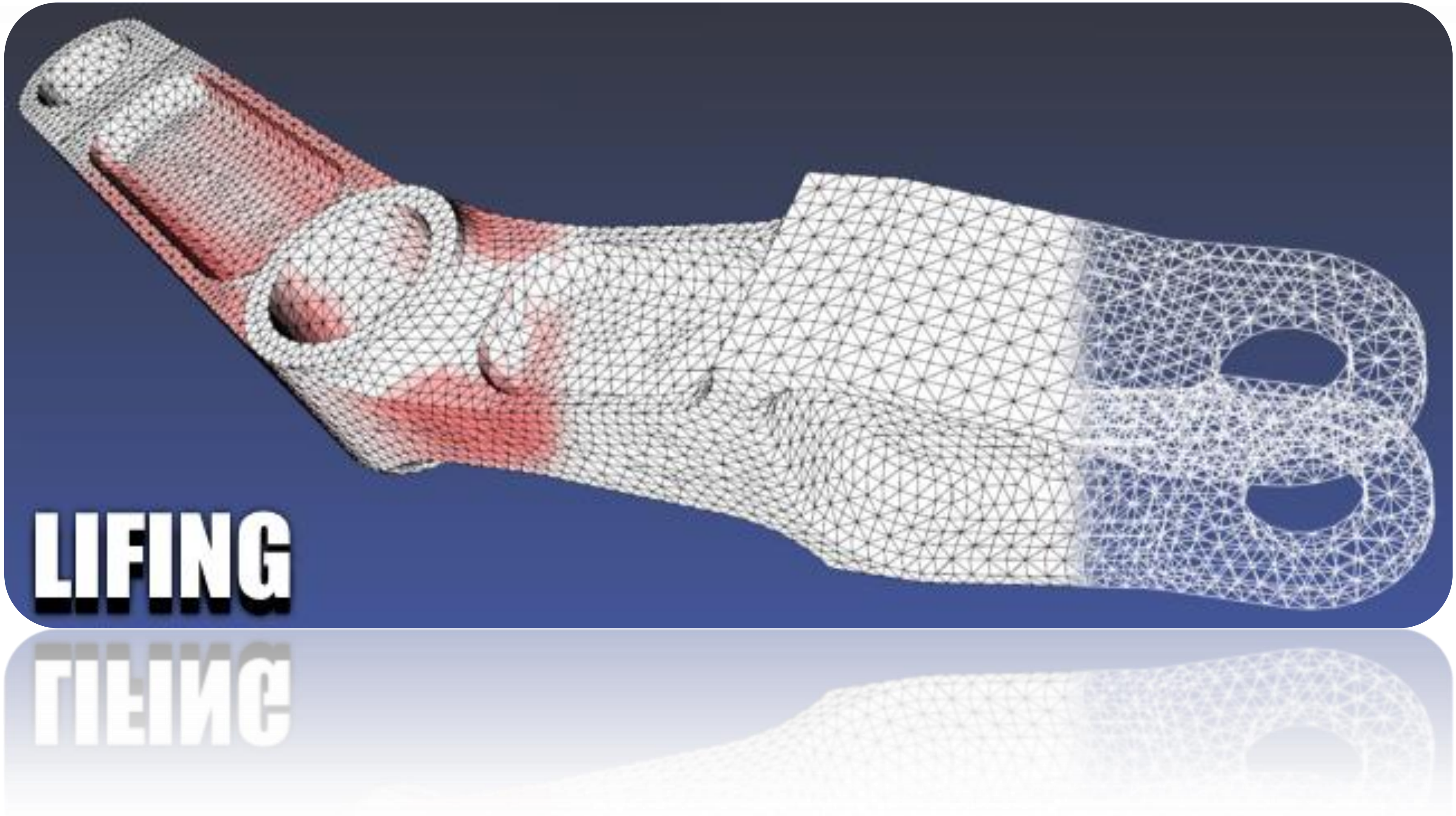


Current Release: 3.4

OUTLOOK

- What is LIFING
- LIFING.Life
  - FEM INTERFACES
  - ANALYSIS PROCESS
  - IMPLEMENTED ANALYSIS METHODS (TIME DOMAIN)
  - FATIGUE BASED ON PSD (FREQUENCY DOMAIN)
  - VIRTUAL STRAIN GAUGE
- LIFING.Growth
  - ANALYSIS PROCESS
  - IMPLEMENTED ANALYSIS METHODS
- QUICK2DFEM
- 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 and 3D FEM can be solved to extract Stress Intensity Factors (internal FEM solver and mesher; J-Integral calculated at crack tips in 2D FEM where «quarter point elements» are used; M-Integral calculated at crack fronts in 3D FEM where «quarter point elements» are used.
- 3D Solid Mesh Sections can be solved (Stress Intensity Factors are calculated based on Newman-Raju solution corrected with Glinka's Weight Functions using stress distributions automatically extracted at section cuts, or imported from file)
- NASGRO model or da/dN in tabular format used for integrating Crack Growth Life curve

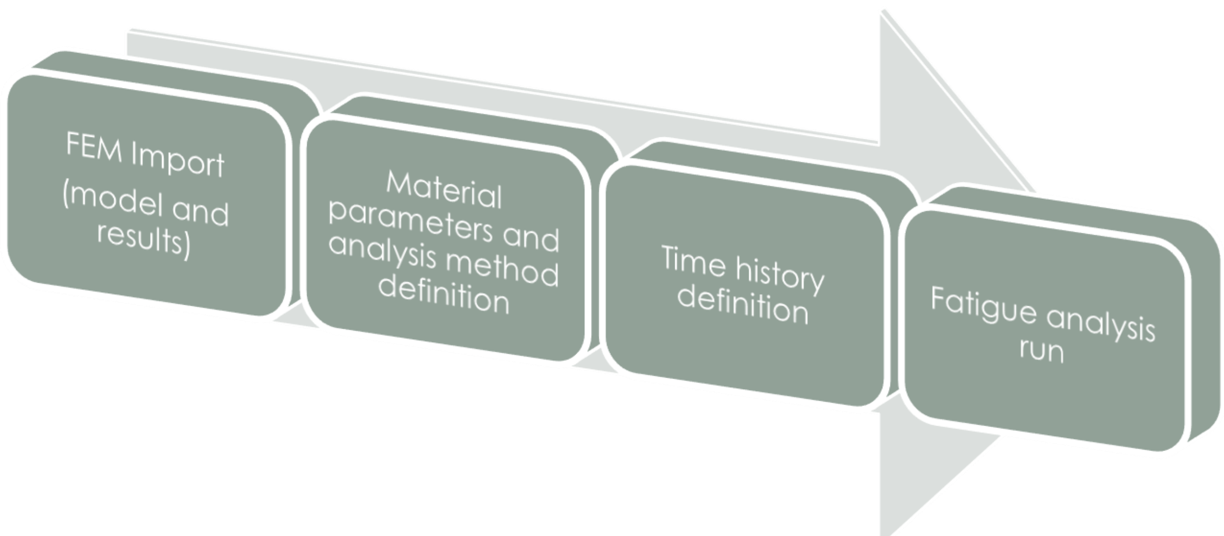
WHAT IS LIFING

QUICK2DFEM

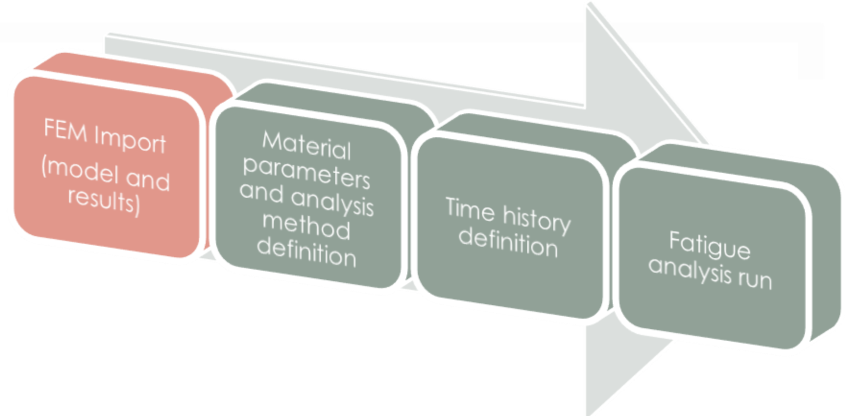
- Creation of 2D models on the basis of simple scripts
- 2D FEM Solver
- Creation of 3D models from extrusion of 2D models
- 3D FEM Solver
- Capability to introduce crack fronts and calculate Stress Intensity Factors
- Cost effective to design repairs, assess stress concentration factors, estimate quickly design deviations, ...

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:



LIFING.Life



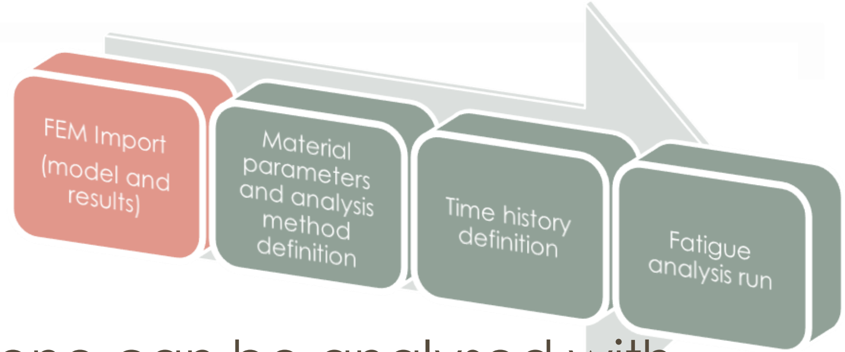
- LIFING includes FEM interfaces for:

  - **NASTRAN, OPTISTRUT**
    - Import of model (ASCII file NASTRAN format) and results files OP2 (bynary) and PCH (ASCII) generated by NASTRAN/OPTISTRUT

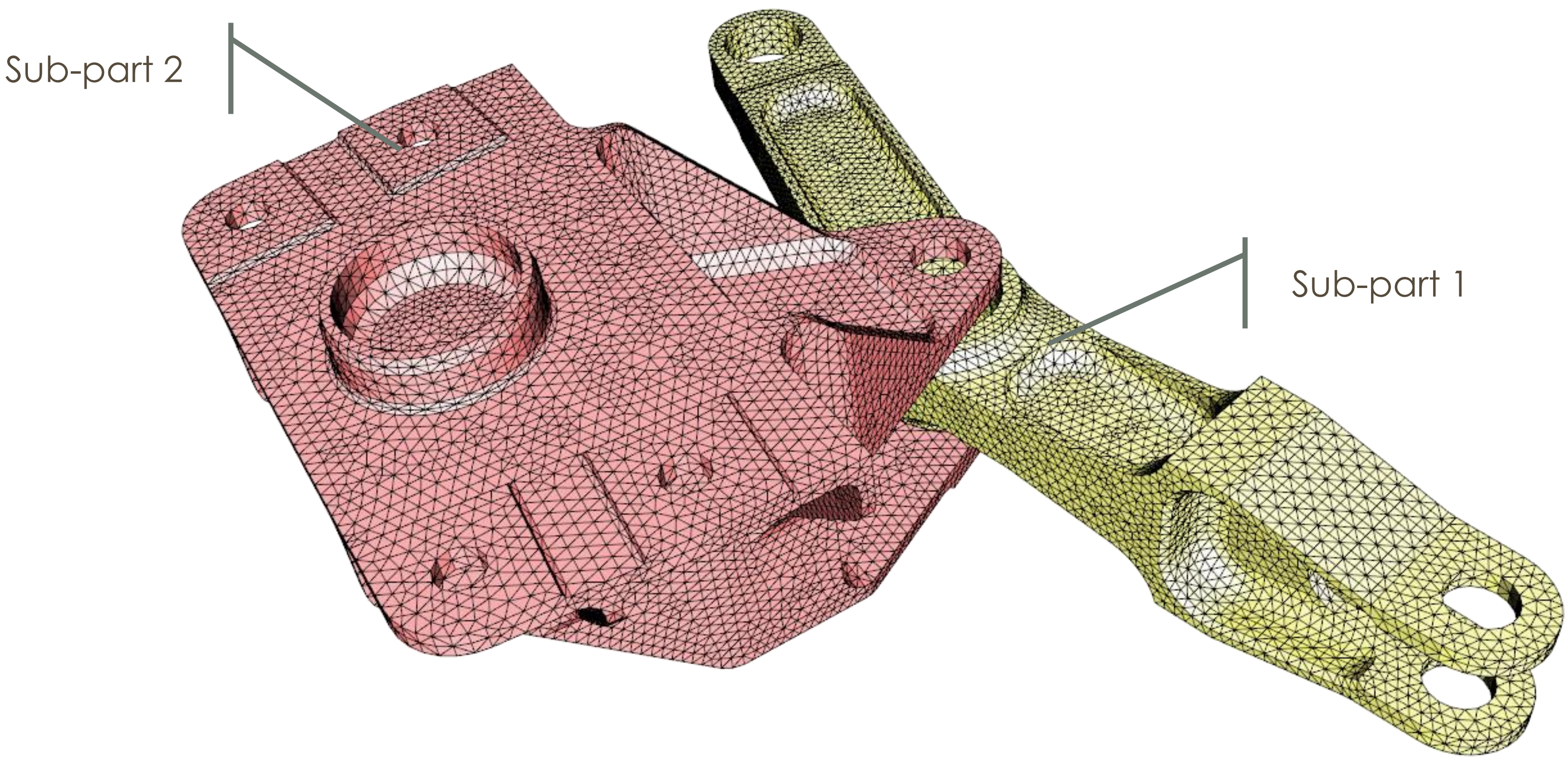
  - **ANSYS**
    - Import of model (ASCII file ANSYS format) and results files (ASCII) generated by ANSYS (the CDB bynary file interface is not available)

  - **ABAQUS, CALCULIX**
    - Import of model (ASCII file ABAQUS/CALCULIX format) and results files FIL (ASCII) generated by ABAQUS and FRD (ASCII) generated by CALCULIX

