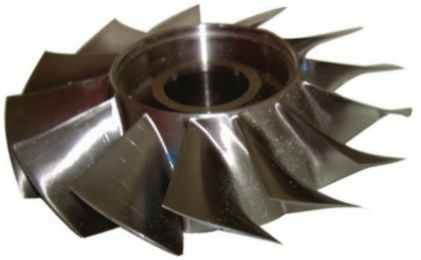
Z-set FE solver tutorial

Turbine disc application

December 11, 2017

Turbine disk

Dimensioning of the turbine disk is of paramount importance for the integrity of the engine in service.



(a) Monoblock

(b) Disc with notches

(c) Turbine blades assembled into notches

Turbine disk

Objective : Material characterization and life time estimation of a representative laboratory model of the turbine disc.

The following simplifications are adapted for the representative model:

- The blades of the assembly are excluded to simplify the geometry.
- Linear loading histories for thermal and mechanical loading are used.
- Simple material and damage models are used.

Process flow

1.FE setup

- Mesh generation
- Material definitions
- Loading & BC

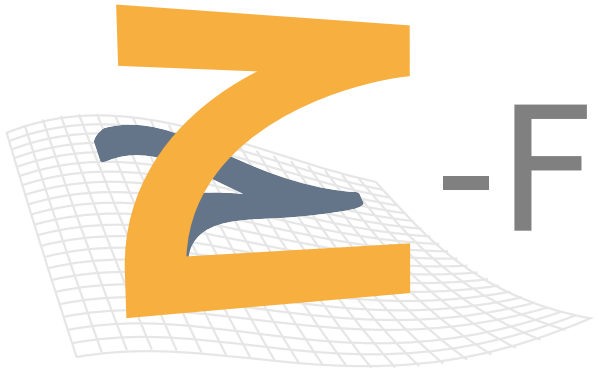
2. Analysis

3.Post processing

- General PP
- Fatigue life estimate
- Hot spot detection

4.Fracture analysis

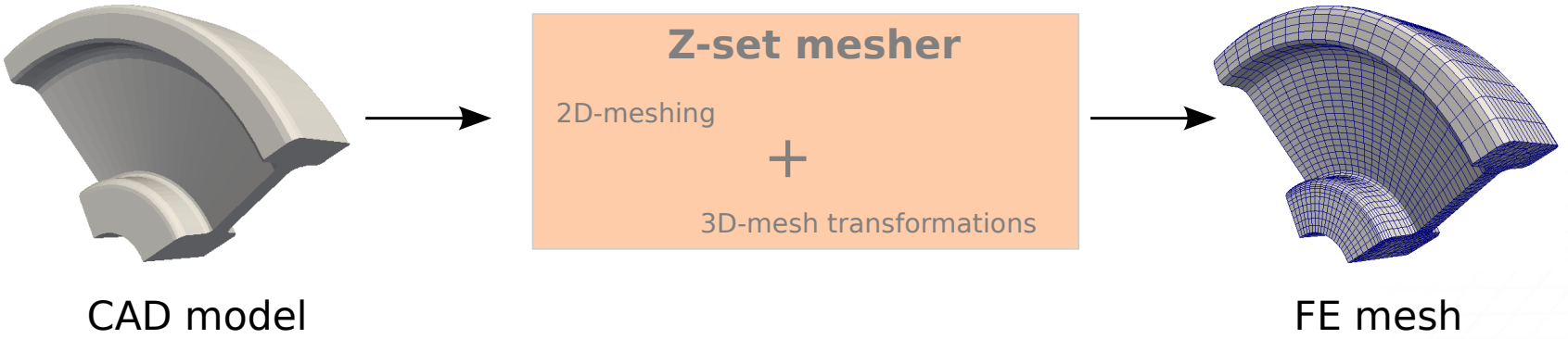
- Crack definition
- Crack propagation



-FE setup

Mesh generation

- With Z-set native CAD tools (Zmaster)

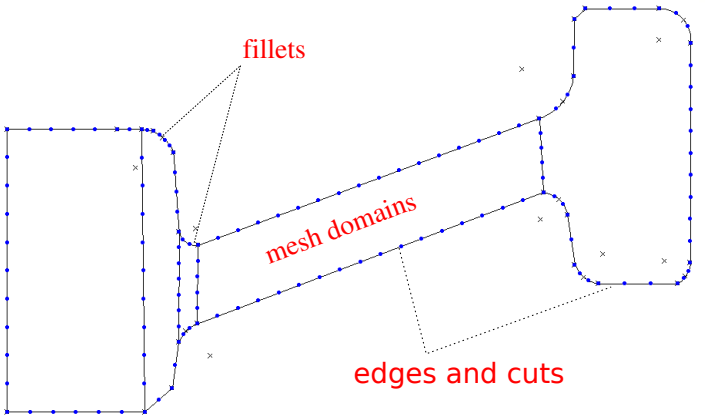


- Import from external software : Abaqus, Ansys, gmsh, LS-dyna, universal, FORGE...

