

Multibody
Simulation Software

SIMPACK User Meeting 2007

Graphical Representation of Stresses in SIMPACK's Postprocessor

Agenda

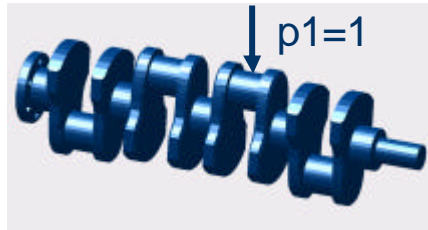
- redesign of FE-interfaces (stress calculation, import of flexible bodies)
- **stress calculation in SIMPACK**
- **modal stress recovery**
- **steps of the process**
- **export to durability software**

- import of flexible bodies into SIMPACK



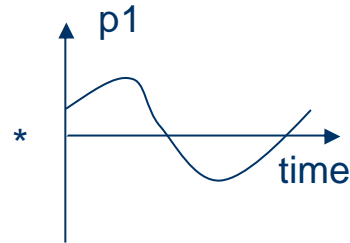
Modal Stress Recovery

forces



(unit load case)

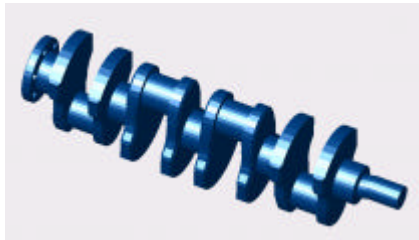
→ stress(p1=1)



= stress of load 1

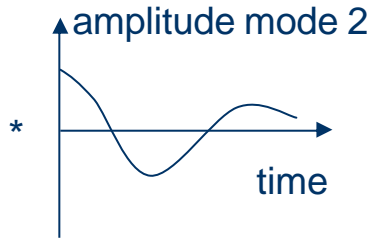
+
⋮

free oscillations



(mode 2 - lateral bending)

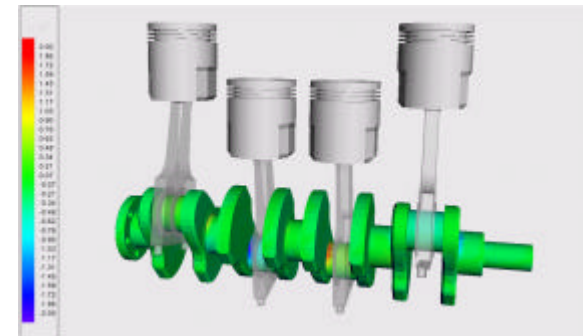
→ stress(mode2=1)