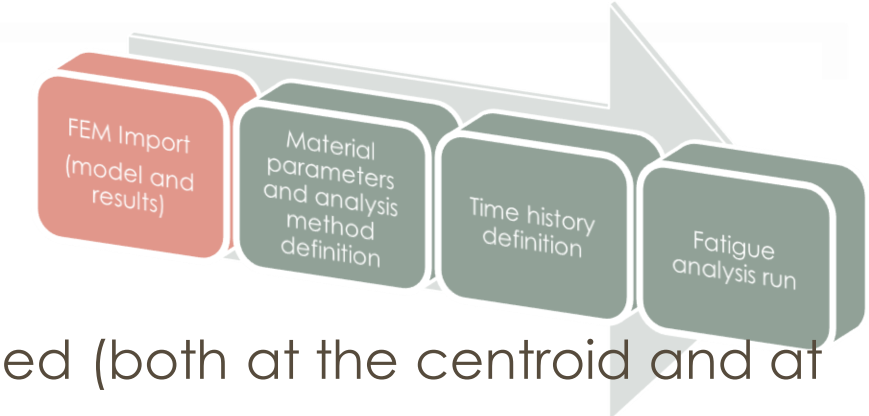
LIFING.Life



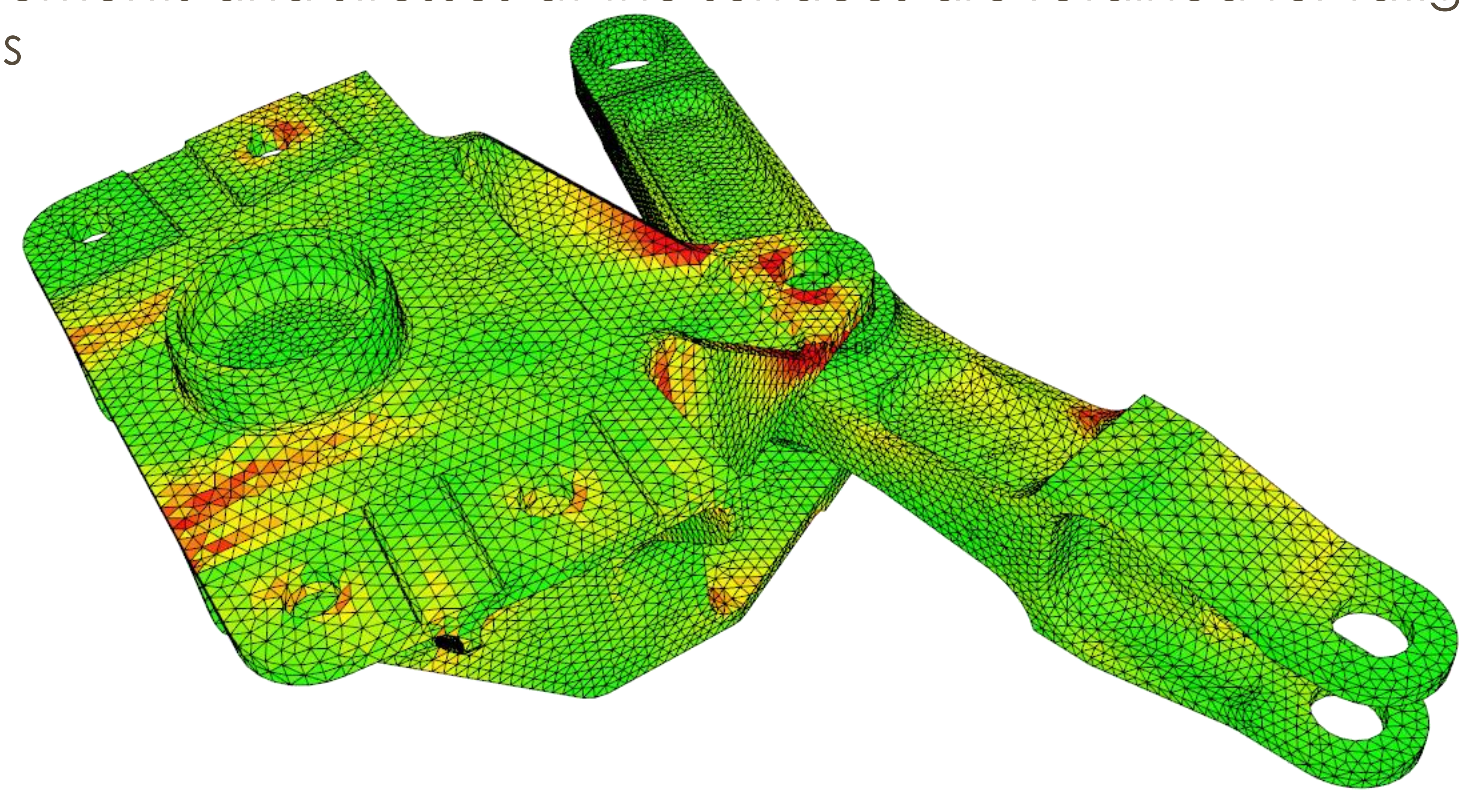
- The FEM can be split in sub-parts (each one can be analysed with different methods and/or parameters, e.g. for accounting local surface treatments)



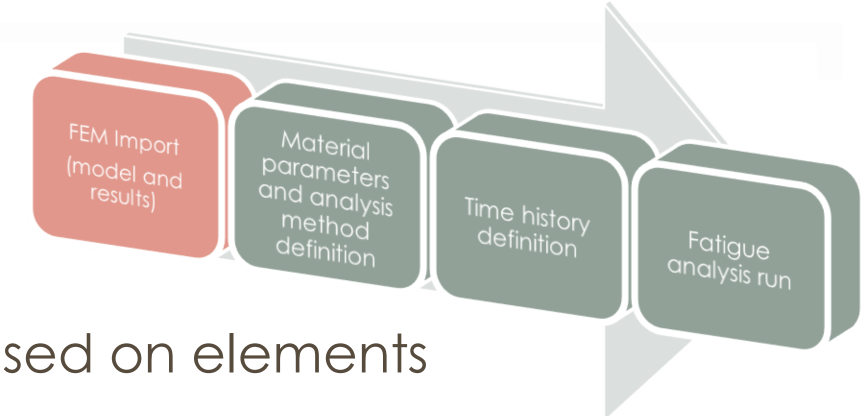
LIFING.Life



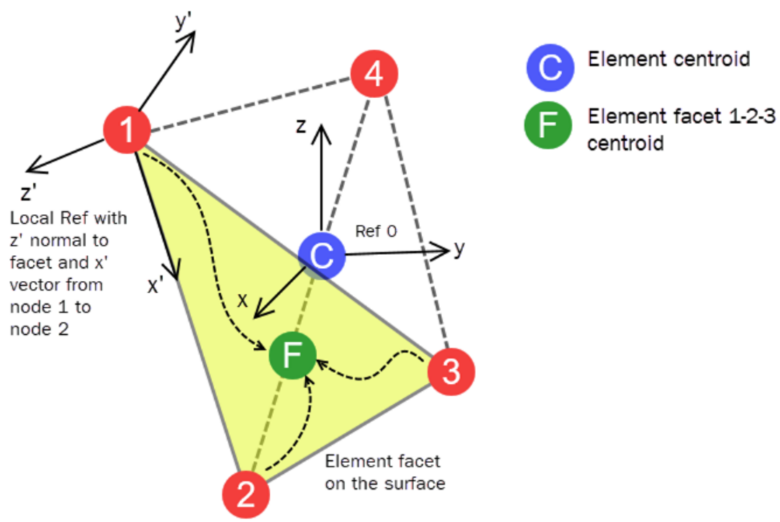
- Elemental stresses can be post-processed (both at the centroid and at the surfaces)
- Only elements and stresses at the surfaces are retained for fatigue analysis



LIFING.Life

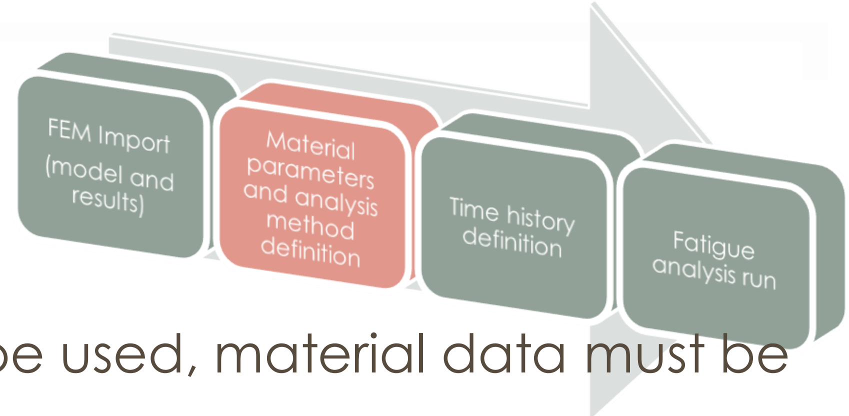


- LIFING.LIFE performs fatigue analysis based on elements
- Stresses at facets are:
  - Either taken from centroidal stresses and rotated to local reference system with Z' axis perpendicular to the element facet
  - Either derived from nodal stresses (if available in the result file) as shown in the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



  - Gradient effects can be accounted or stresses can be derived at a critical distance from the surface

LIFING.Life

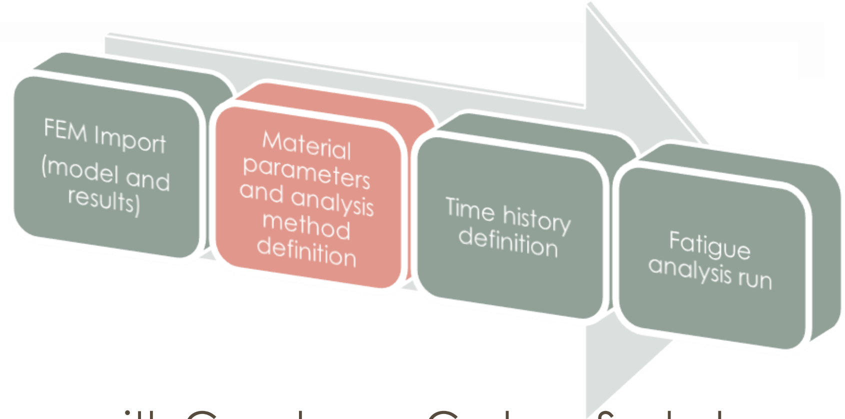


- Depending on the analysis method to be used, material data must be provided
  - S-N curves (generic format or military standard format)
  - Strain-life and cyclic stress-strain curves

- 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

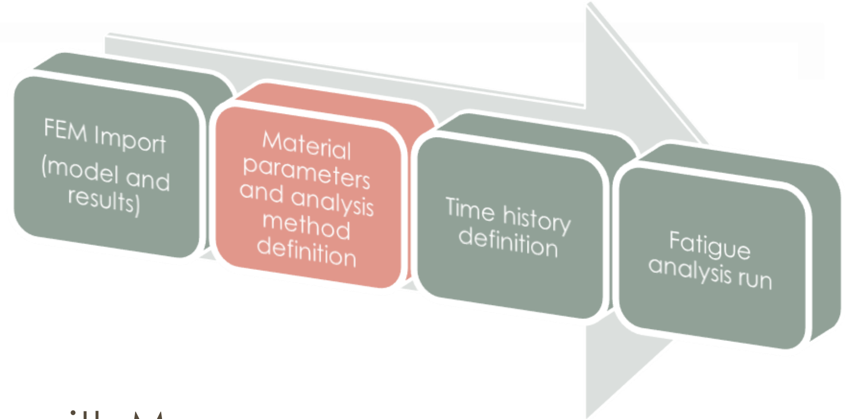
LIFING.Life



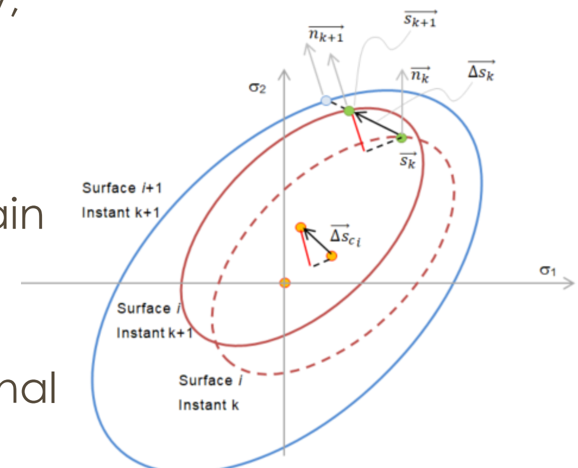
- Stress based methods
  - Uniaxial reduction (*) analysis based on S-N curves with Goodman, Gerber, Soderberg, Walker, Morrow, Smith-Watson-Topper, Haigh methods for mean stress levels accounting (it can also be decided not to consider mean stress corrections)
  - Uniaxial reduction (*) analysis based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

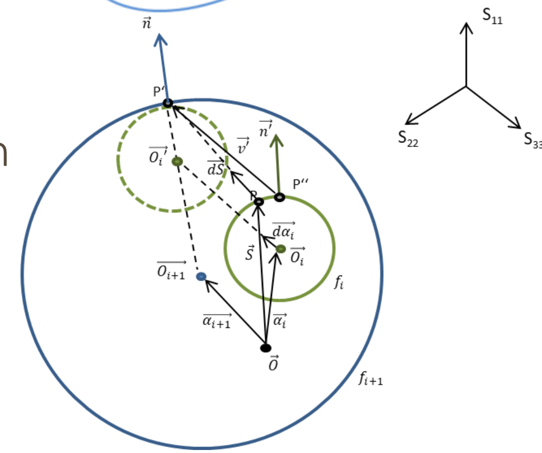
- (*) Uniaxial Reduction consists in a fatigue analysis carried out considering one stress contribution: the elemental stress tensor time history is projected at different angles and for each one the fatigue Life is calculated. The minimum Life is retained as element Life

LIFING.Life

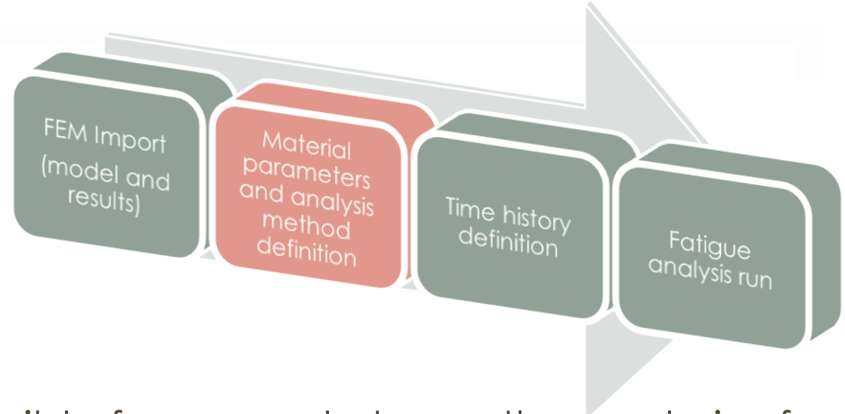


- Strain based methods
  - Uniaxial reduction analysis based on e-N curves with Morrow, Smith-Watson-Topper, Manson-Halford methods for mean stress levels accounting (it can also be decided not to consider mean stress corrections)
    - Plastic stress-strains derived with Neuber or Equivalent Strain 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



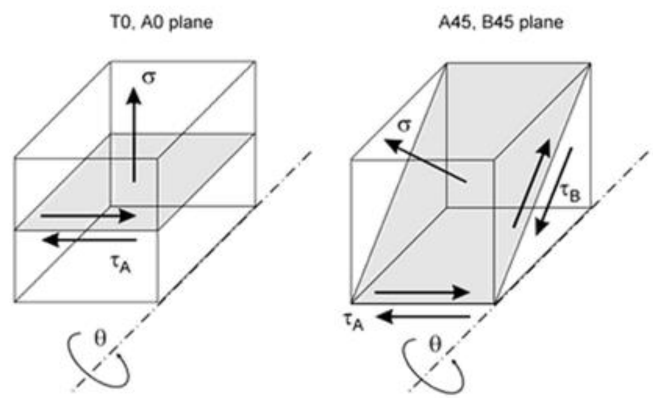


LIFING.Life

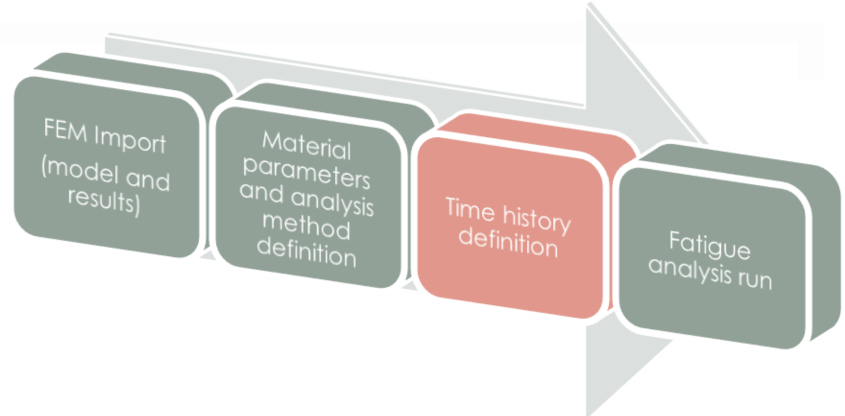


- Strain based methods
  - The user decides which method to use (it is possible for example to run the analysis of Non-Proportional Loading cases with Hoffman-Seeger or Dowling methods: in this case ‘average’principal angles and biaxiality ratios are calculated and the analysis is carried out considering the stress time history projected on this plane and accounting the calculated biaxiality ratio