Mesh generation with Zmaster

2D mesh generation steps:

- points, lines, arcs definitions.
- creation of mesh domains
  - . free/ruled
  - . linear/quadratic
  - . reduced/full integration

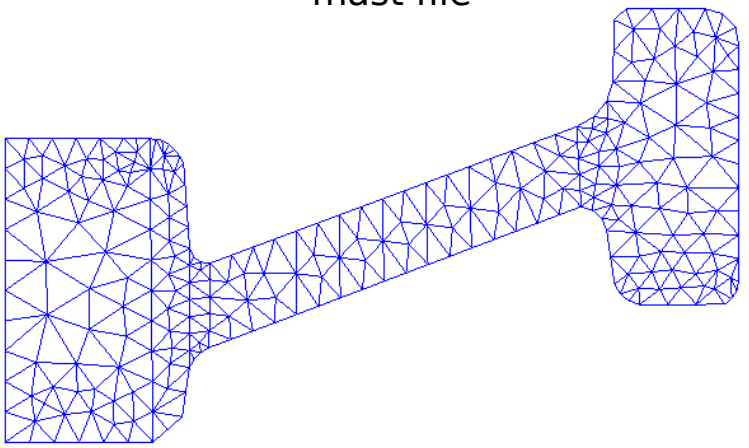


mast file

Z-commands

> open mast file ("Zmaster XYZ.mast")

> open geof file ("Zmaster XYZ.geof")



geof file

Extra commands

> launch terminal

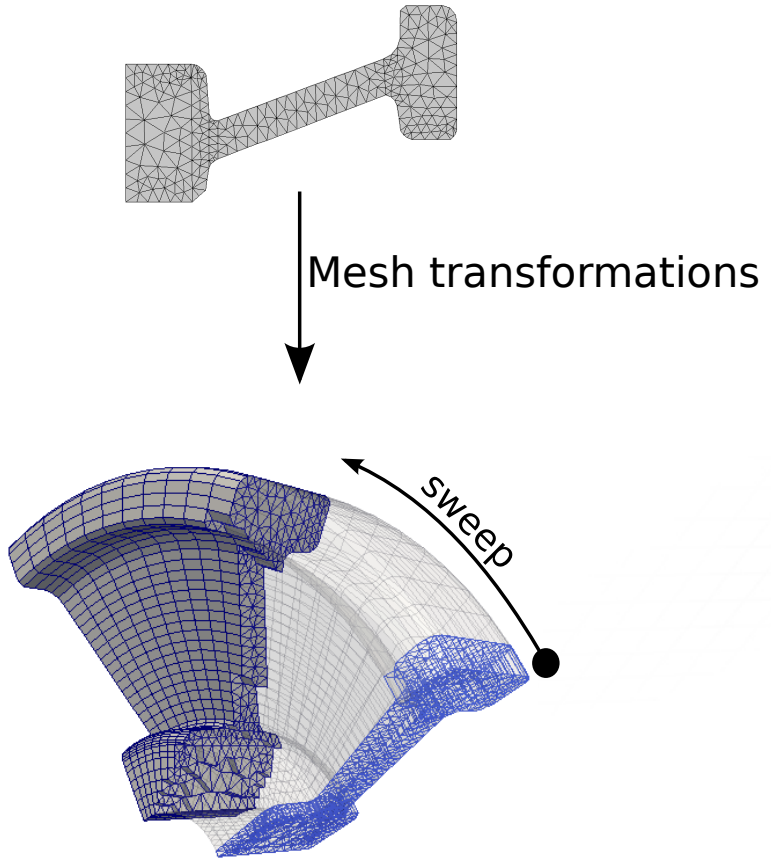
> clean folder

> reset project

Mesh transformations with Zmaster

Example 3D mesh transformations:

- sweep, rotate
- user defined mesher functions on existing mesh entities (node sets (nset), boundary sets (bset))
- linear ↔ quadratic mesh



Z-commands

> launch terminal

> check mesher file

> 2D→3D transform ("Zrun XYZ.inp")

> view with Zmaster ("Zmaster XYZ.geof")

3D re-meshing

MeshGems tools for re-meshing

- Hexahedral→Tetrahedral mesh
- allow local mesh refinement
- preserve nsets, fasets.

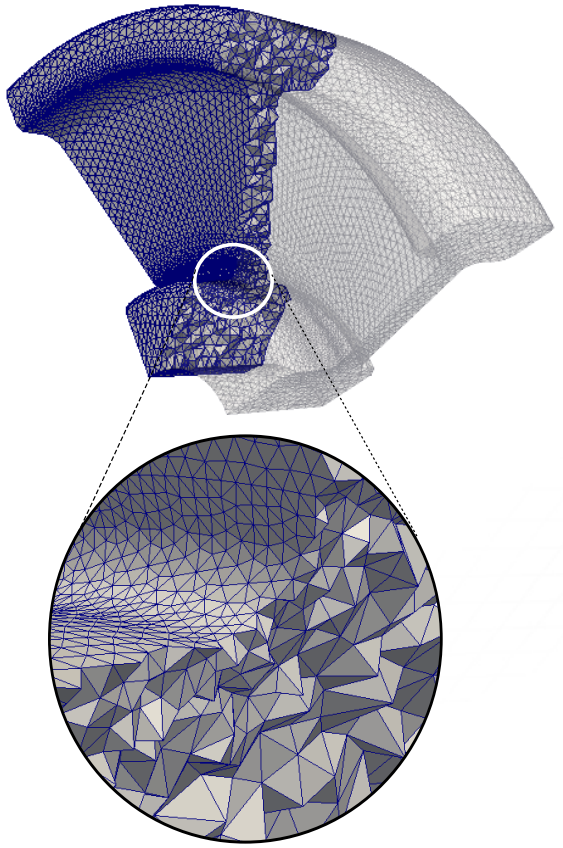
Z-commands

> launch terminal

> check mesher file

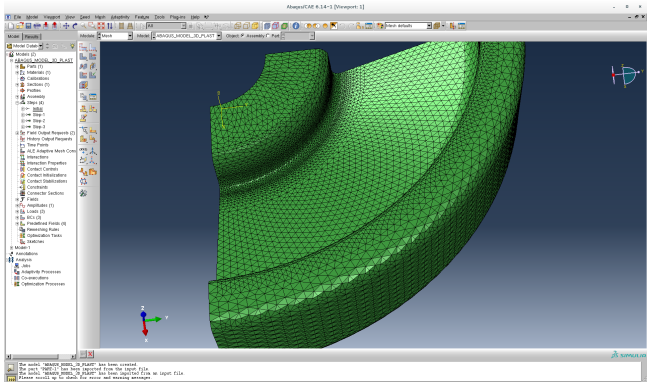
> run Re-meshing ("Zmaster XYZ.inp")

