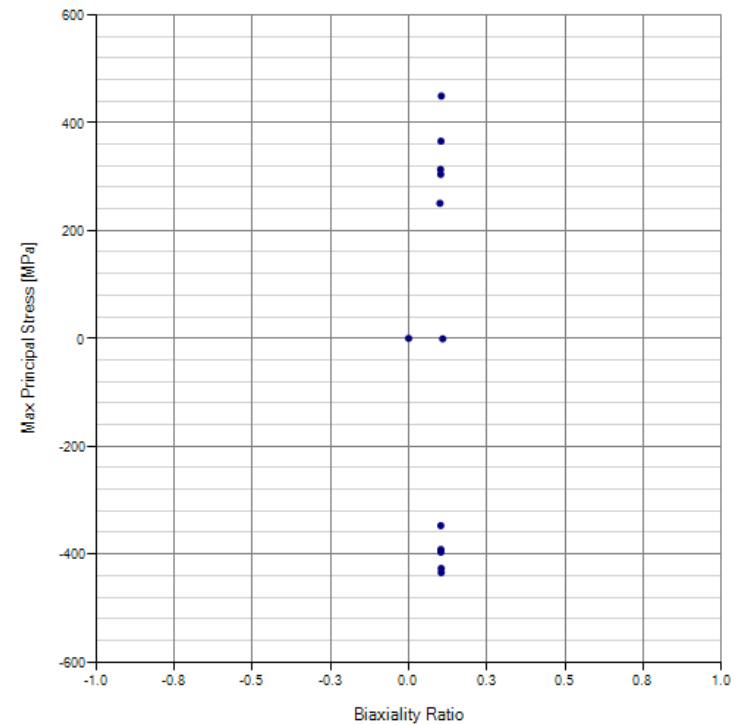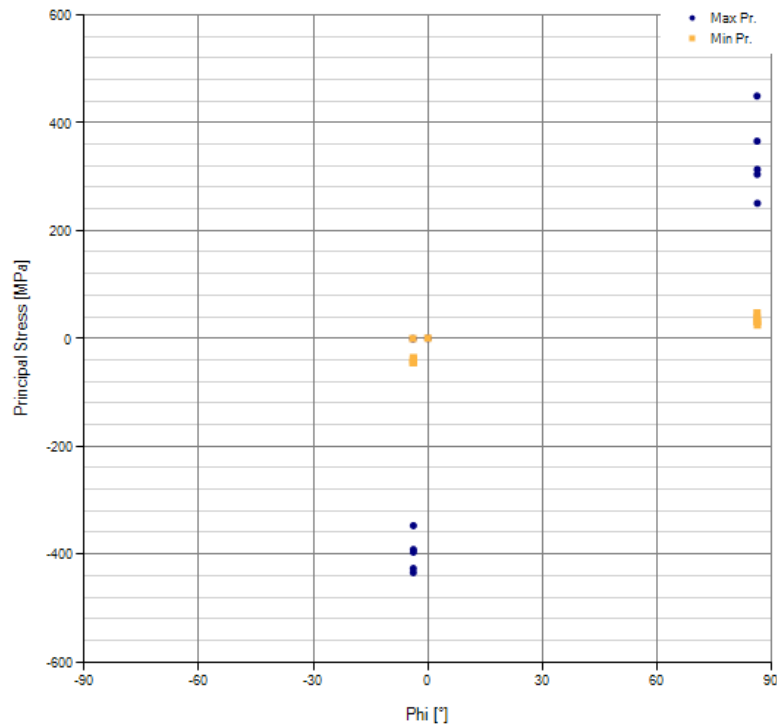
LIFING.Life