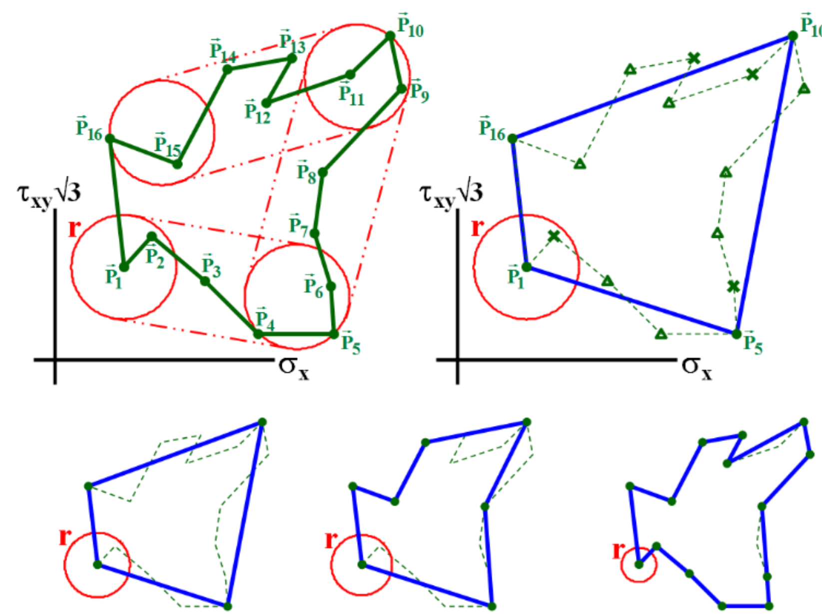
  - Implemented Fatigue parameters in case of Multiaxial Fatigue analysis:
    - Smith-Watson-Topper (plane T0)
    - Fatemi-Socie (planes A0, A45, B45)
    - Brown-Miller (with and without mean stress corrections) (planes A0, A45, B45)



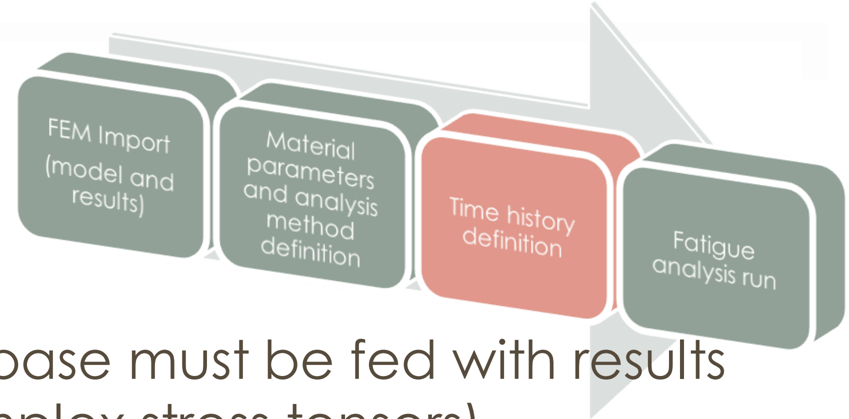
LIFING.Life



- 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)

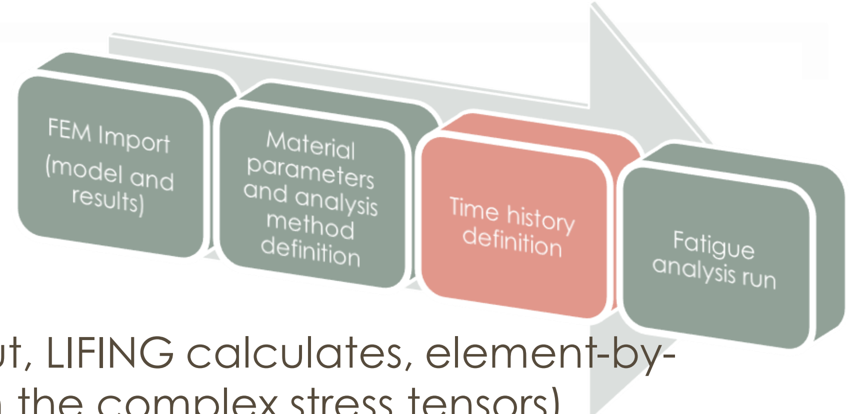


LIFING.Life

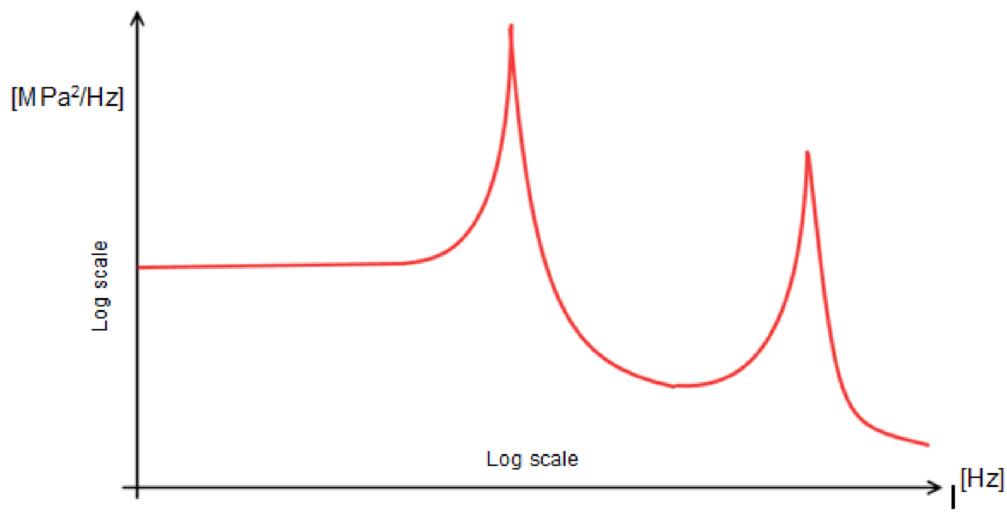


- If the analysis is based on PSD the database must be fed with results from Frequency Response analysis (complex stress tensors).
- LIFING calculates the corresponding Von Mises stress, from the complex tensor, for each frequency.
- An input PSD is provided, which acts as multiplicator for the Frequency Response analysis results.
- Cycles are calculated with Dirlik, Narrow Band and Stainberg methods

LIFING.Life

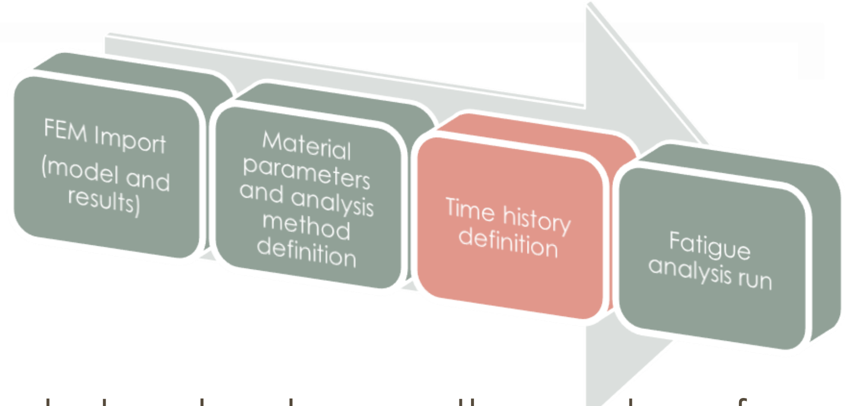


- From the FEM Frequency Response Analysis output, LIFING calculates, element-by-element, the Von Mises stress per g [MPa/g] (from the complex stress tensors).
- This quantity, to the power of 2, is scaled by the user defined accelerations PSD [g^2/Hz] and the Von Mises PSD [MPa^2/Hz] is derived.

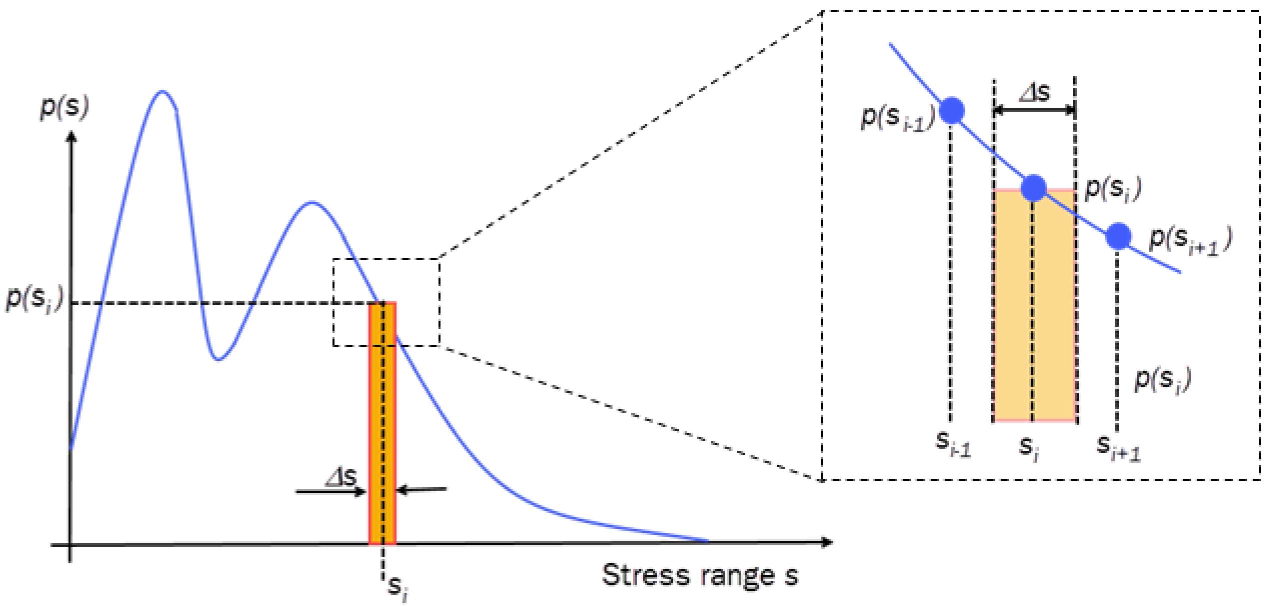


- The Dirlik or Narrow band can be therefore be applied to calculate the PDF (probability density function) of stress ranges s.

LIFING.Life

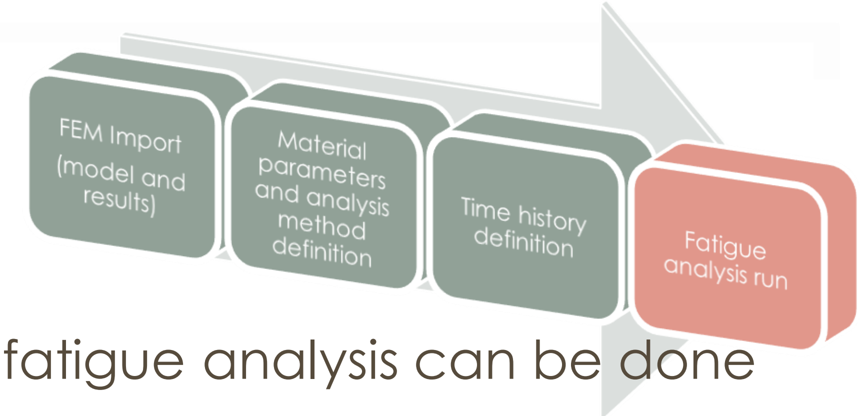


- The Dirlik method can be used to calculate, for each stress level range, the number of occurrences, to be used with stress or strain based methods and calculate fatigue Life.

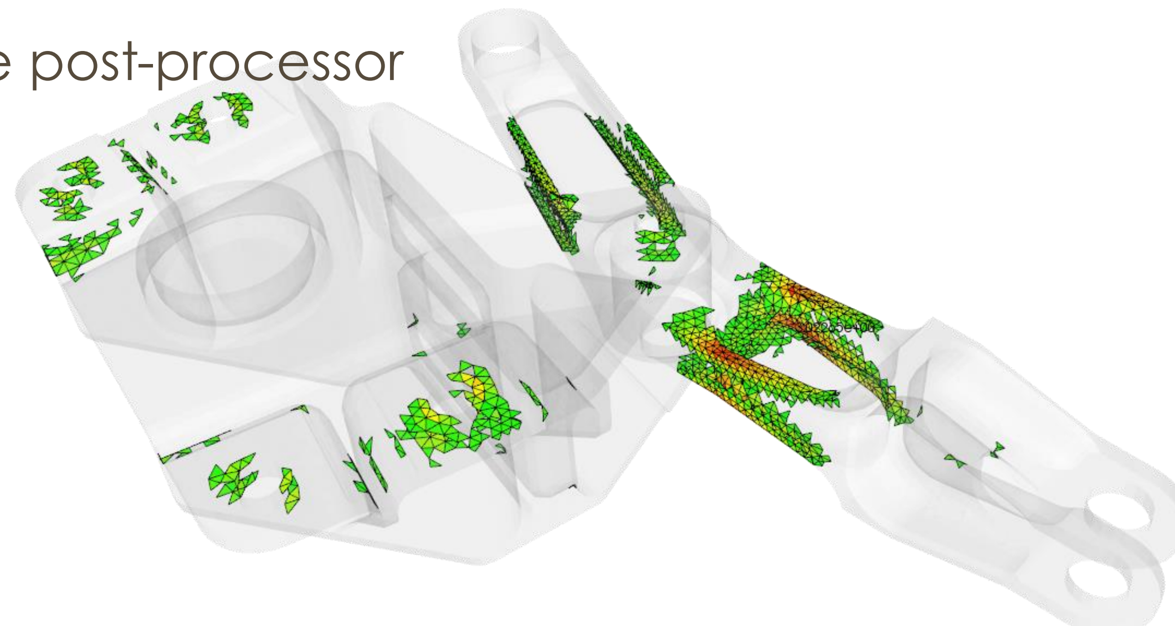


- Other methods such as Narrow Band and Stainberg can be used.