> view transformed mesh with Zmaster ("Zmaster XYZ.geof")

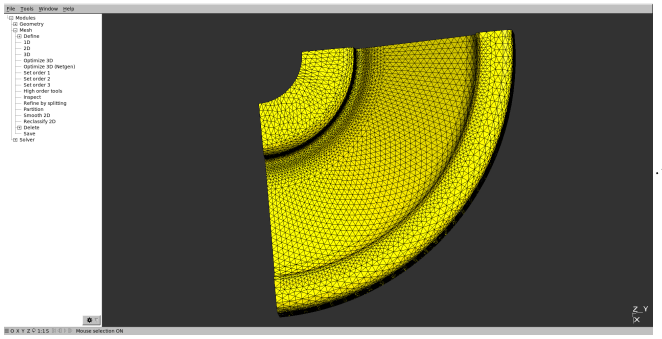


local mesh refinement

Import mesh

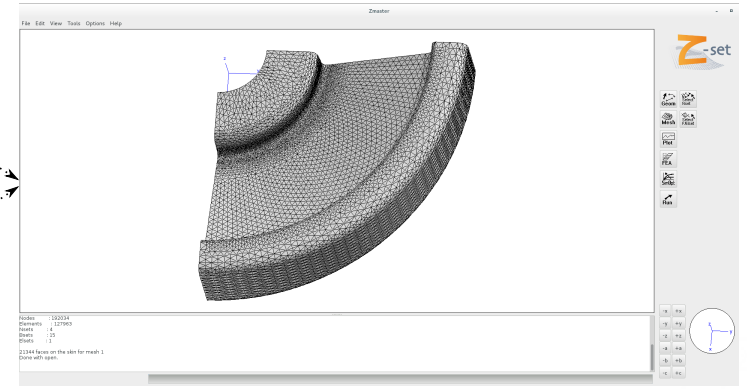


Abaqus



Gmsh

Import into Z-set



Zmaster

Z-commands

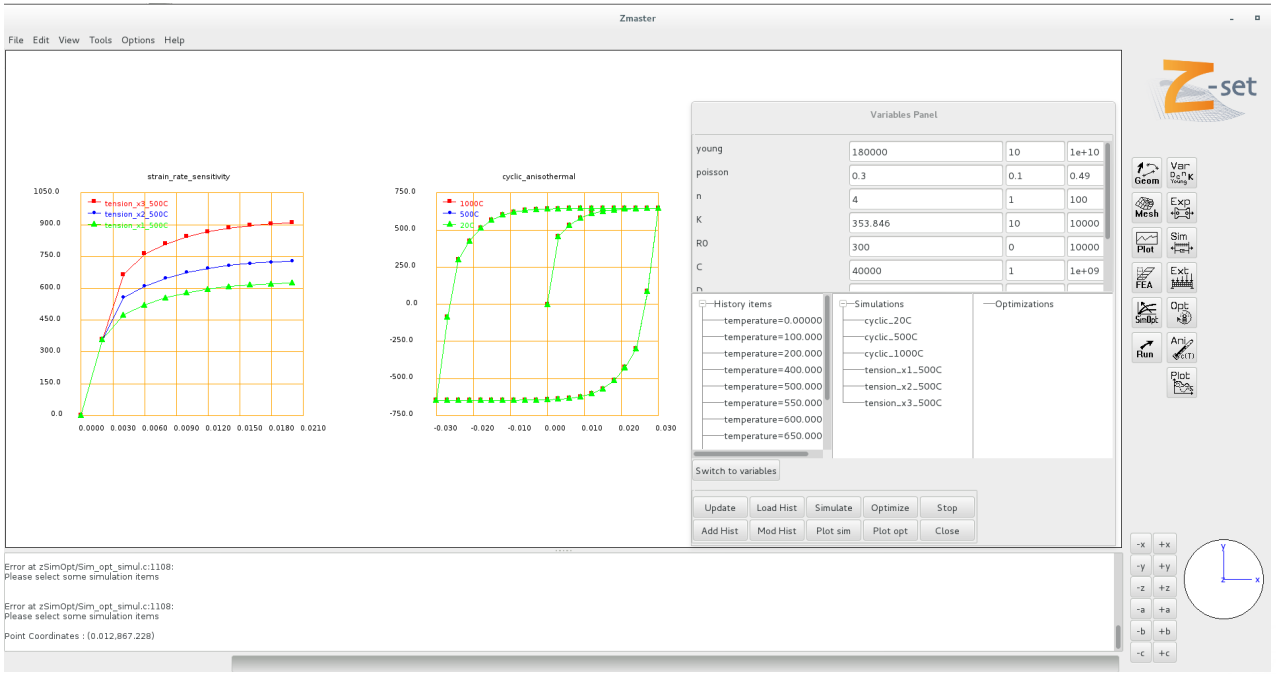
- check mesher file
- run Import operation
- view imported mesh with Zmaster

Extra commands

- launch terminal
- clean folder
- reset project

Material definitions & calibration

Z-SimOpt : Integrated Z-Sim and Z-Opt tools for an interactive capability to simulate and calibrate material parameters.