- Additionally the following can be derived:
  - Biaxiality plots and histograms
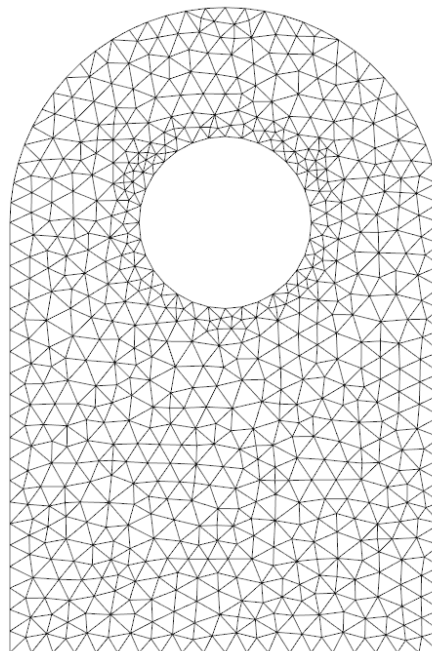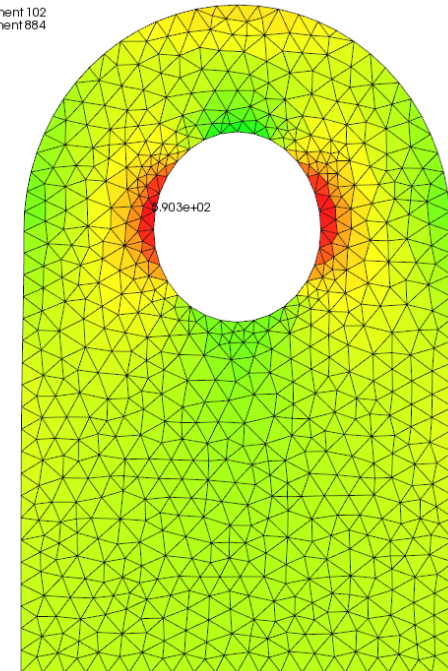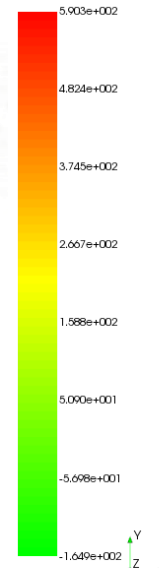 - Mohr circles at given instants of the time history
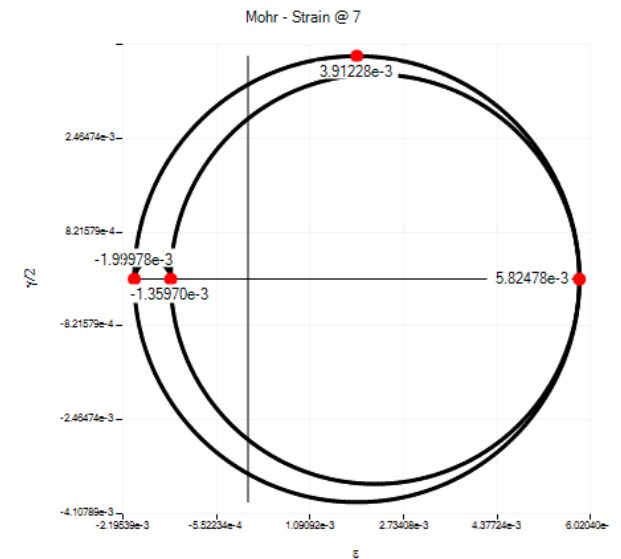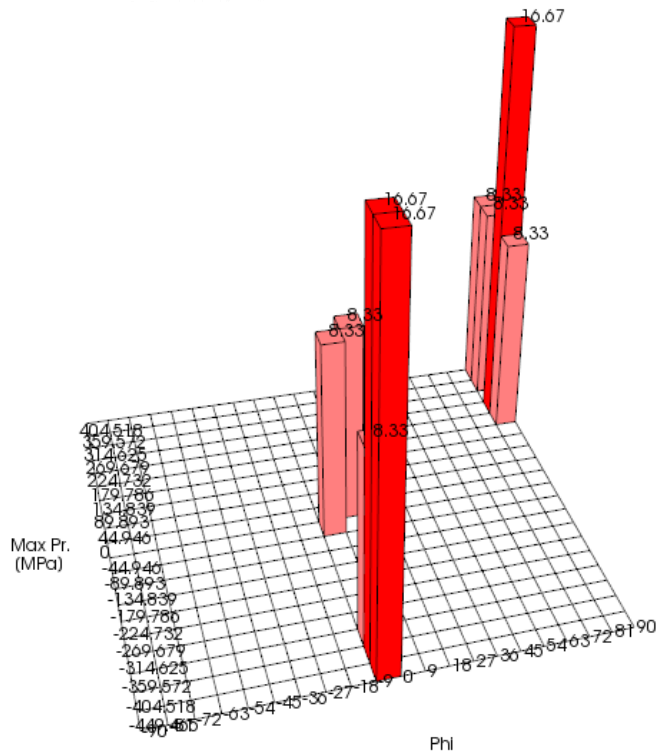
LIFING.Life

- QUICK 2D FEM
  - LIFING provides the capability to run fatigue analyses on 2D FEM generated with 'quick and dirty' approach with QUICK 2D FEM sub-module.
  - The user defines model edges (straight or curved) with number of elements, holes, boundary conditions and material properties, in a simple file format, ...
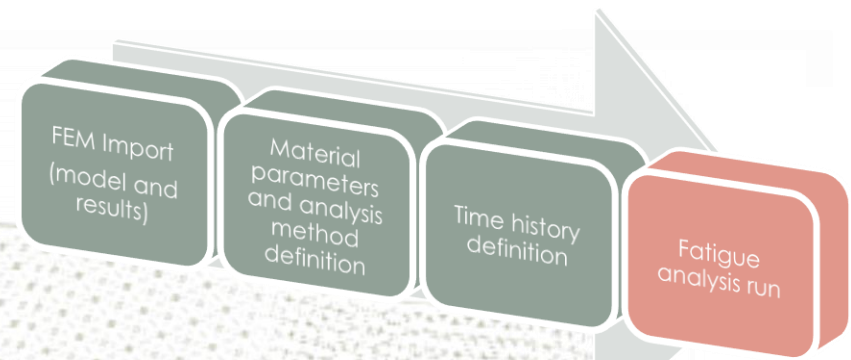 - ... the model is immediately solved and visualized within the LIFE module

```
thick 1
smode 0
material 70000 .33
point 1 0 0
point 2 10 0
point 3 10 10
point 4 0 10
point 5 5 15
point 6 5 0
edge 1 0 1 6 10
edge 2 0 6 2 10
edge 3 0 2 3 20
edge 4 1 3 4 5 34
edge 5 0 4 1 20
hole 1 5 10 2 32
dedge 1 1 2 0
dedge 1 2 2 0
dpoint 1 1 12 0
fhole 1 1 1000 90 180
dump spc gpf
```


LC 1 Elm2D. Smax+
Max value: 5.903e+02 @ Element 102
Min value: -1.649e+02 @ Element 884

5.903e+002
4.824e+002
3.745e+002
2.667e+002
1.588e+002
5.090e+001
-5.698e+001
-1.649e+002