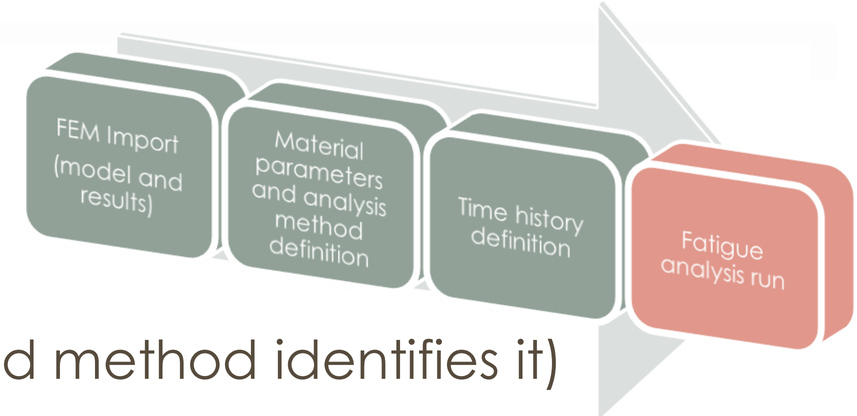
LIFING.Life



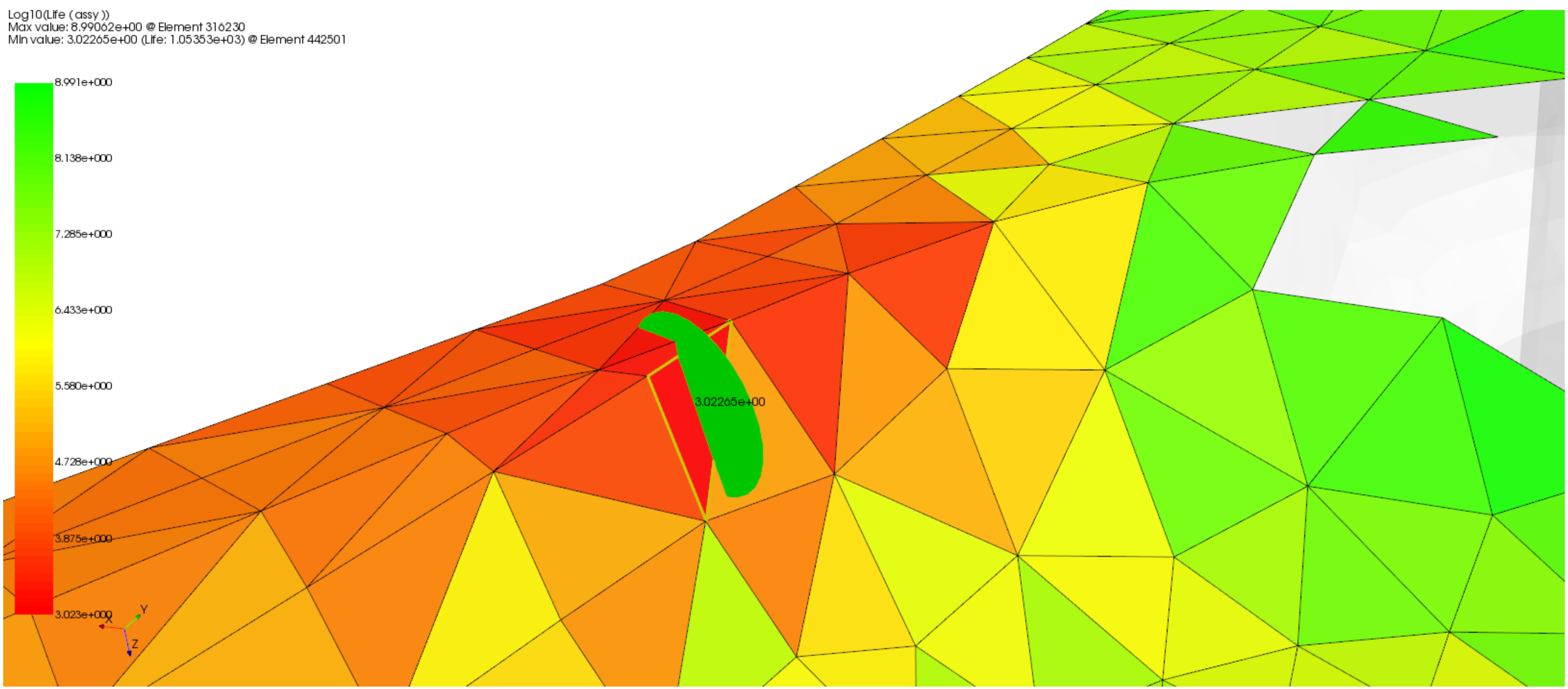
- Once the previous steps are done, the fatigue analysis can be done
  - On the entire FEM
  - On a defined set of Sub-Parts
  - On a defined Element
- Based on the defined analysis method, material properties, time history, etc.
- Fatigue Life is shown in the post-processor



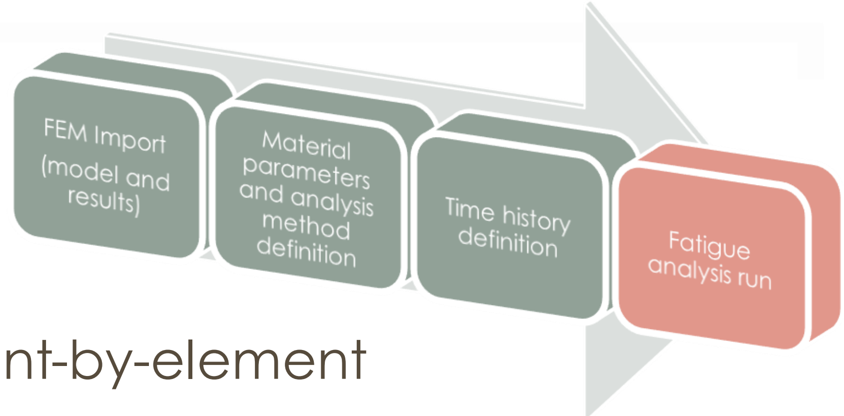
LIFING.Life



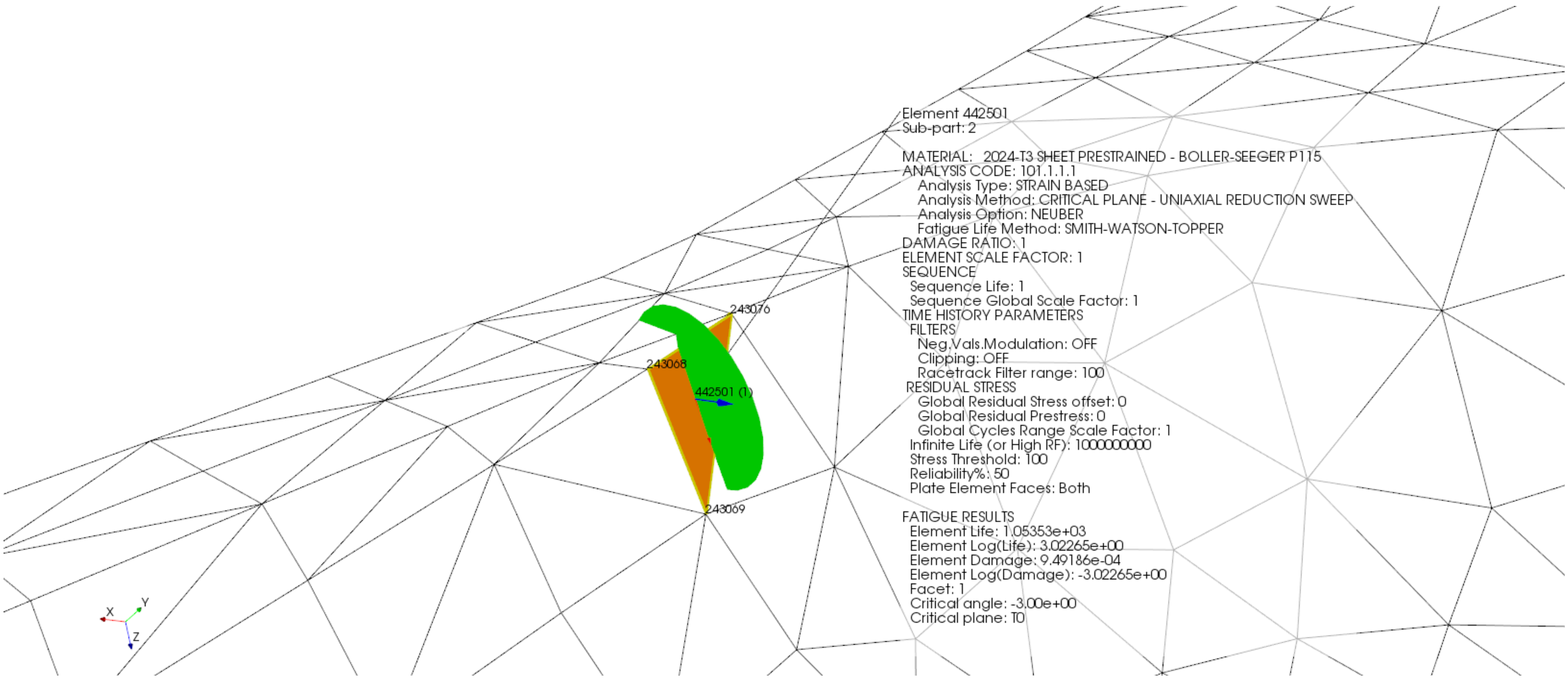
- The critical plane is shown (if the defined method identifies it)



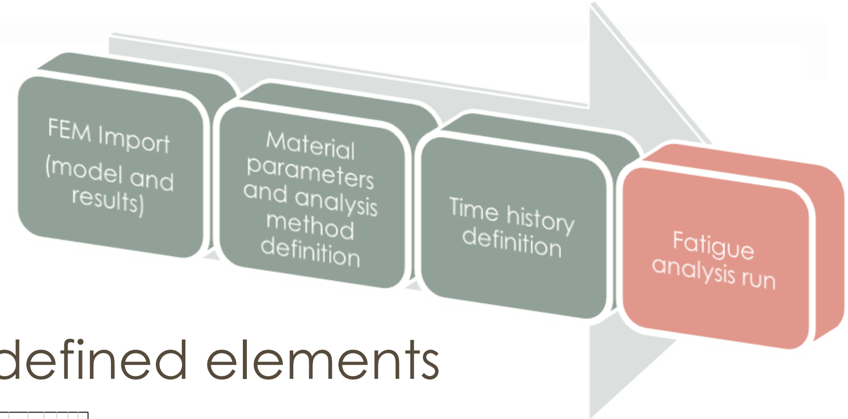
LIFING.Life



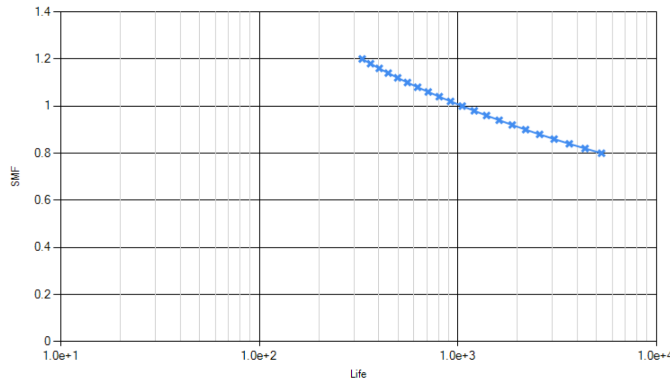
- The solution can be interrogated element-by-element



LIFING.Life



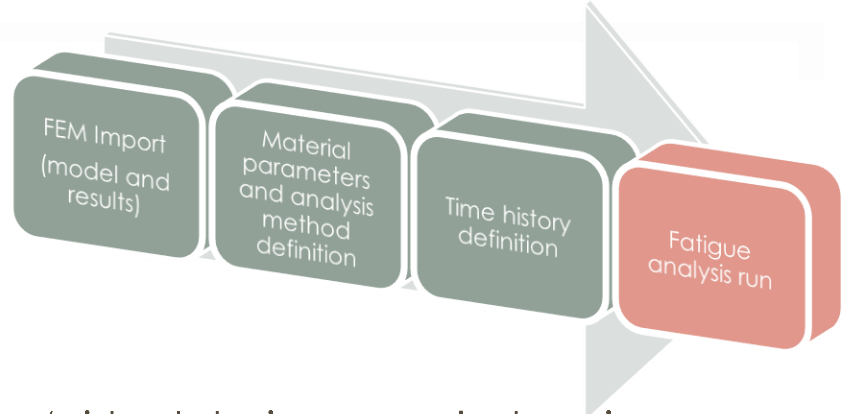
- The user can run 'sensitivity analysis' at defined elements



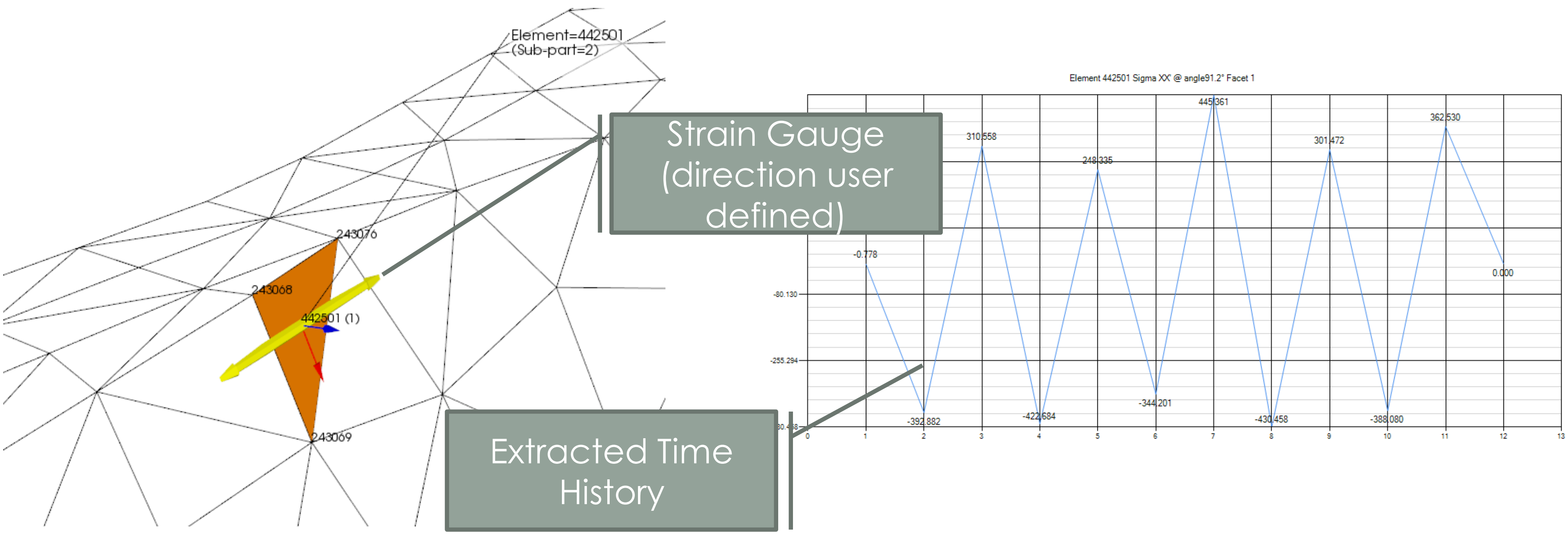
- LIFING provides in a ASCII file the breakdown of the damage calculation (useful for debugging the analysis or for further investigations)

```
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
```

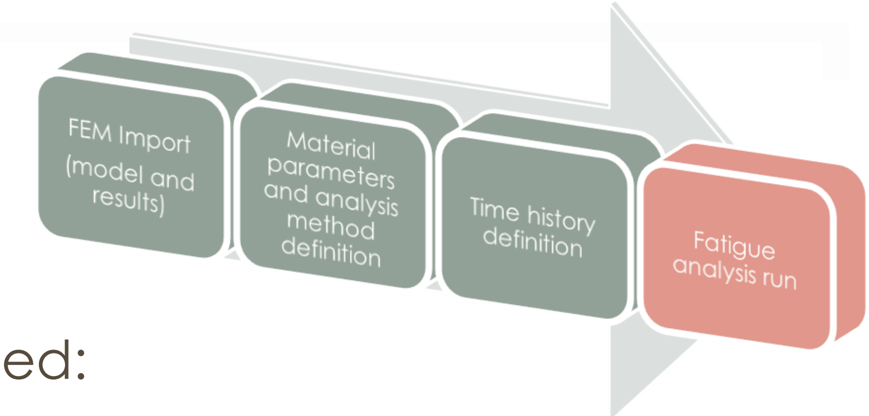
LIFING.Life



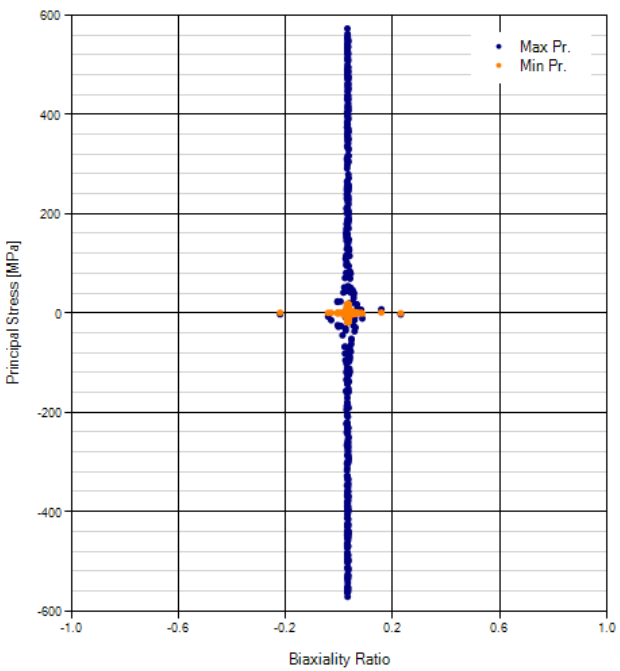
- VIRTUAL STRAIN GAUGE
  - The user can interrogate the model by putting a ‘virtual strain gauge’ at a given element, orient it at a given angle, and the related stress time history can be directly extracted