Example Nickel alloy material behavior simulation

- visco-plasticity (cyclic an-isothermal)
- strain rate sensitivity analysis

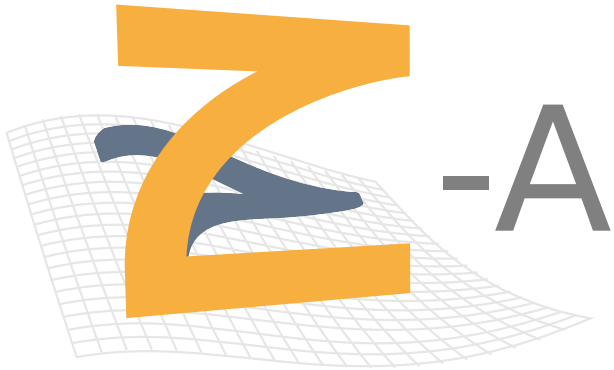
Z-commands

> check material template

> Open ZsimOpt GUI ("Zsopt XYZ.mast")

> launch terminal

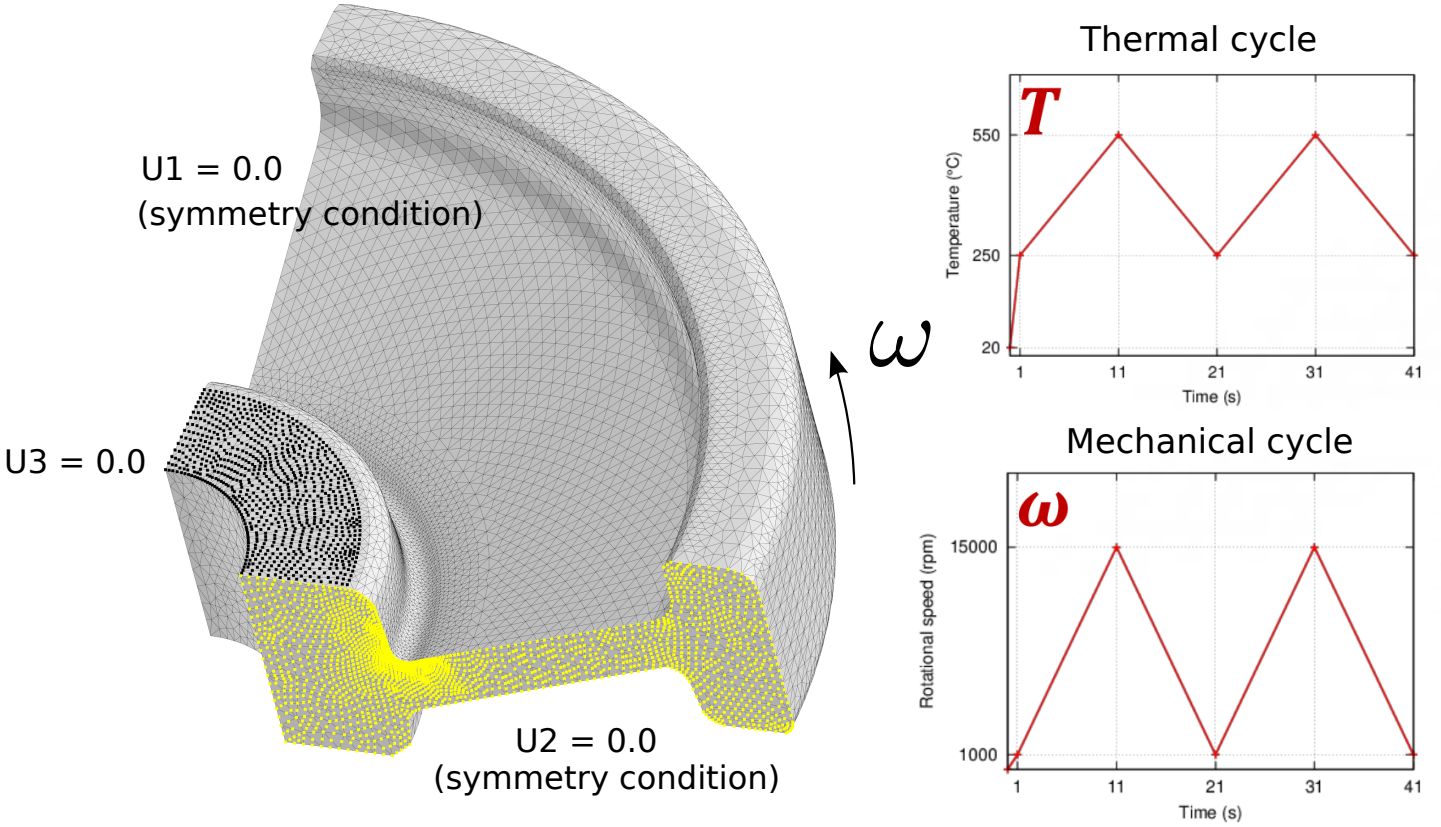
> reset project



-Analysis

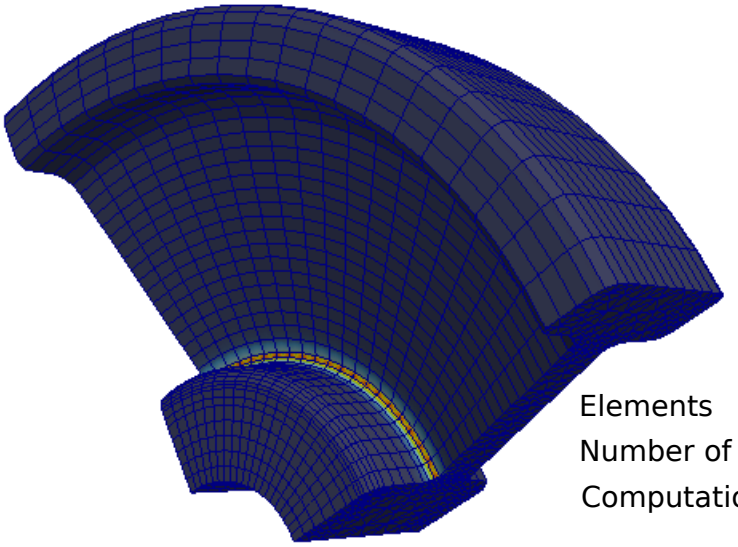
Loading and Boundary conditions

In the current problem setting, we apply a thermo-mechanical loading.



FE analysis

- Disc bending under the action of the centrifugal force
- Bending stresses are maximum on the surface
- Stress concentration on the fillet



Elements : 8860
Number of dofs : 79740
Computation time : 441.2 sec

Z-commands

> check inp file

> run simulation

> view results (Zmaster)

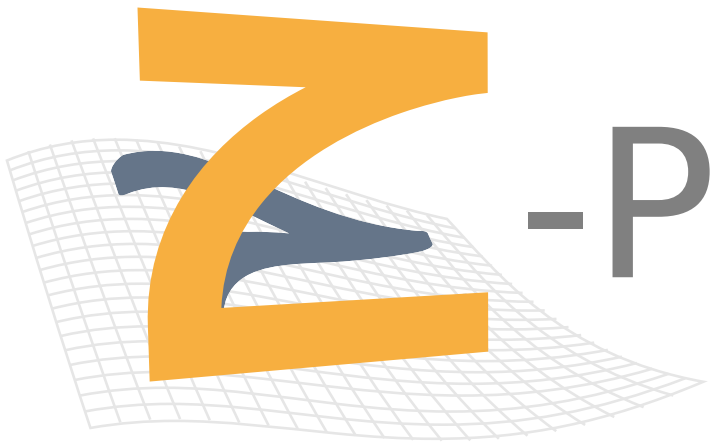
Extra commands

> launch terminal

> clean folder

> reset project

Note: To run a multithread simulation the command is 'Zrun -smp 4 XYZ.inp'



-Post

Post processing