LIFING.Life

FEM Import (model and results) → Material parameters and analysis method definition → Time history definition → Fatigue analysis run
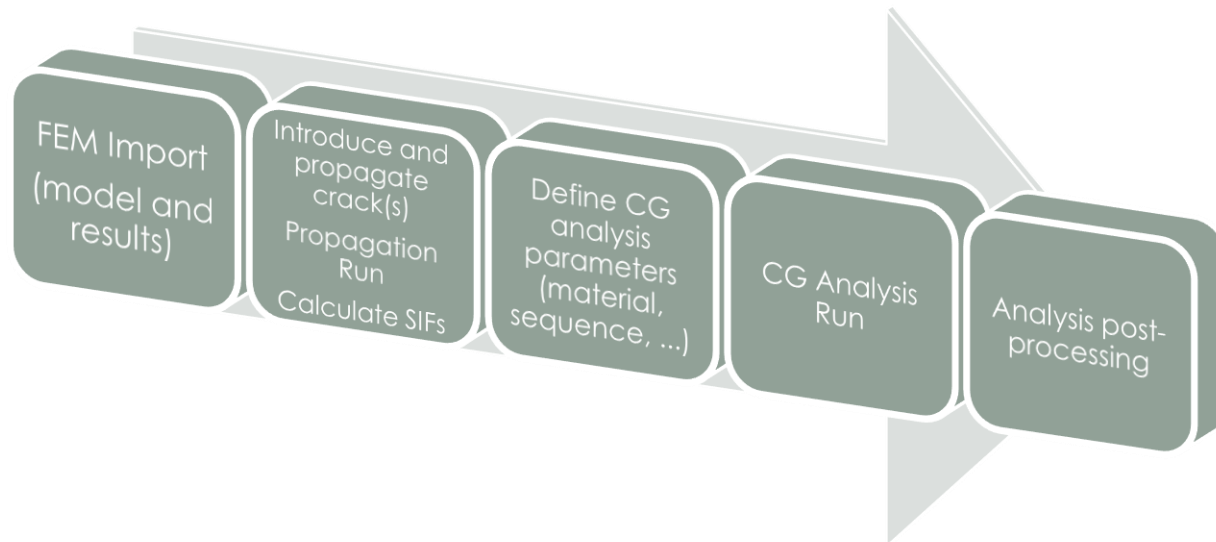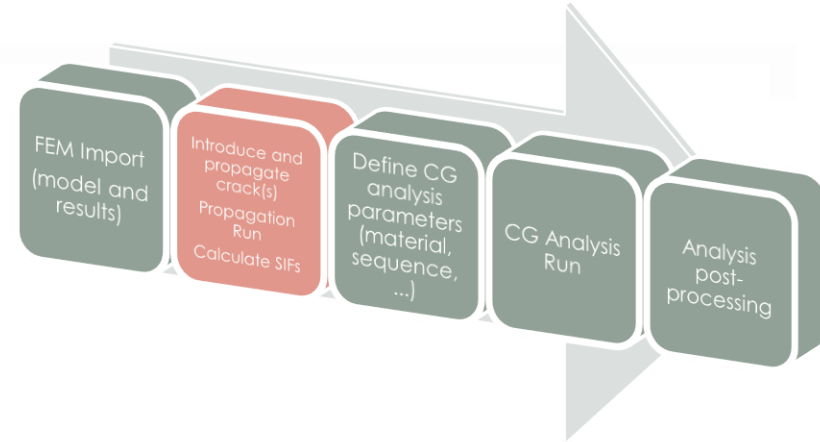
- **QUICK 2D FEM**
  - Loaded holes can be properly simulated by applying load or constraint at the bore with a user defined angle (i.e. The contact bore-fastener is captured).

  - QUICK 2D FEM feature is at the basis of the JOINT 2D analysis capability.
    - The user extracts from the entire FEM the stress time histories close to a fastener
    - The time history is internally converted in a bypass stress time history and a bearing stress time history (two beraing channels depending on the direction)
    - Then Fatigue or Crack Growth can be analysed at the joint

LIFING.Growth

- LIFING has the capability to calculate crack growth in a shell FEM model
- Crack Growth in 2D (planar) geometries or Section Cuts of 3D meshes is solved.
- The workflow

FEM Import (model and results) → Introduce and propagate crack(s) Propagation Run Calculate SIFs → Define CG analysis parameters (material, sequence, …) → CG Analysis Run → Analysis post-processing
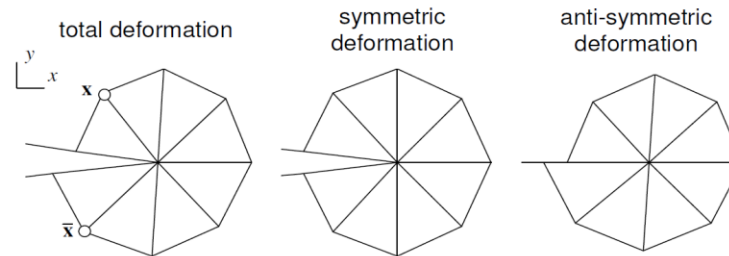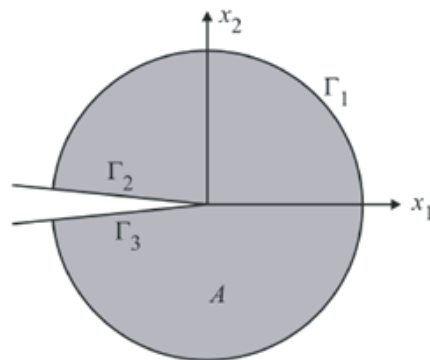
LIFING.Growth

- The analysis is carried out in two steps:

- **Crack(s) propagation simulation in 2D FEM**
  - One or more cracks are introduced
  - The FEM is remeshed, basd on the crack propagation
  - At the crack(s) tip, Stress Intensity Factors are calculated with J-Integral method

$$J = \int_\Gamma \left(W n_x - T_i \frac{\partial u_i}{\partial x} \right) ds \;\rightarrow\; \bar{J} = \int_A \left[\sigma_{ij} \frac{\partial u_i}{\partial x_1} - W \delta_{1j} \right] \frac{\partial q_1}{\partial x_j} \, dA \;\rightarrow\; J = G = \frac{1-v^2}{E} (K_I^2 + K_{II}^2)$$

total deformation   symmetric deformation   anti-symmetric deformation

$$G_I = J_I = J(u_{sym}, \sigma_{sym})$$

$$G_{II} = J_{II} = J(u_{anti-sym}, \sigma_{anti-sym})$$
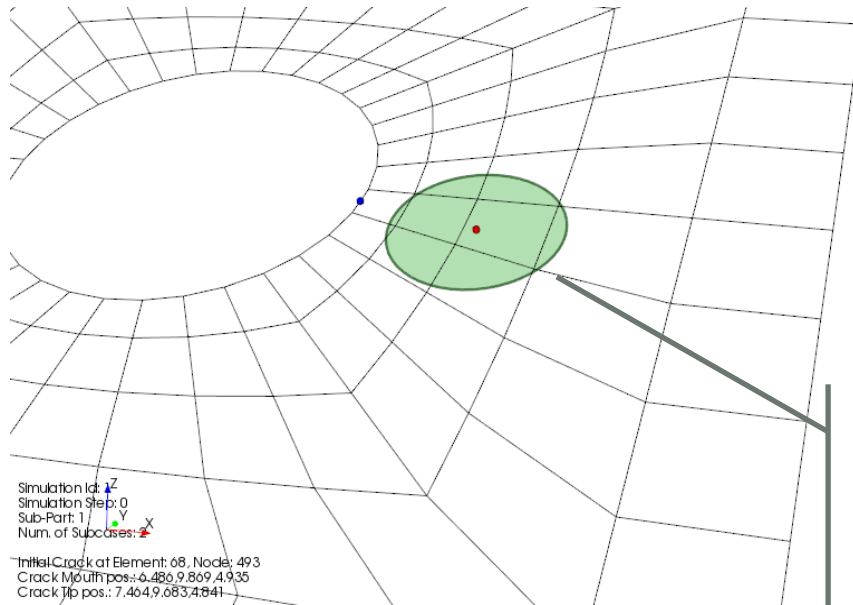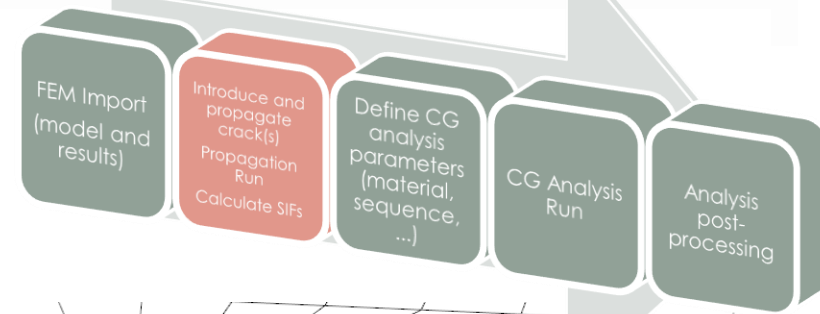
$$u = u_{sym} + u_{anti-sym} \qquad u_{sym} = \tfrac{1}{2} \left\{ \begin{matrix} u + \bar{u} \\ v - \bar{v} \end{matrix} \right\} \qquad u_{anti-sym} = \tfrac{1}{2} \left\{ \begin{matrix} u - \bar{u} \\ v + \bar{v} \end{matrix} \right\}$$

LIFING.Growth

FEM Import (model and results)

Introduce and propagate crack(s)
Propagation Run
Calculate SIFs

Define CG analysis parameters (material, sequence, ...)

CG Analysis Run

Analysis post-processing

Simulation Id: 1
Simulation Step: 0
Sub-Part: 1
Num. of Subcases: 2

Initial Crack at Element: 68, Node: 493
Crack Mouth pos.: 6.486,9.869,4.935
Crack Tip pos.: 7.464,9.683,4.841

Initial Crack Definition
The analyst decides the orientation, length and level of refinement