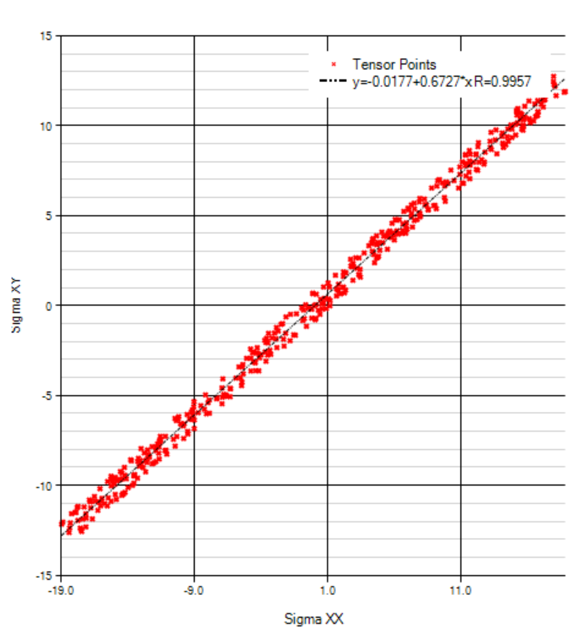


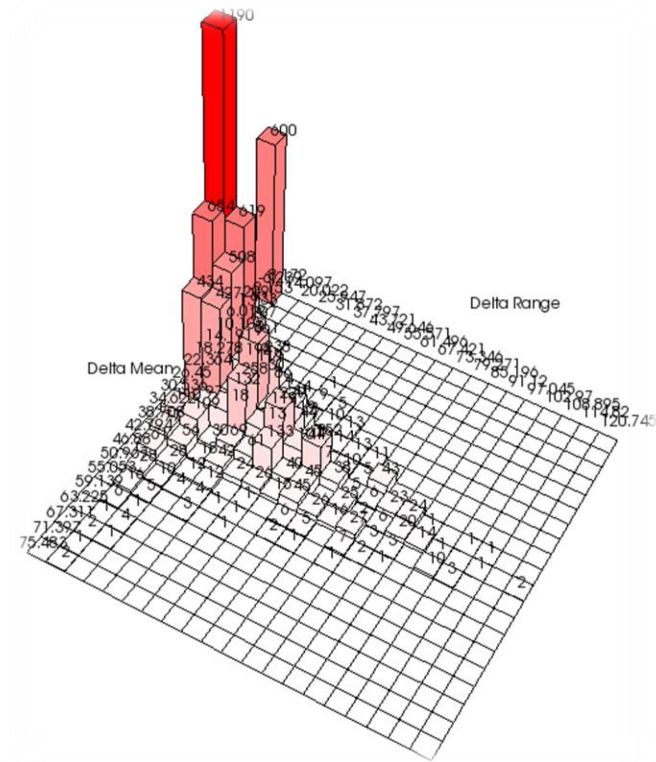
LIFING.Life



- Additionally the following can be derived:
  - Biaxiality plots and histograms
  - Time histories cycle counting/filtering, ...



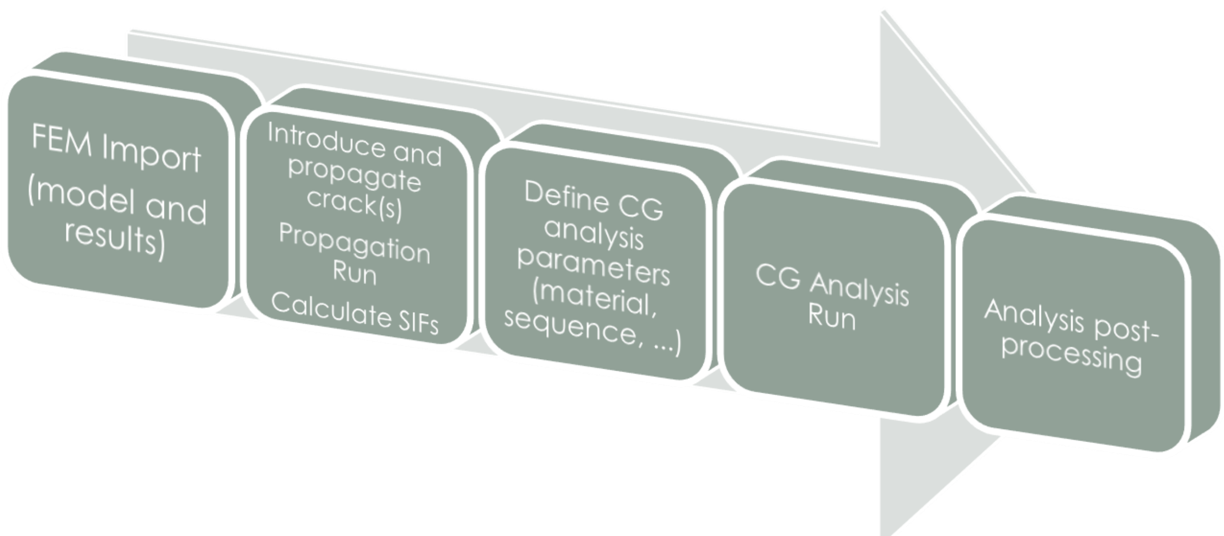




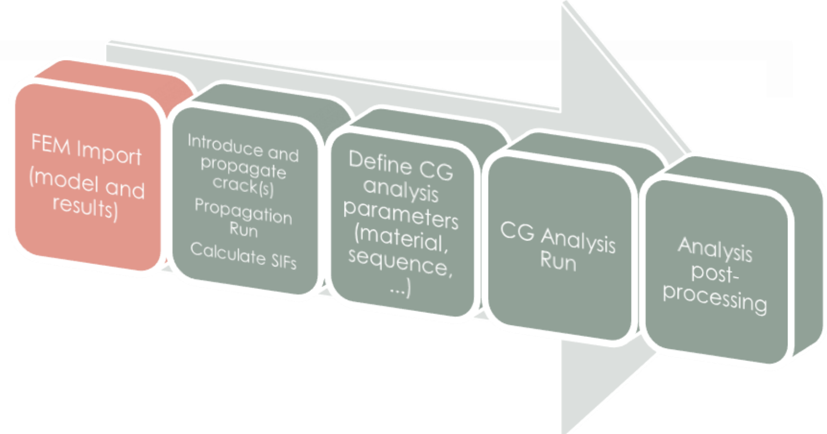
LIFING.Growth

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

The workflow:

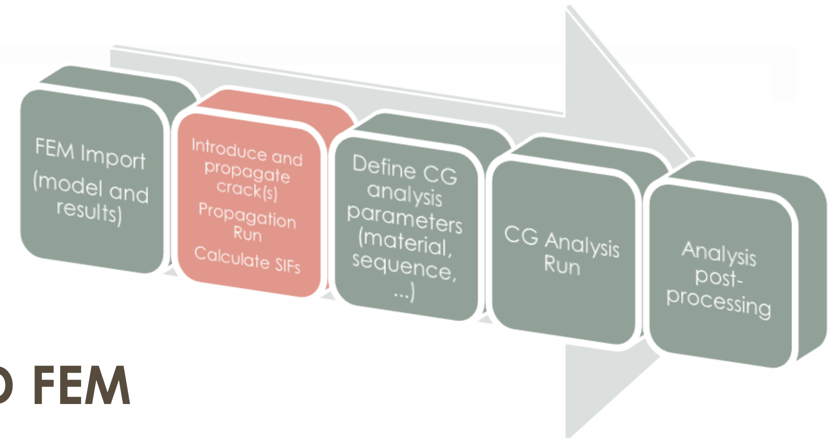


LIFING.Growth



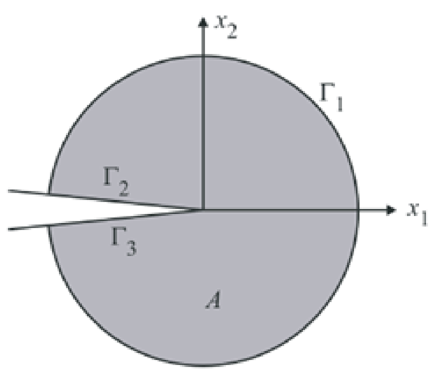
- **FEM Import**
  - A FEM can be imported or internally generated with the QUICK2DFEM module.
  - Depending on the FEM type, LIFING.GROWTH will calculate Stress Intensity Factors with J-Integral (2D) or M-Integral (3D) with the internally coded 2D and 3D solvers.
  - In both cases the FEM is created such to have 2nd order elements with special templates at the crack tip (2D) or crack front (3D).

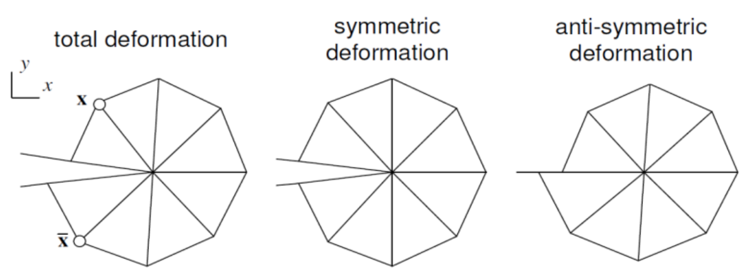
LIFING.Growth



- **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 (using quarter point elements)

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



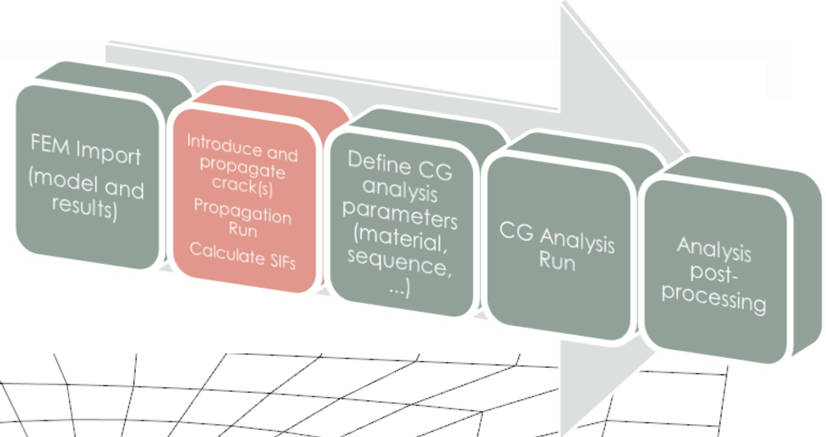


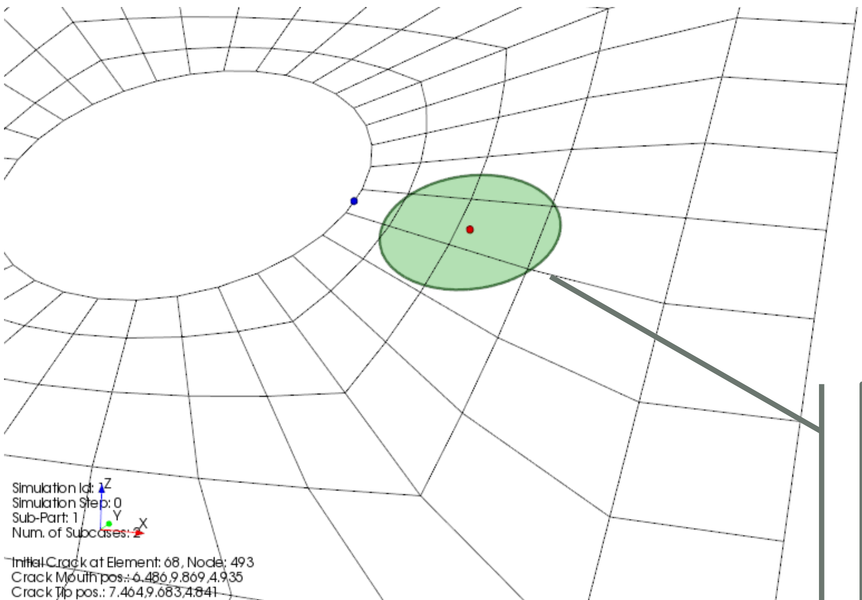
$$G_I = J_I = J(\mathrm{u}_{sym}, \boldsymbol{\sigma}_{sym})$$

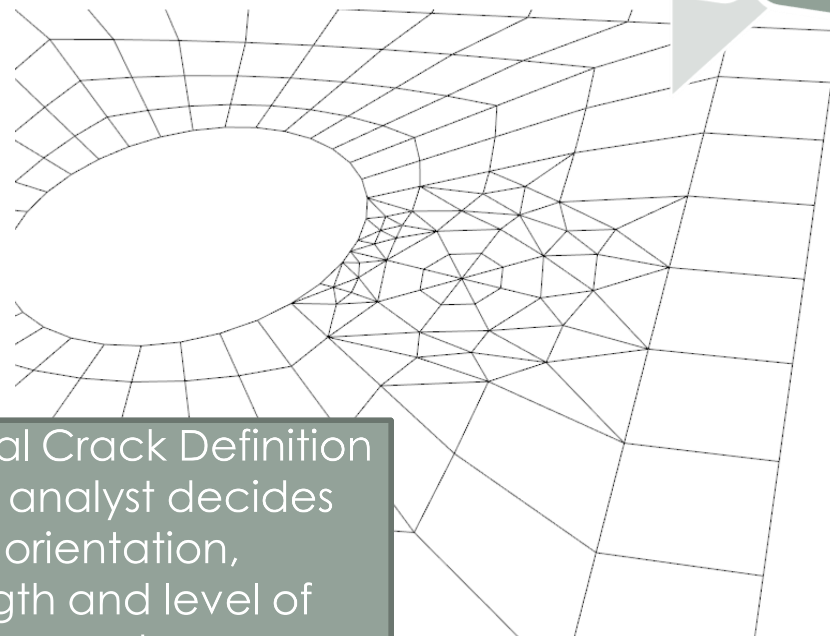
$$G_{II} = J_{II} = J(\mathrm{u}_{anti-sym}, \boldsymbol{\sigma}_{anti-sym})$$

$$\mathrm{u} = \mathrm{u}_{sym} + \mathrm{u}_{anti-sym} \qquad \mathrm{u}_{sym} = \tfrac{1}{2}\begin{Bmatrix} u + \bar{u} \\ v - \bar{v} \end{Bmatrix} \qquad \mathrm{u}_{anti-sym} = \tfrac{1}{2}\begin{Bmatrix} u - \bar{u} \\ v + \bar{v} \end{Bmatrix}$$

LIFING.Growth



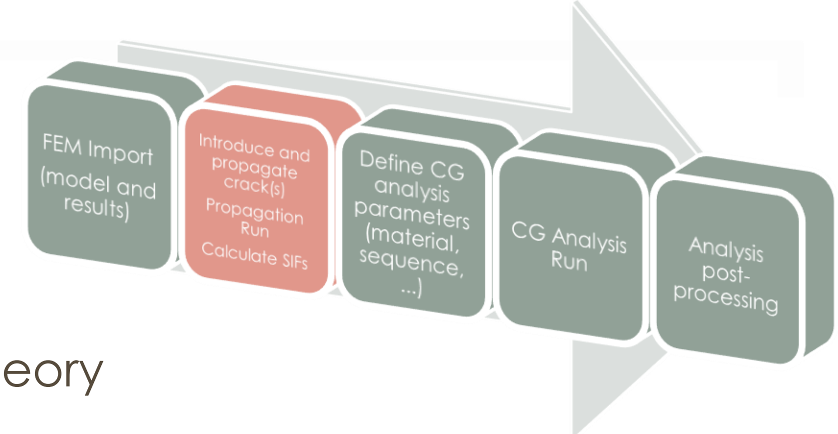




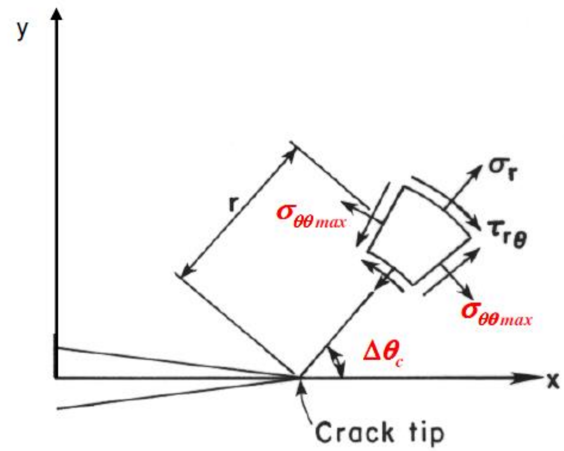
Initial Crack Definition
The analyst decides the orientation, length and level of refinement