Elementary post computations on standard FE results (displacements, stresses).

For example:

- extrapolation of integration point quantities to nodes.
- expression : apply simple math operations on results.
- eigen2 : compute the eigen values of a symmetric second order tensor.
- trace : calculate the trace of a second order tensor.

Z-commands

> check post input file

> run general post-processing

> view post-processing results (Zmaster)

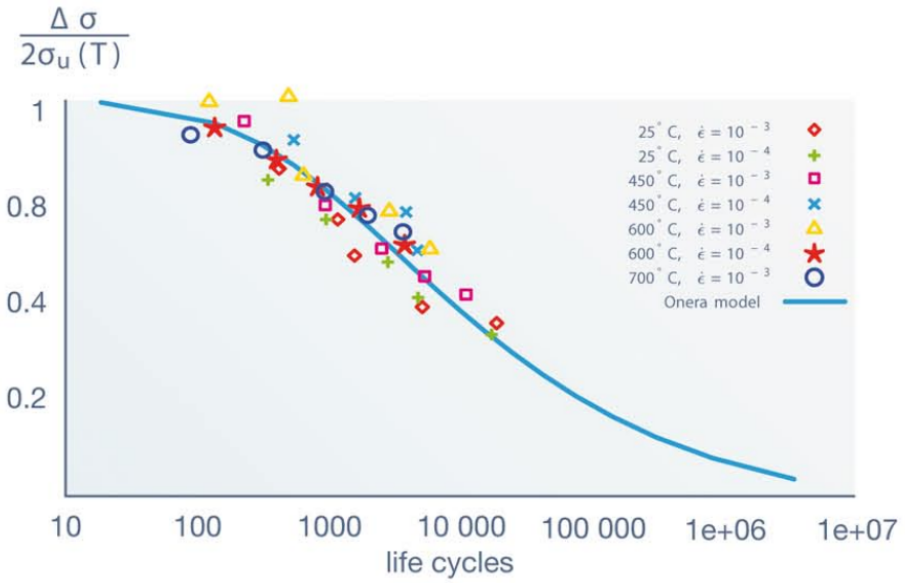
Extra commands

> launch terminal

> clean folder

> reset project

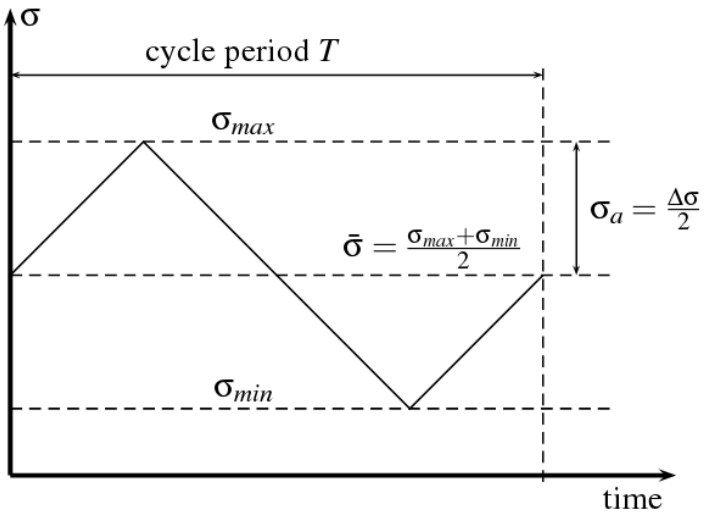
Fatigue life time estimation model



Main capabilities:

- valid in both LCF and HCF regimes (i.e., the entire S-N curve)
- multiaxial capabilities
- modeling of the mean stress influence (Haigh diagram, loading factor influence on S-N curve)
- possible coupling with creep damage to account for frequency effects
- possible nonlinear accumulation of damage for the correct representation of experimentally observed effects.