- After being introduced, the crack(s) propagation is performed automatically
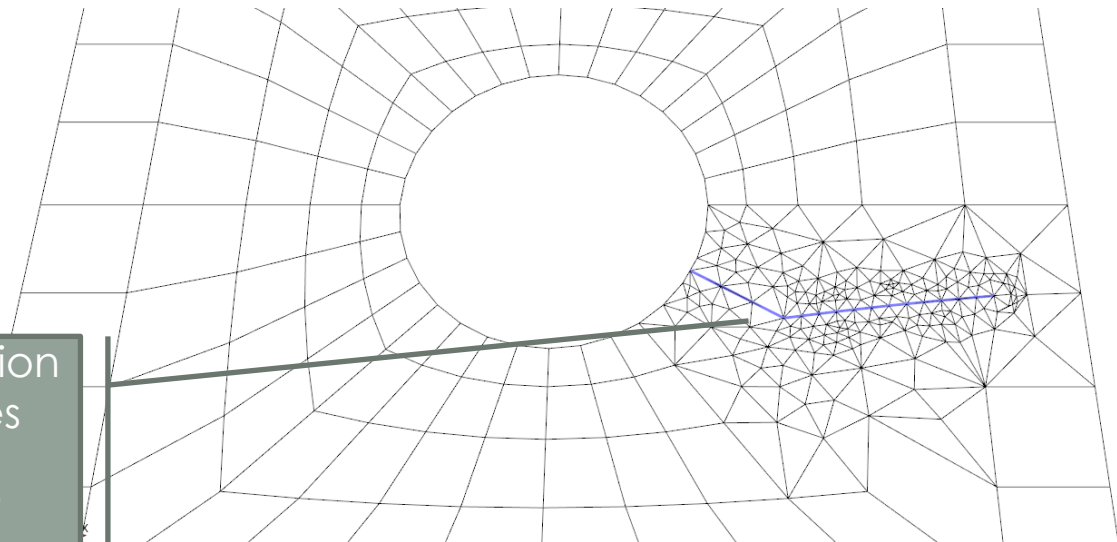
LIFING.Growth
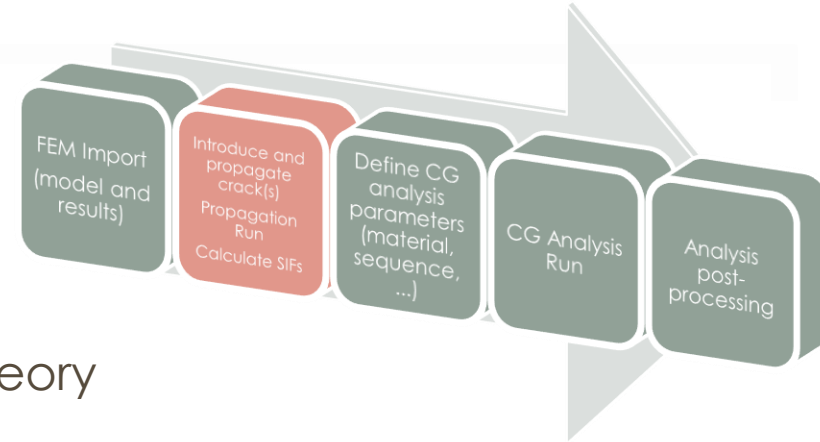
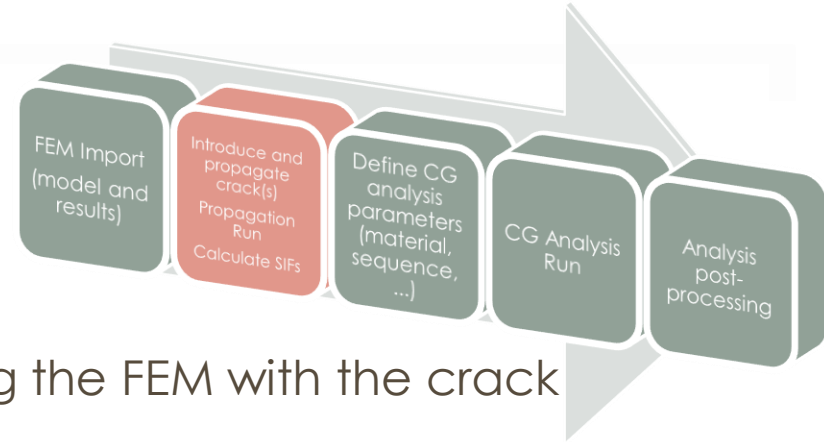- Crack Kinking calculated with the $\sigma_{\theta\theta,max}$ theory
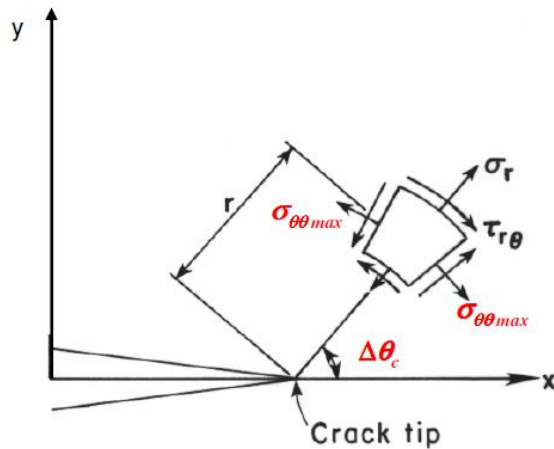
$$\frac{K_{II}}{K_I} = \frac{-\sin\Delta\theta_c}{(3\cos\Delta\theta_c - 1)} \quad \rightarrow \quad \Delta\theta_c = 2\tan^{-1}\left(\frac{1 - \sqrt{1 + 8(K_{II}/K_I)^2}}{4(K_{II}/K_I)}\right)$$

Initial Crack Definition
The analyst decides the orientation, length and level of refinement

LIFING.Growth

- An internal mesher is in charge of rebuilding the FEM with the crack discontinuities
- An internal solver is in charge of calculating stress and deformations at each crack propagation step
- Not only cracks can be introduced, but also holes to stop drill cracks