- After being introduced, the crack(s) propagation is performed automatically

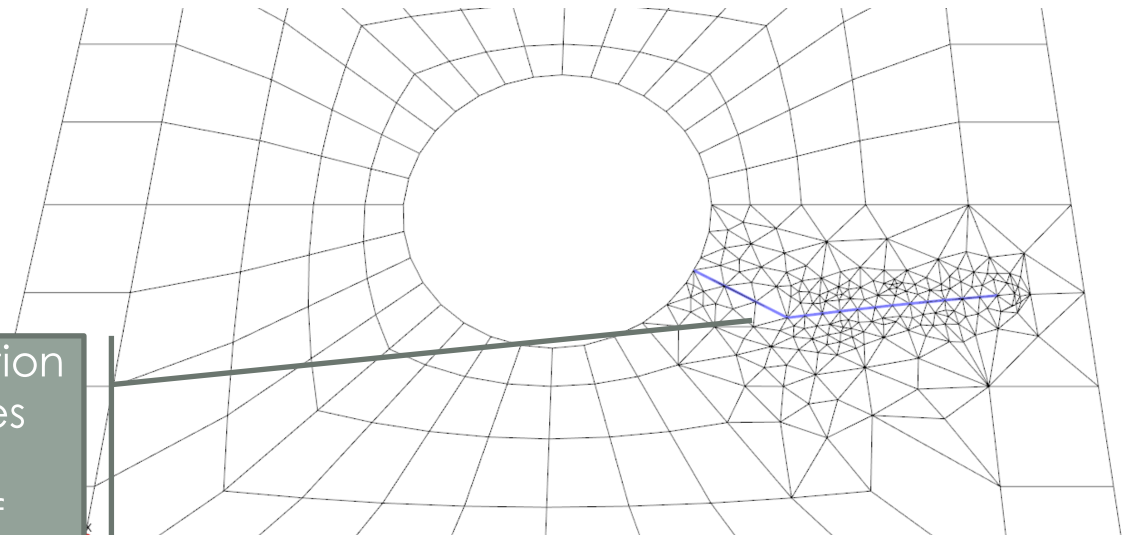
LIFING.Growth



- Crack Kinking calculated with the $\sigma_{\theta\theta,max}$ theory

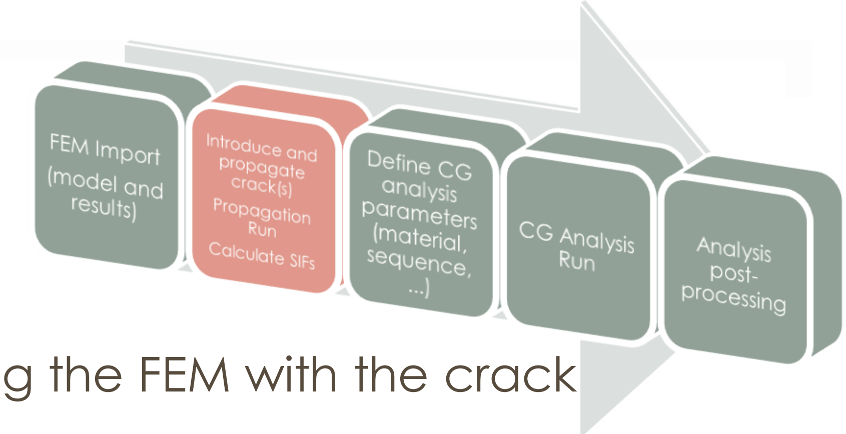


$$\frac{K_{II}}{K_I} = \frac{-\sin\Delta\theta_c}{(3\cos\Delta\theta_c - 1)} \quad \rightarrow \quad \Delta\theta_c = 2\tan^{-1}\left(\frac{1-\sqrt{1+8(K_{II}/K_I)^2}}{4(K_{II}/K_I)}\right)$$

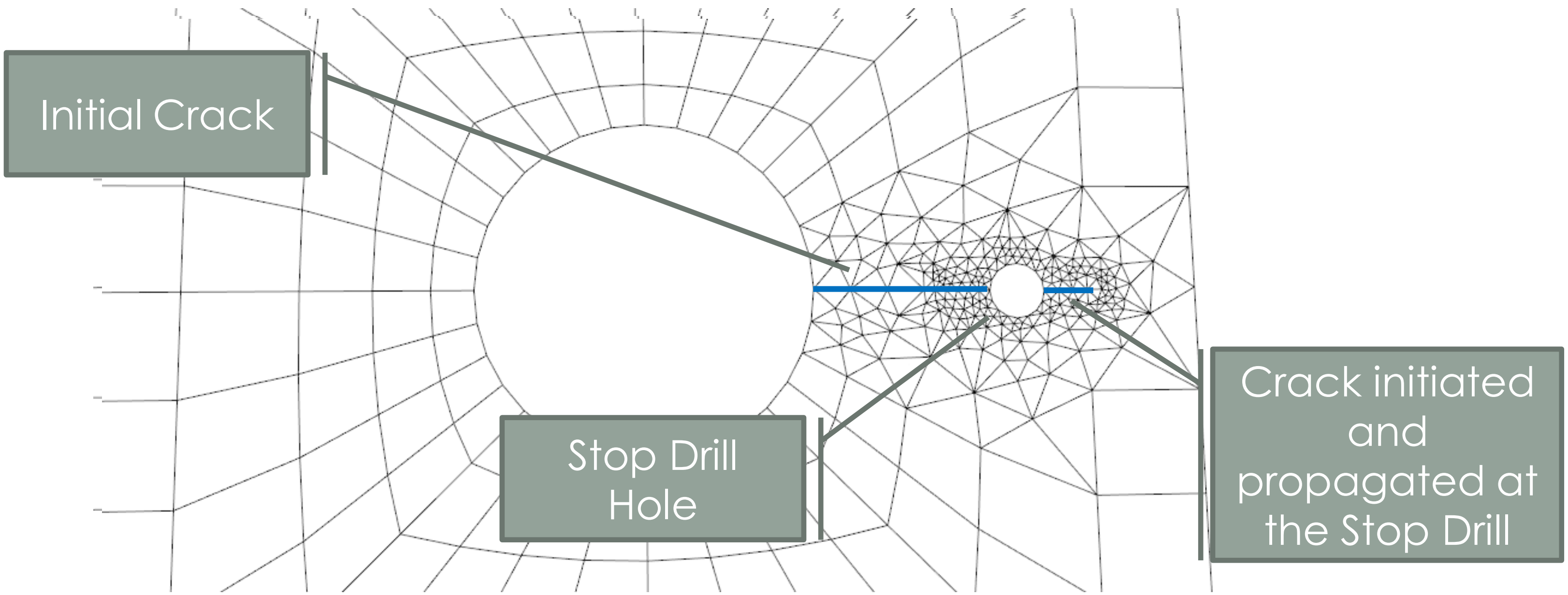


Initial Crack Definition
The analyst decides the orientation, length and level of refinement

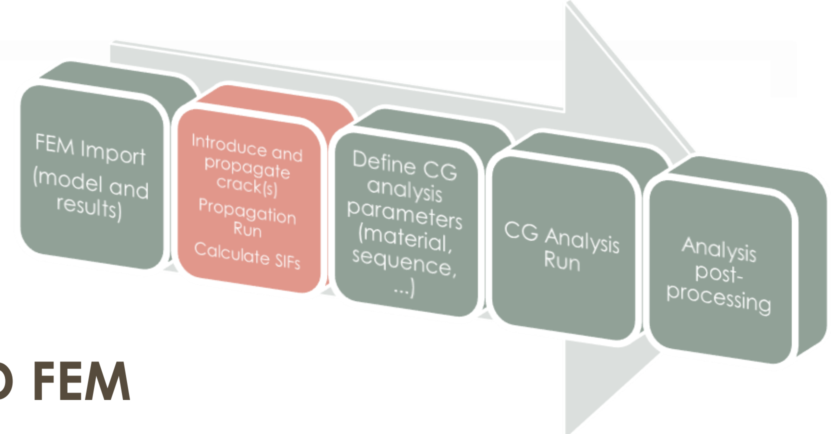
LIFING.Growth



- 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



LIFING.Growth

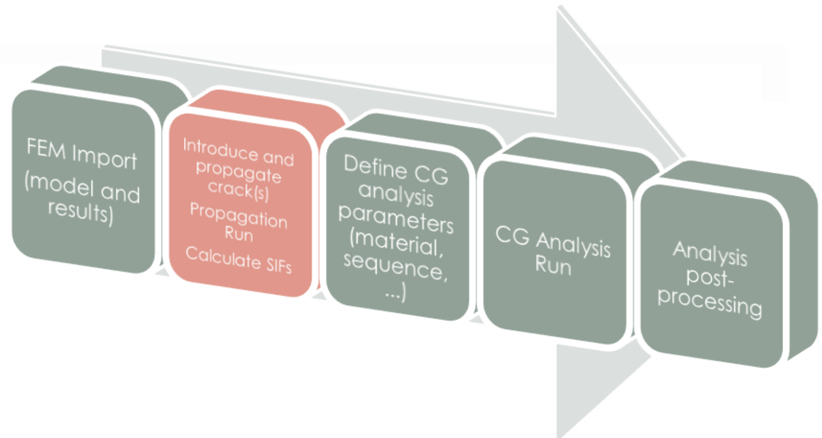


- **Crack(s) propagation simulation in 3D FEM**
  - The module Quick2DFEM can be used to generate a 3D by extrusion of a 2D mesh which is simply generated with the QUICK2DFEM script.
  - Both linear and circumferential extrusions are allowed.
  - A special QUICK2DFEM card is used to introduce a crack front (elliptic or parabolic or linear shape).
  - Once generated, the FEM with the crack front is solved and Stress Intensity Factos are calculated with M-Integral

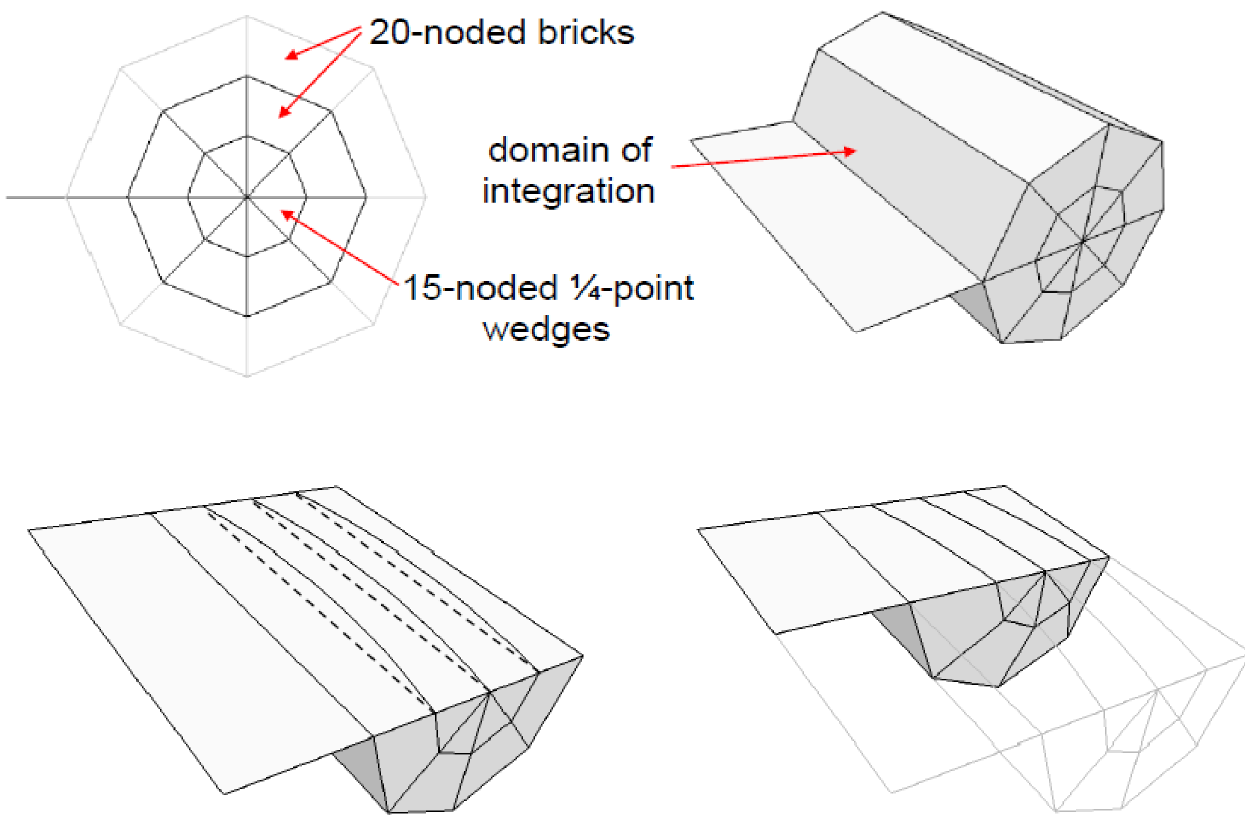
$$M^{(1,2)} = \int_{\Gamma} \left({\sigma_{ij}}^{(1)} \frac{\partial {u_i}^{(2)}}{\partial x_1} + {\sigma_{ij}}^{(2)} \frac{\partial {u_i}^{(1)}}{\partial x_1} - W^{(1,2)} \delta_{1j} \right) \frac{\partial q}{\partial x_j} ds =$$

$$= \frac{1 - v^2}{E} K_I^{(1)} K_I^{(2)} + \frac{1 - v^2}{E} K_{II}^{(1)} K_{II}^{(2)} + \frac{1 + v^2}{E} K_{III}^{(1)} K_{III}^{(2)}$$