Fatigue life time estimation model

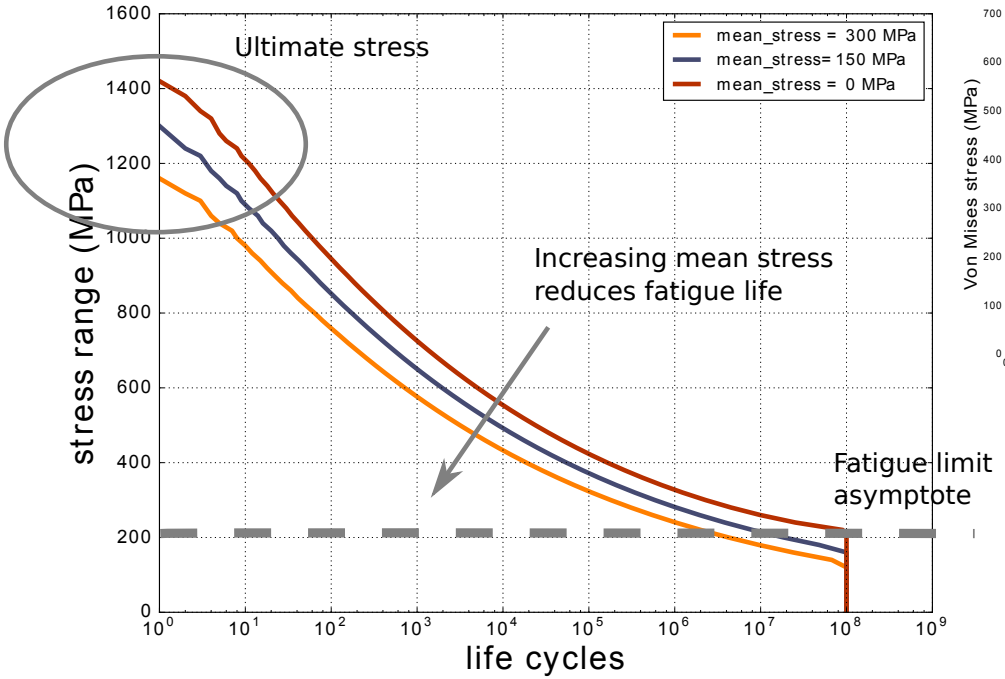


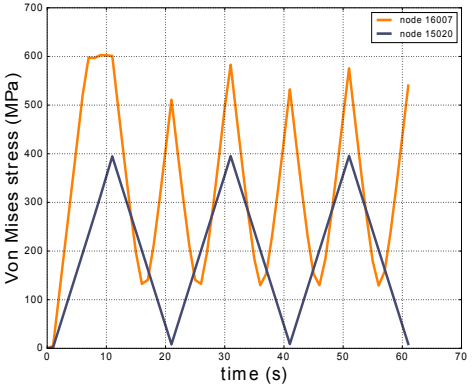
Chaboche fatigue model (S-N curve) : $N_f = \dfrac{< \sigma_u - \sigma_{max} >}{a(\beta+1) < \sigma_a - \sigma_l(\bar{\sigma}) >} \left[\dfrac{\sigma_a}{M(\bar{\sigma})}\right]^{-\beta}$

Usage:

- Input from cyclic FE computation: stress history on a cycle (stabilized cycle) at each point of the structure (IP or node)
- Evaluate stress range σ_a, mean stress $\bar{\sigma}$ and maximum stress σ_{max} on a cycle
- Compute life time N_f
- User parameters : a, β, M, σ_l, σ_u
- External parameters: temperature or any other field parameter

Fatigue model calibration





- fitting of the fatigue model using ZsimOpt GUI
- fatigue_s : using the stress as critical value, the fatigue life can be predicted.