Initial Crack

Stop Drill Hole

Crack initiated and propagated at the Stop Drill
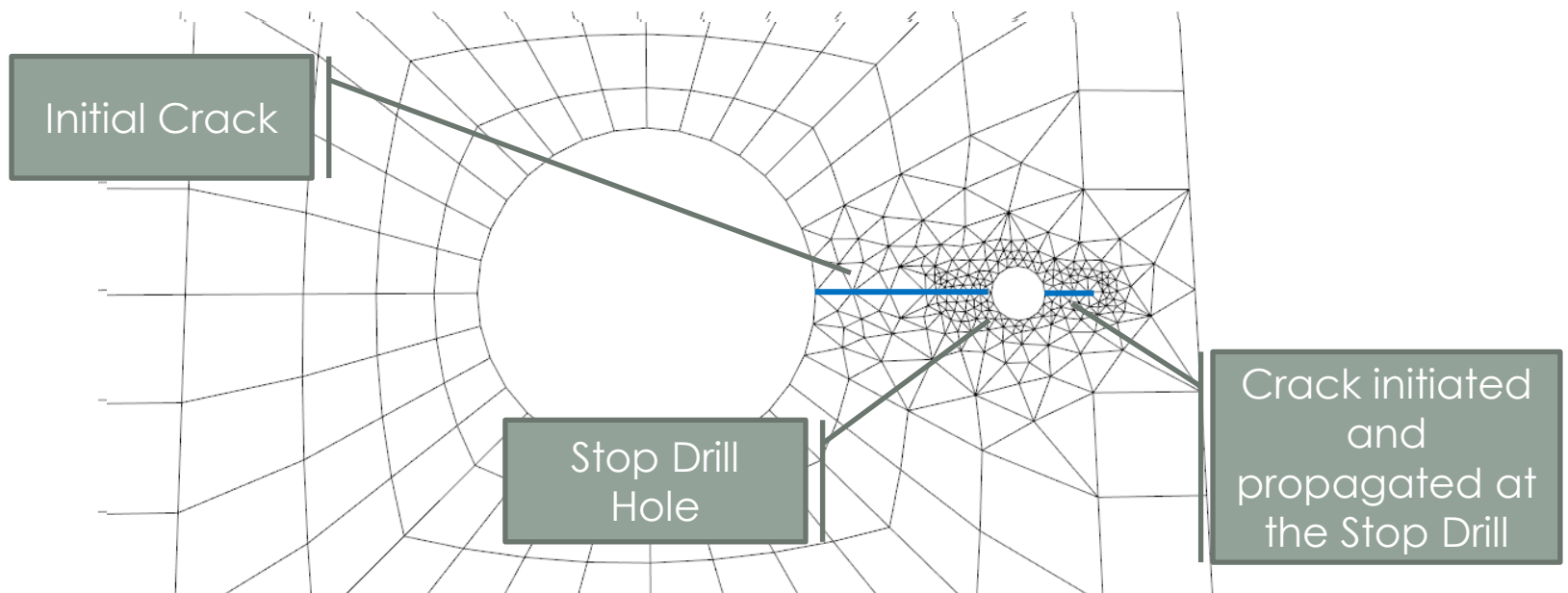
LIFING.Growth
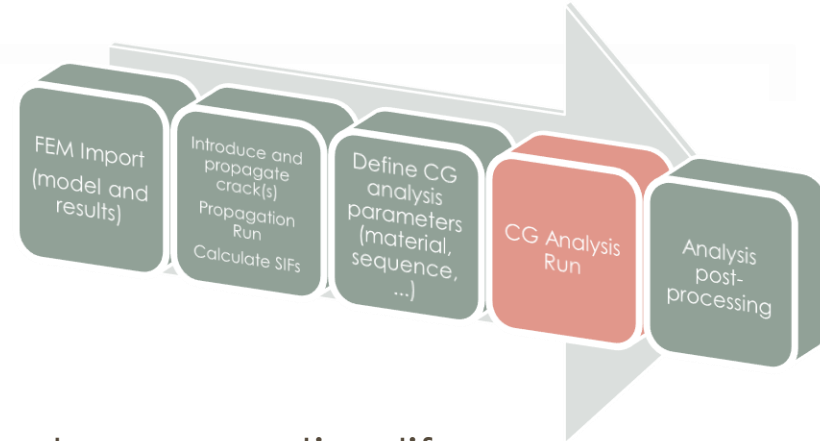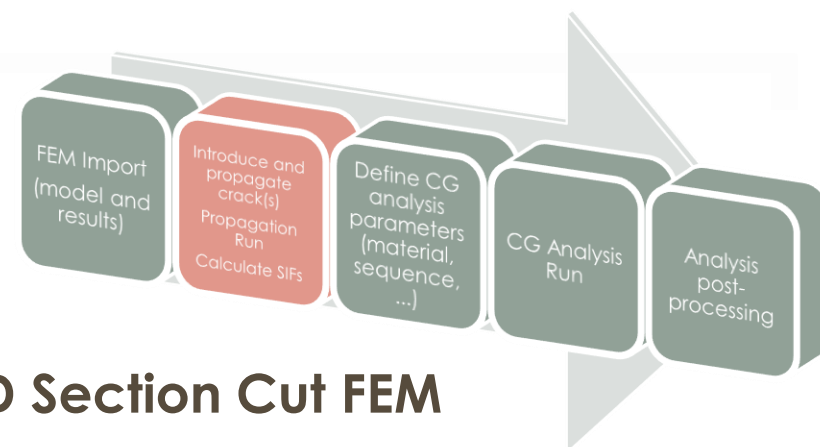

- **Crack Growth calculation**
  - NASGRO model is used to calculate crack propagation life

$$\frac{da}{dN} = C\left[\left(\frac{1-f}{1-R}\right)\Delta K\right]^n \frac{\left(1 - \frac{\Delta K_{th}}{\Delta K}\right)^p}{\left(1 - \frac{K_{max}}{K_{crit}}\right)^q}$$

  - Multiple cracks growth calculated (relative growth rate accounted with Paris law)
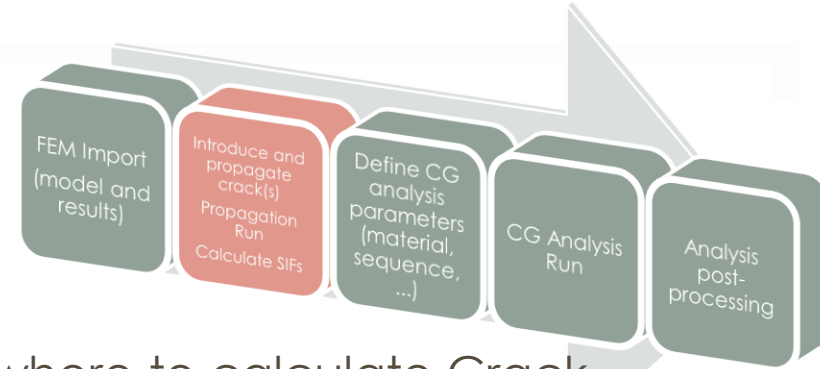 - Generalized Willemborg Crack Closure model implemented