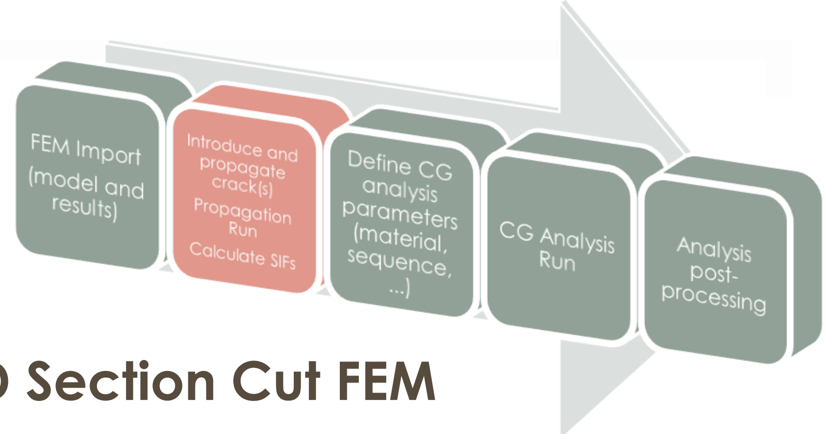
LIFING.Growth



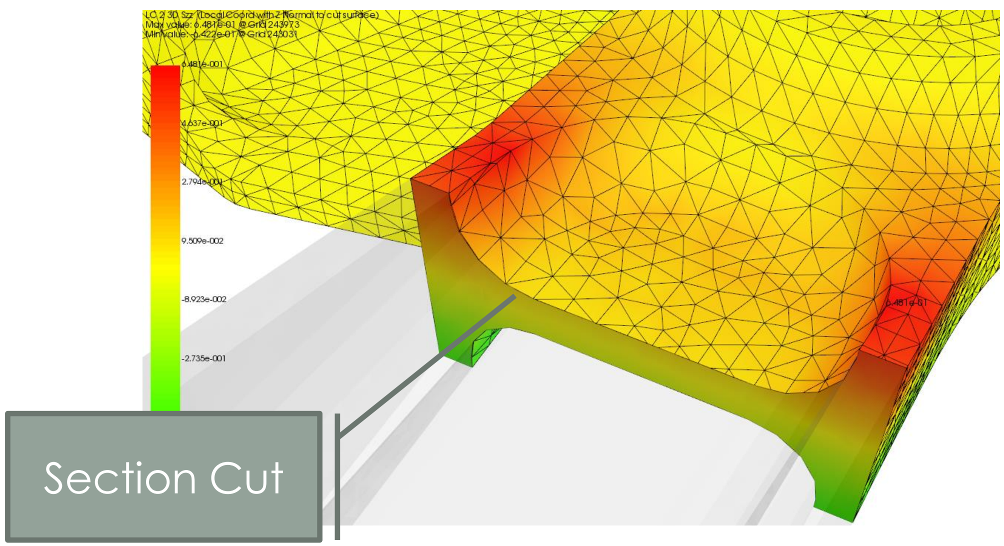
- A special template of elements are used



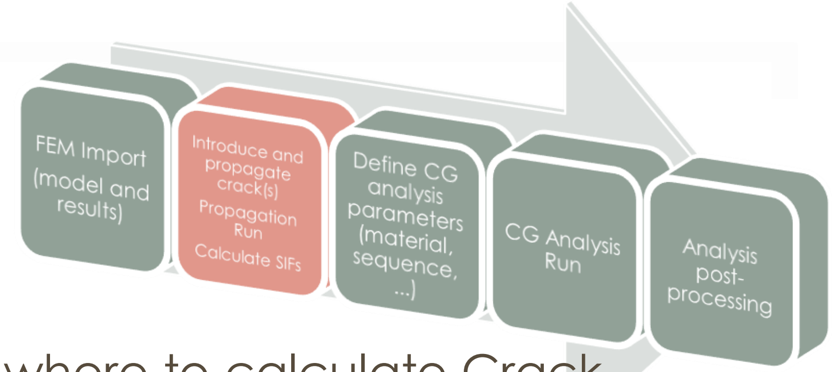
LIFING.Growth



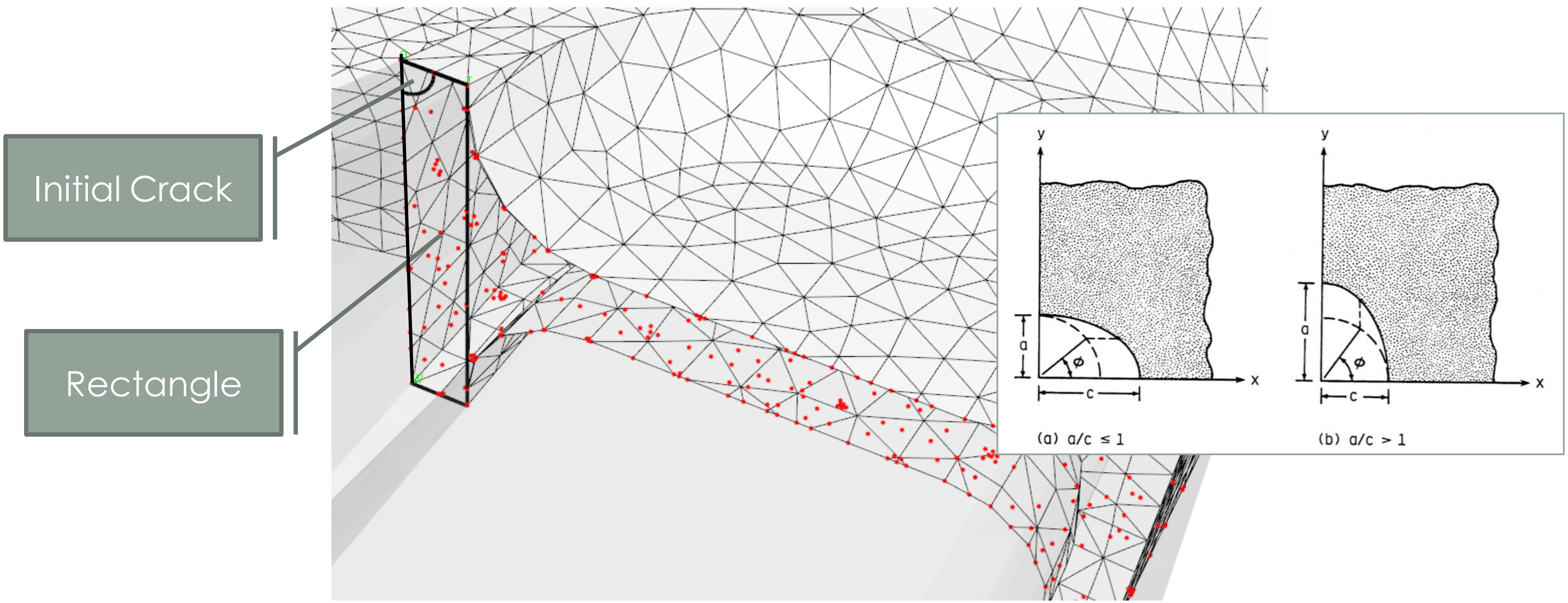
- **Crack(s) propagation simulation in 3D Section Cut FEM**
  - Lifing offers the possibility to perform quick Crack Growth estimation at section cuts
  - Nodal stress results must be preliminarily imported, in order to have stress data «inside» the FEM
  - A Section Cut is defined by the user (or automatically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



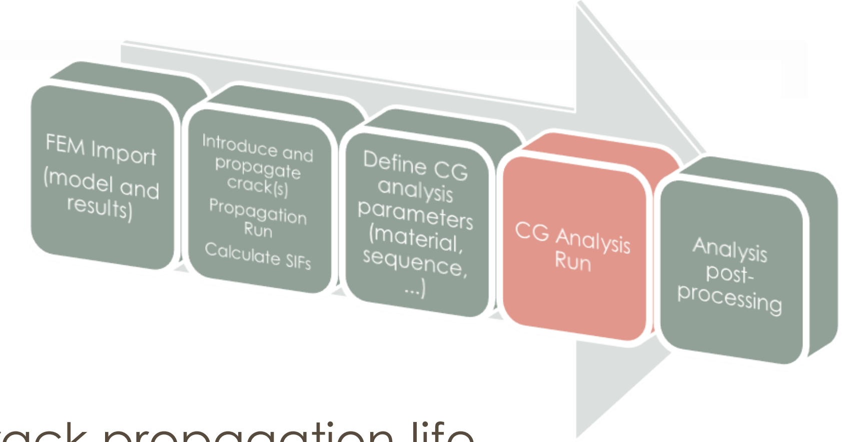
LIFING.Growth



- The user defines a Rectangular 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

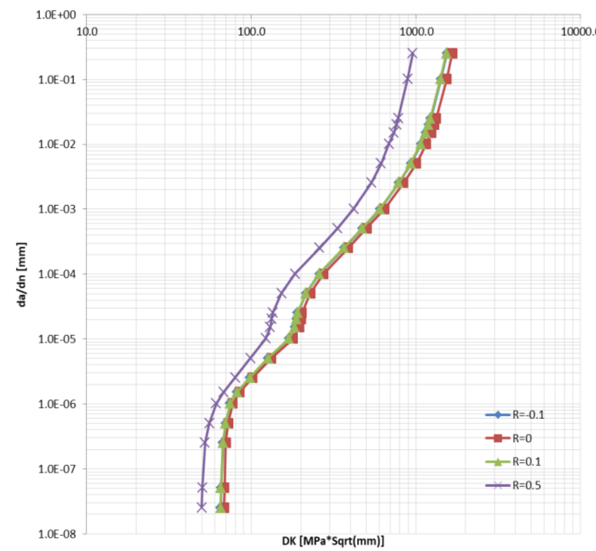


LIFING.Growth



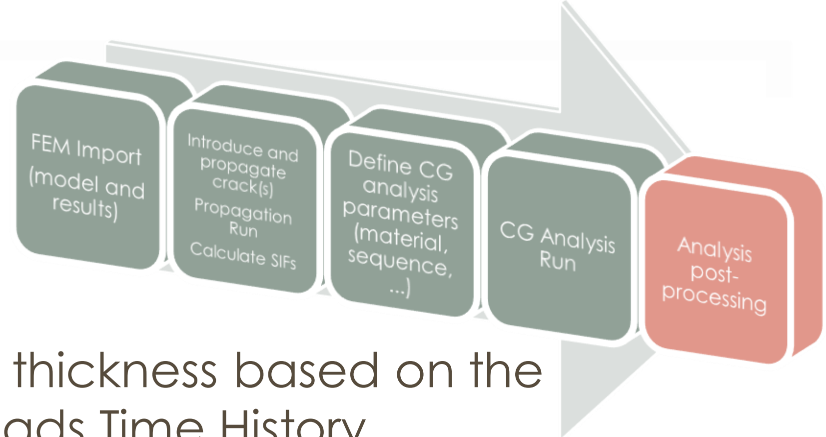
- **Crack Growth calculation**
  - NASGRO model is used to calculate crack propagation life

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

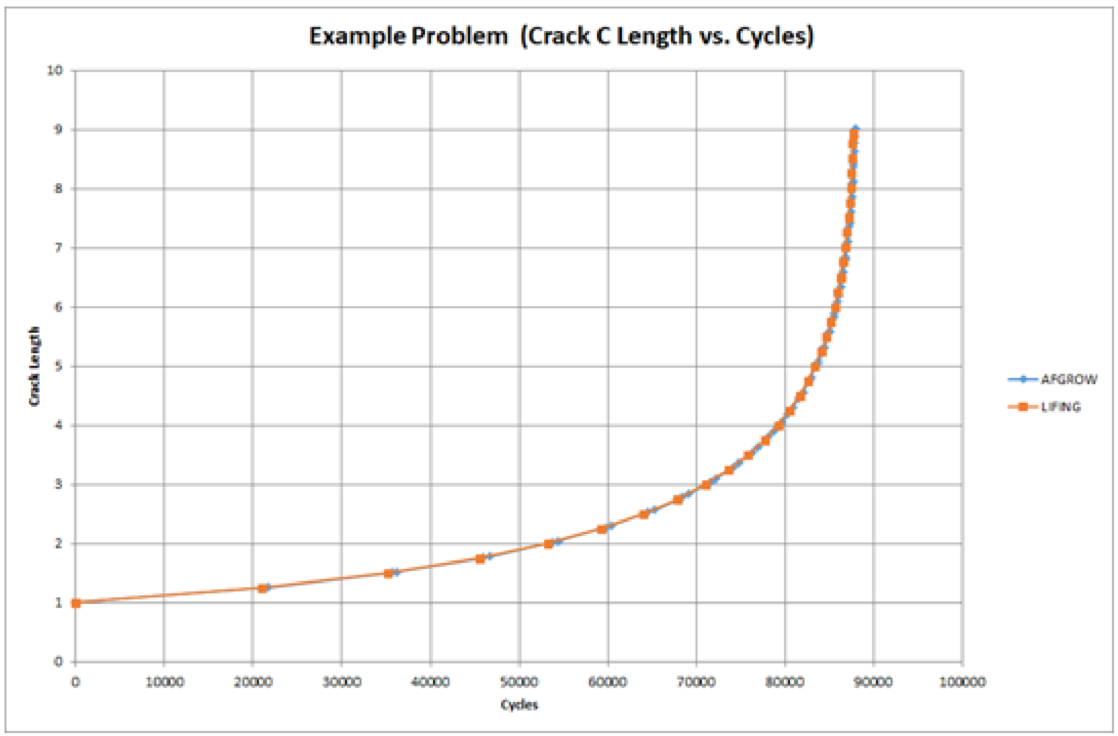


  - Alternatively da/dN in tabular format can be used
  - Multiple cracks growth calculated (relative growth rate accounted with Paris law)
  - Generalized Willemborg Crack Closure model implemented

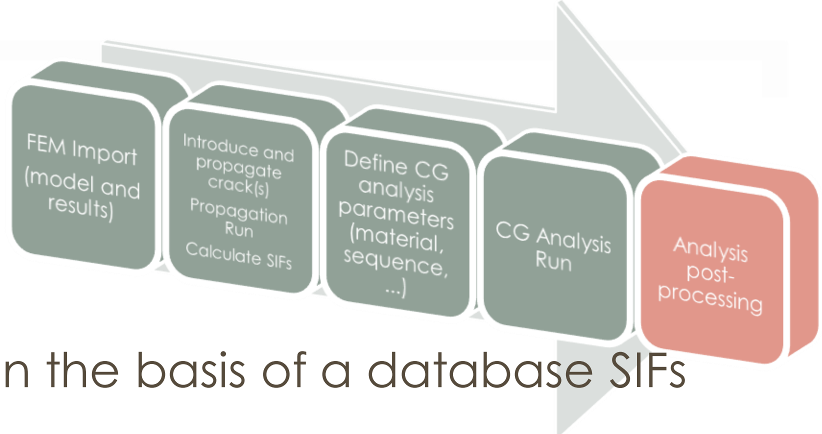
LIFING.Growth



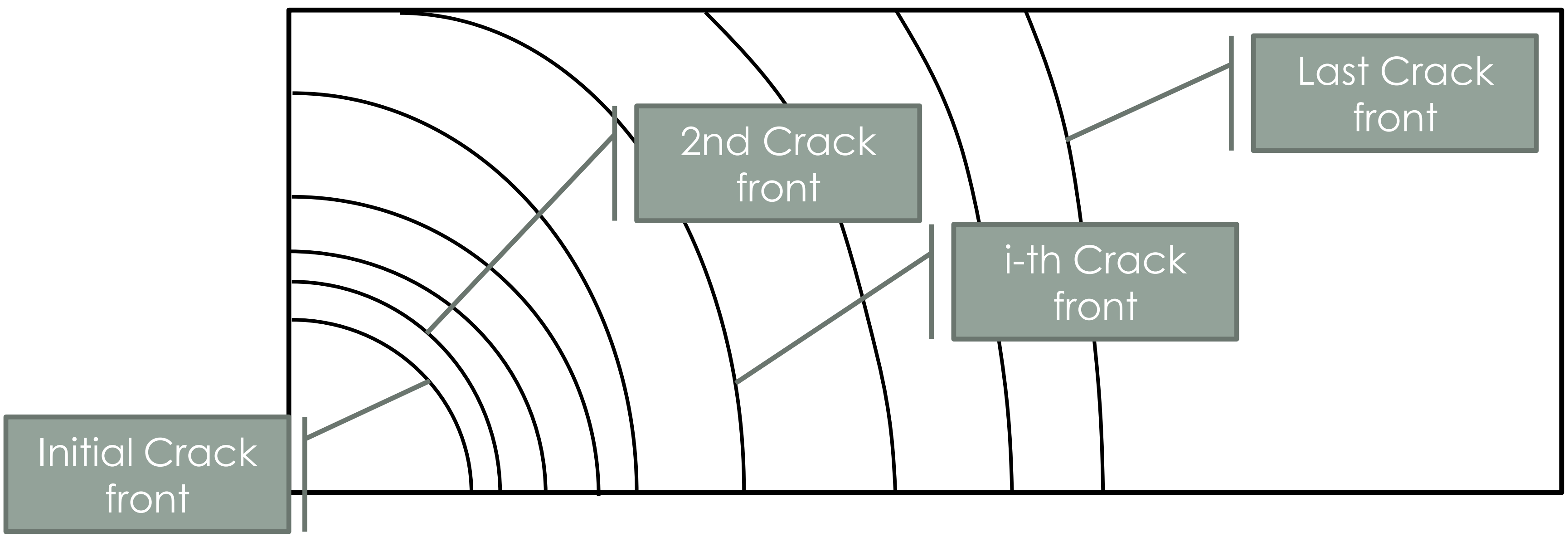
- The crack is automatically grown in the thickness based on the material NASGRO data and applied Loads Time History
- Implemented approach provides excellent adherence to AFGROW solution