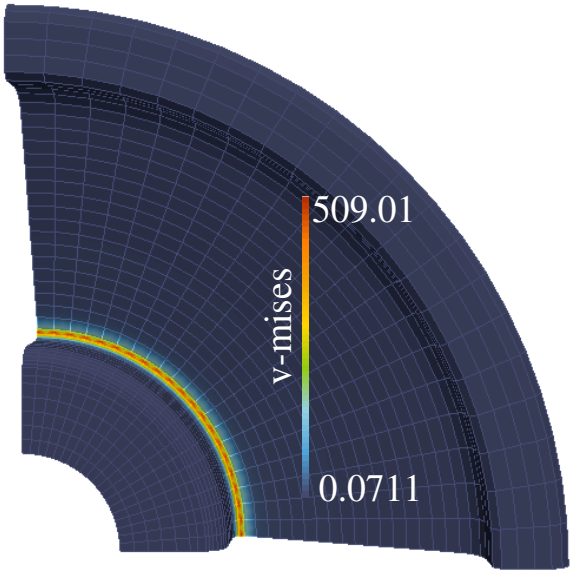
Z-commands

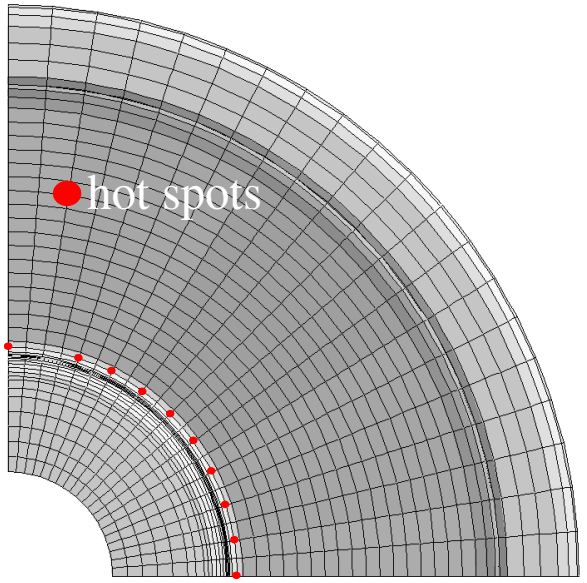
> open ZsimOpt GUI and calibrate the fatigue model

> launch terminal

> reset project

Post processing...hot spots





- fatigue analysis : calculate fatigue life at all nodes
- hot_spot : find locations which are local maxima or minima of a given variable (eg. number of cycles to failure).