LIFING.Growth

- **Crack(s) propagation simulation in 3D Section Cut FEM**
  - Nodal stress results must be preliminarily imported, in order to have stress data «inside» the FEM
  - A Section Cut is defined by the user (or automatically at the location showing minimum Crack Initiation Life)
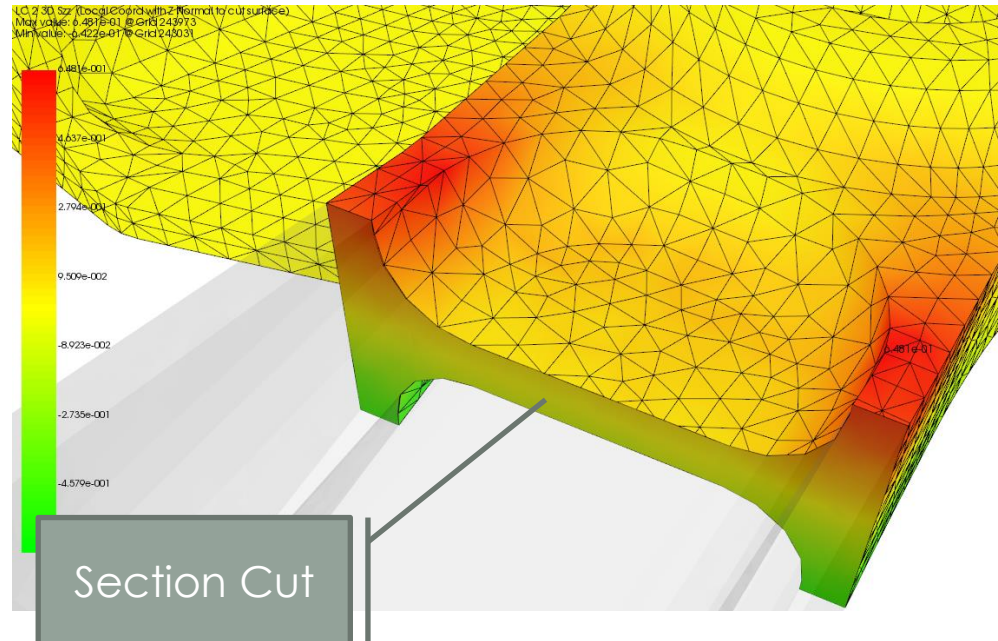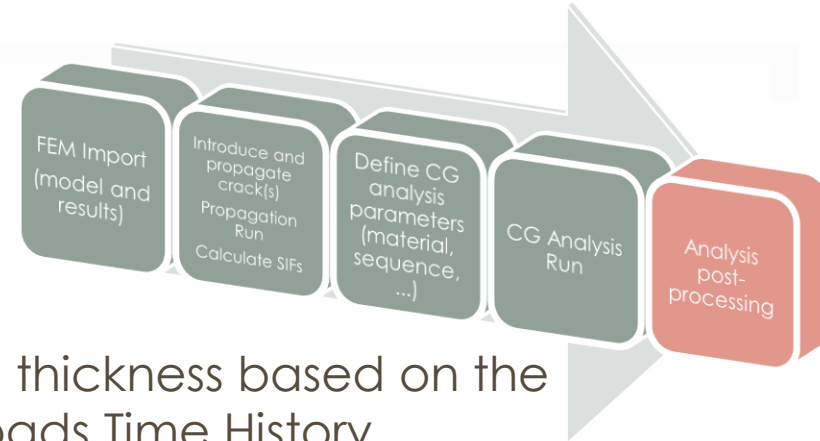
# LIFING.Growth

- The user defines a Rectangula portion where to calculate Crack Growth (engineering approach using Newman-Raju corrected with Glinka's Weight Functions)
- An initial crack (corner or pass-through) is defined