LIFING.Growth

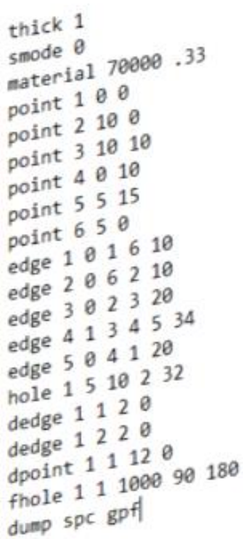


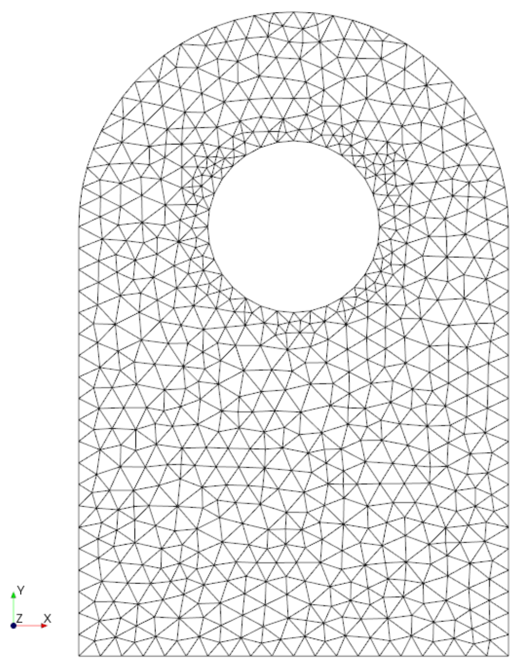
- A CG analysis can also be performed on the basis of a database SIFs externally calculted (from other SW)
- LIFING in this case uses the crack front computed SIFs and uses a 2D interpolator to evaluate correct SIFs while the crack is propagating

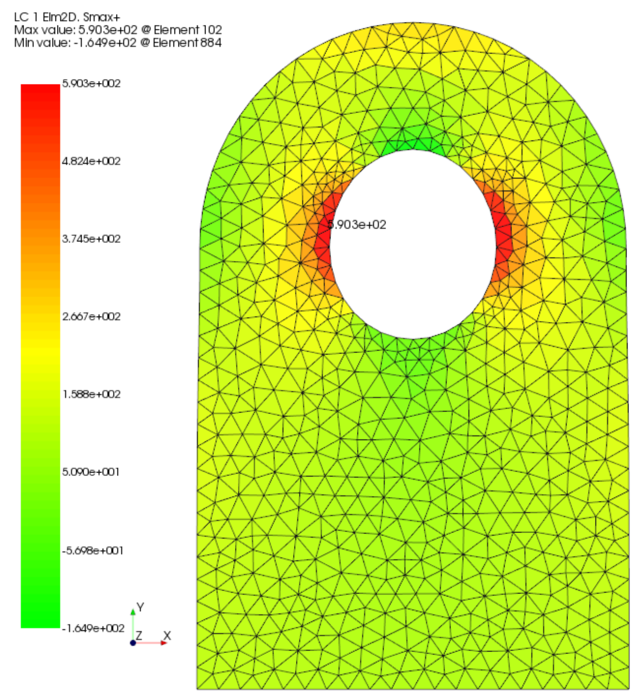


QUICK2DFEM

- QUICK2DFEM provides the capability to run fatigue analyses on 2D and 3D FEM generated with simple script files
- The user defines model edges (straight or curved) with number of elements, holes, boundary conditions and material properties, in a simple file format
- The 2D model is immediately built and solved with the internal 2D solver

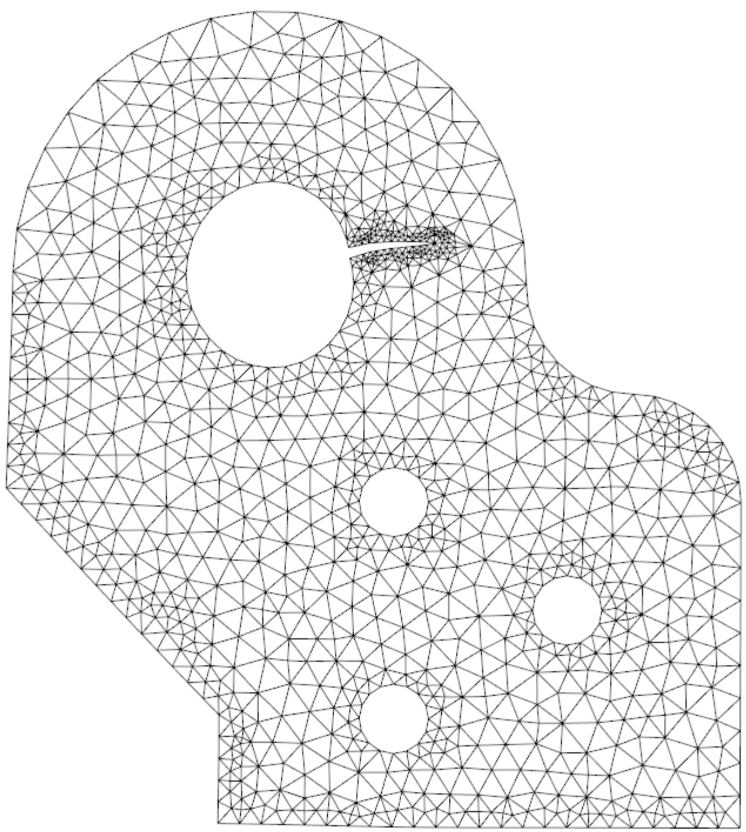


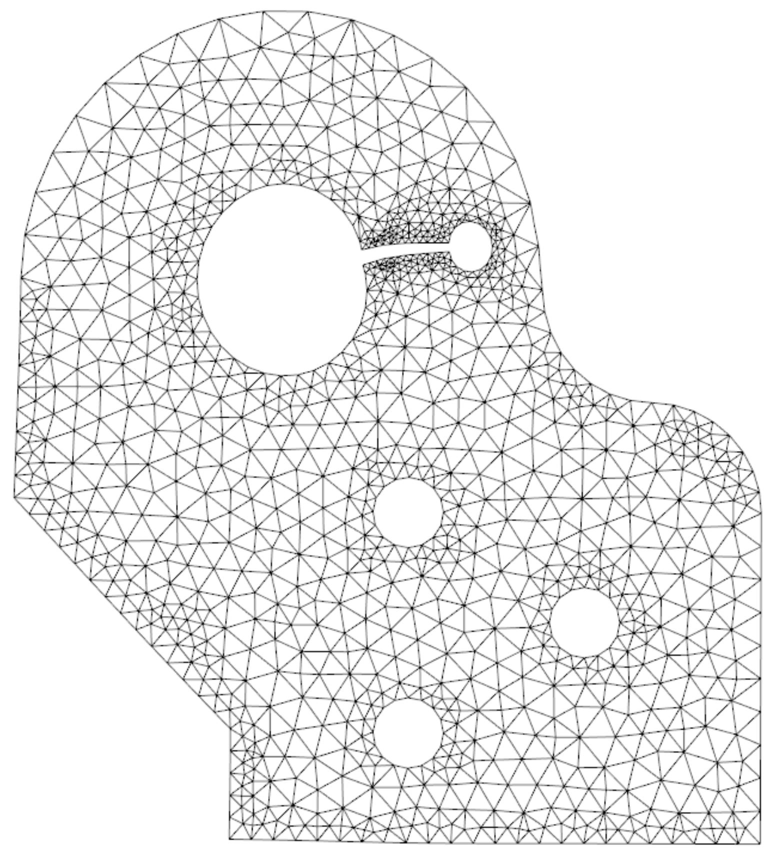




QUICK2DFEM

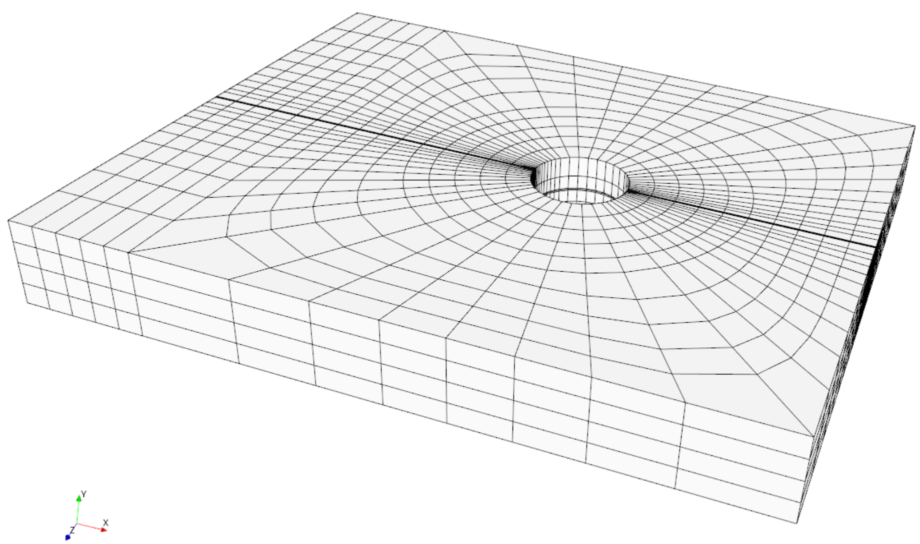
- This capability allows immediate evaluation of Stress Concentration Factors in 2D models and F-DT analysis with the modules LIFING.LIFE or LIFING.GROWTH

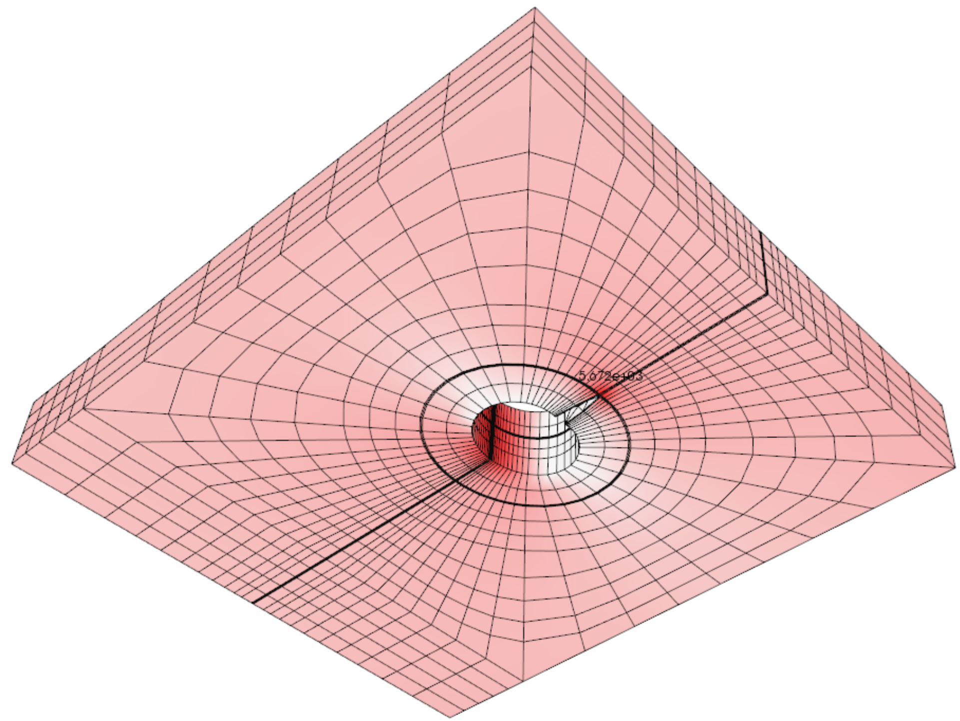




QUICK2DFEM

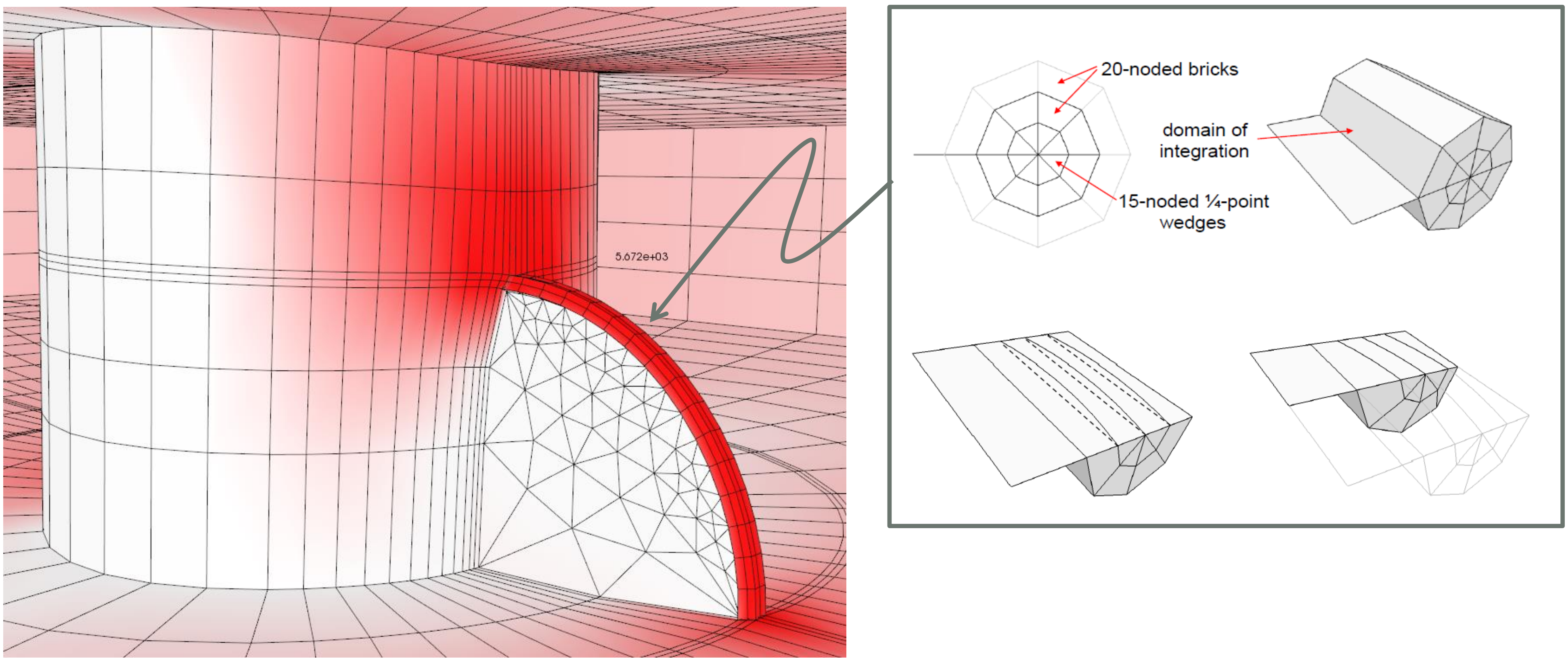
- With QUICK2DFEM it is possible to extrude linearly or circumferentially the generated 2D mesh, that can be solved with the internally coded 3D solver.
- A feature is implemented to introduce crack fronts.





QUICK2DFEM

- The crack front is built with a template of elements, including quarter point elements, such to allow the M-Integral calculation and derivation of Stress Intensity Factors.



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