Z-commands

> check inp file

> find hot spots

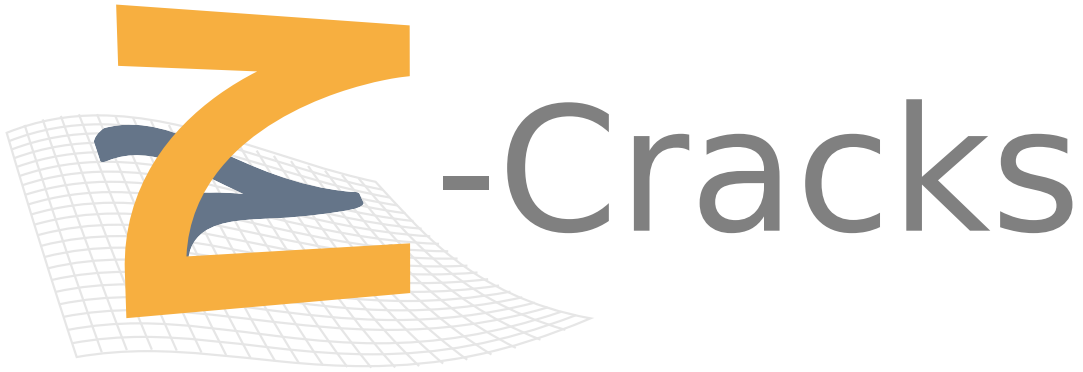
> visualize with Zmaster

Extra commands

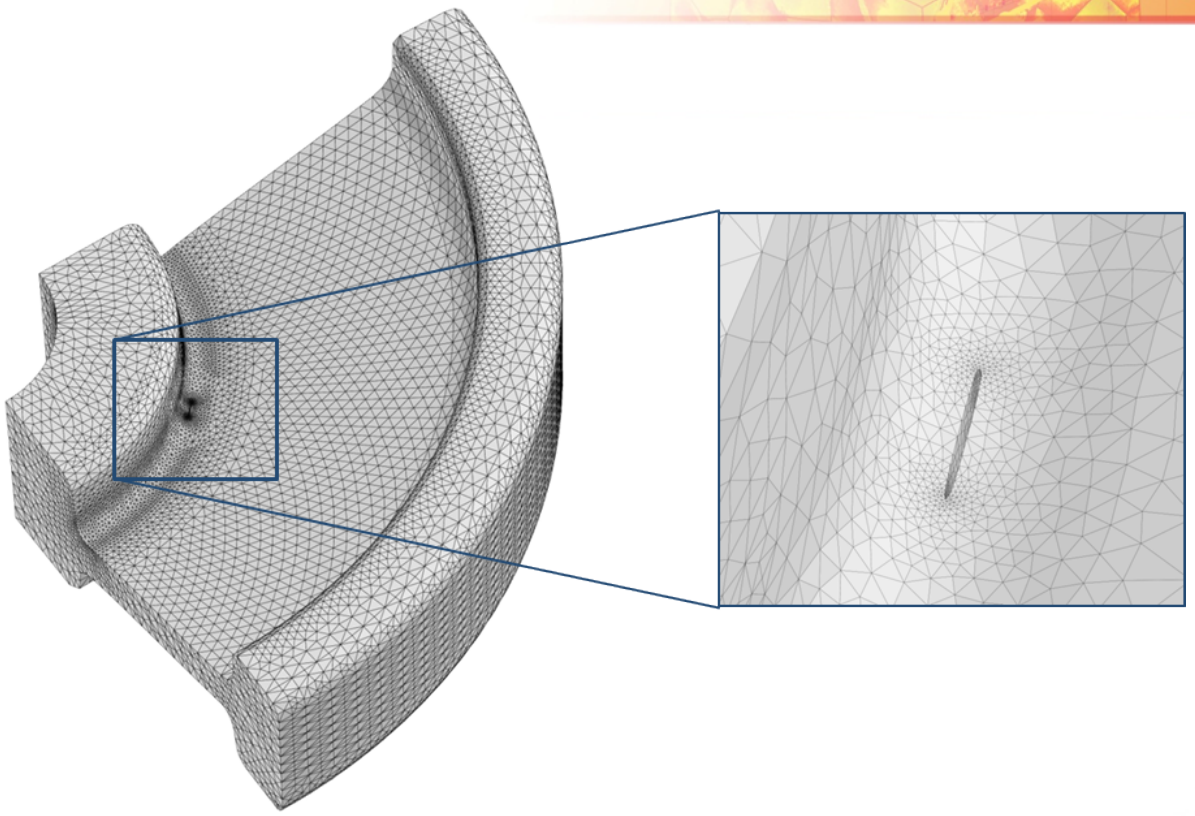
> launch terminal

> clean folder

> reset project


-Cracks

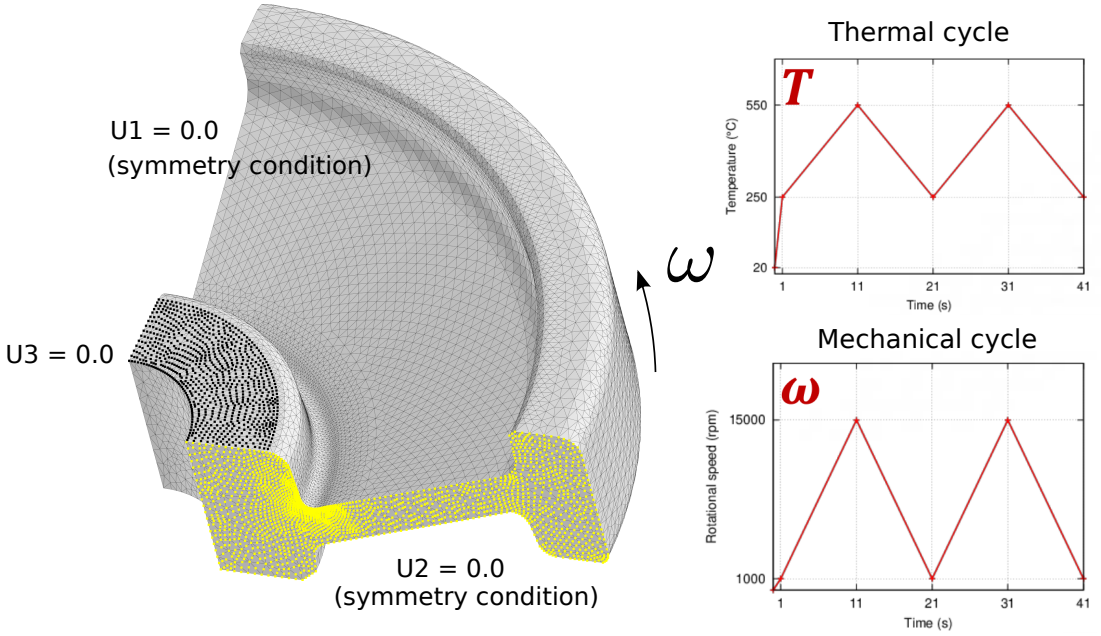
Introduction



- insertion of an arbitrary number of cracks in unstructured FE mesh
- local (in vicinity of the crack tip) or global remeshing
- FE simulation of the cracked component
- static crack SIF computation (G-theta method)
- crack propagation simulation (cyclic loading, plasticity, propagation laws)

Loading and Boundary conditions

- In the following, a linear fracture mechanics case is considered, as an exemple.
- All loadings (thermal fields and mechanical BCs) are kept similar to those used in the previous FE analysis.
- Finite element method is used to study stress/strain distribution in the cracked component, G-theta method is then applied to calculate SIF factors along the crack front.



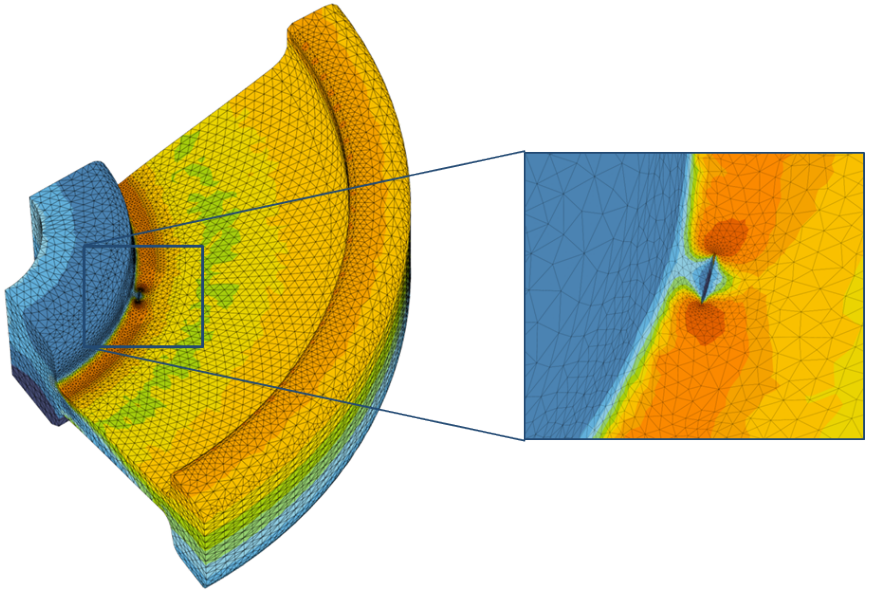
Commands:

> launch terminal

> check initial FE input file

SIF analysis and crack propagation simulation

- Use Z-cracks GUI to perform SIF analysis and set-up a crack propagation simulation



Z-commands

- check Z-cracks data file
- run Z-cracks GUI

Extra commands

- launch terminal
- clean folder
- reset project

- Expected results :