Initial Crack

Rectangle
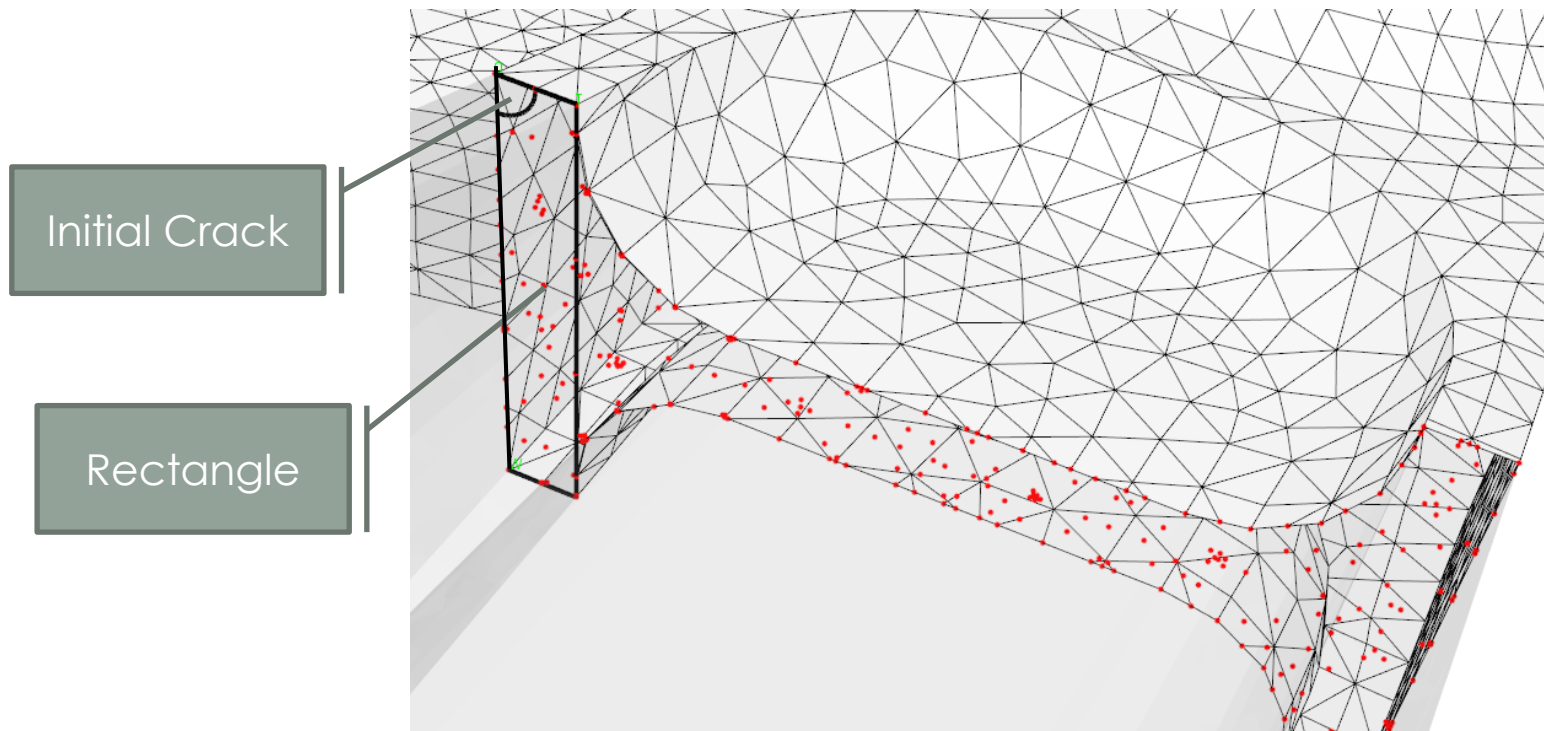
LIFING.Growth

- The crack is automatically grown in the thickness based on the material NASGRO data and applied Loads Time History
- Implemented approach provides excellent adherence to AFGROW solution

Example Problem (Crack C Length vs. Cycles)
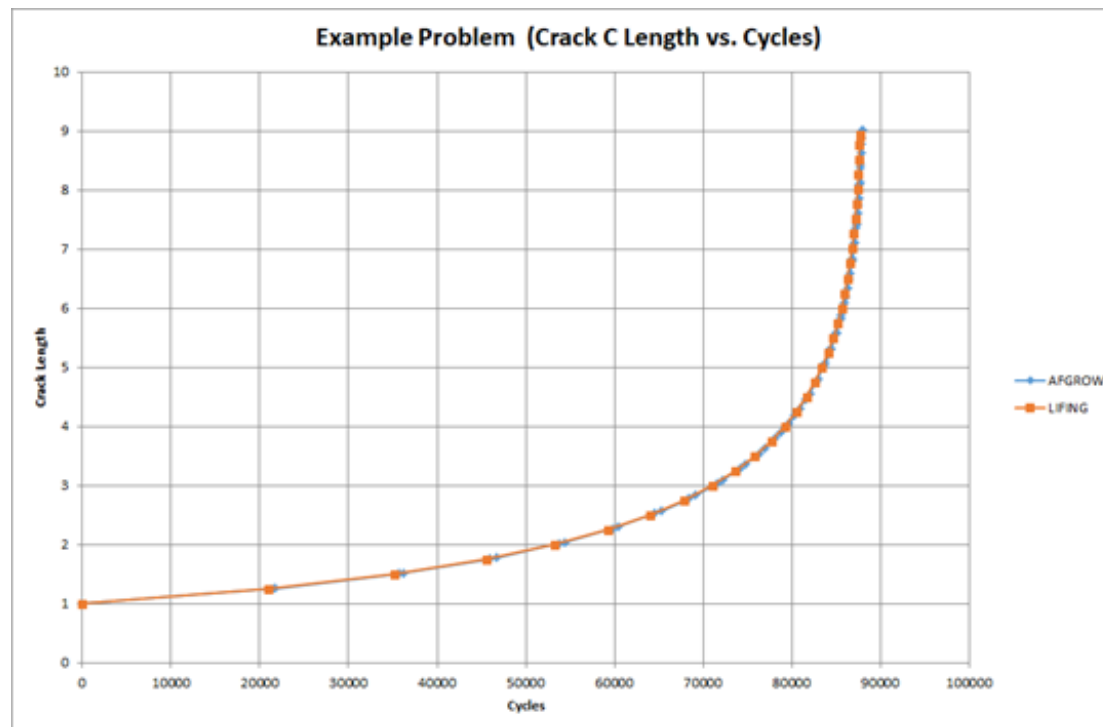
TECHNICAL INFORMATION

- LIFING is developed by Fatigue Experts mainly operating in the Aerospace field, not just software developers, therefore implemented functionalities are effective for real life fatigue and damage tolerance analyses
- Parallel Processing is implemented (high performance)
- Graphics with VTK technology
- LIFING database consists of a generated SQLite database, easily readable and accessible with free packages (no need to keep the license to access, once expired, Fatigue calculation results)
- Results files are produced, such to allow the users to fully 'debug' their analysis
- Under constant improvement and upgrades

- For further info mail to: support@lifing-fdt.com

REFERENCES

Tata Consultancy Services

Maserati

UniParma

Pilatus Aircraft

Aerea

Flowserve

Aquila Engineering

CST Firenze