= stress of mode 2

+
⋮

superposition in SIMPACK postprocessor





Modal Stress Recovery

1) FE Analyses for unit load cases and normal modes

- > FE - input decks (written by measurement configuration)
- > FE result files

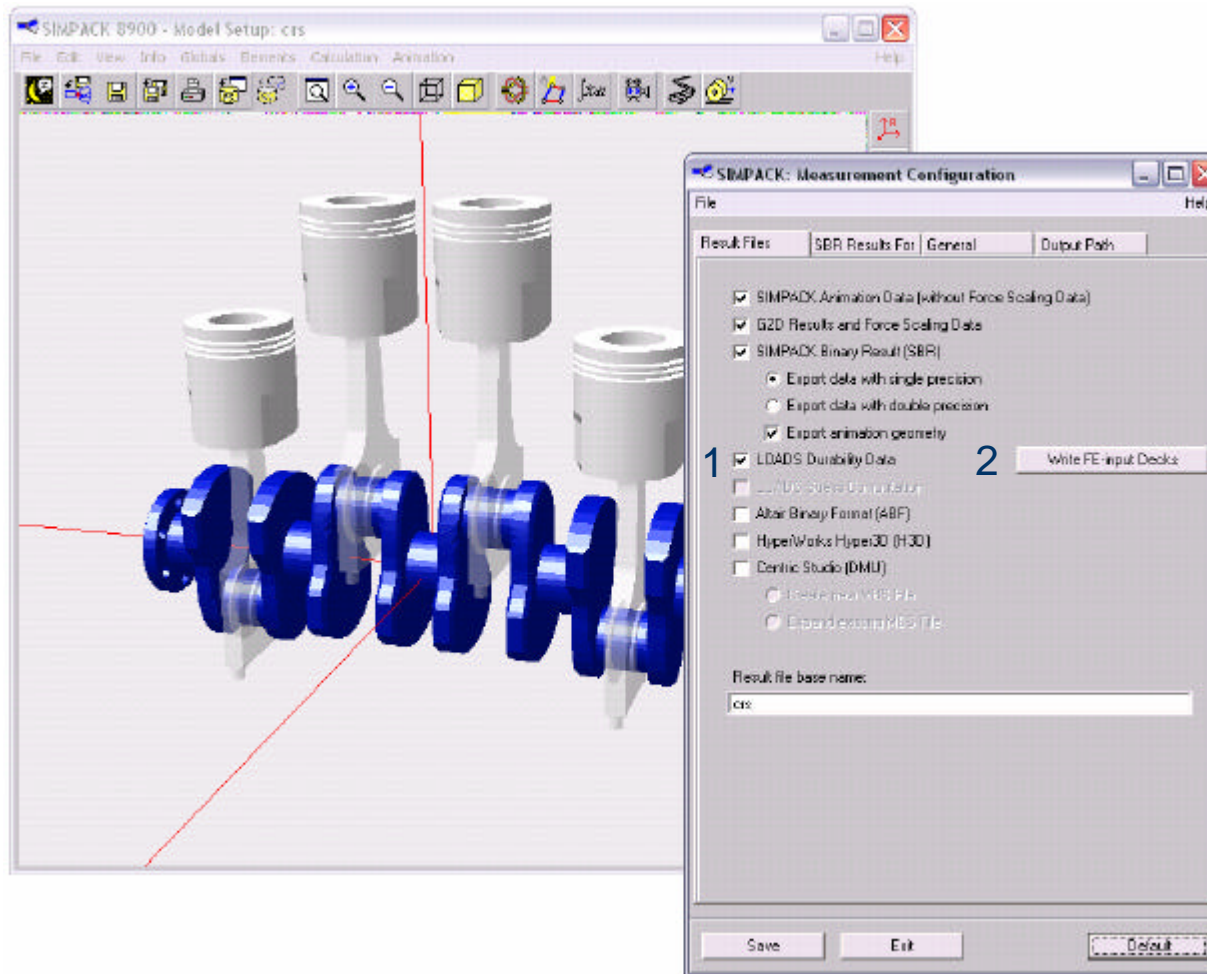
2) time integration and measurements to obtain the forces and modal amplitudes

- > additional sbr file

3) SIMPACK post-processing

- > superposition and graphical representation

Steps of the Process – Calculation of Stresses

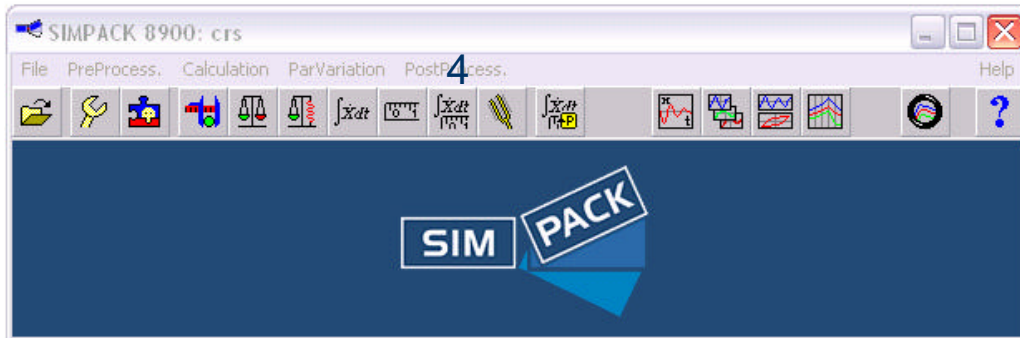


1. request forces and modal coordinates over time
2. write unit forces and mode selection as input deck
3. start FE-solver to obtain the stresses for forces and normal modes



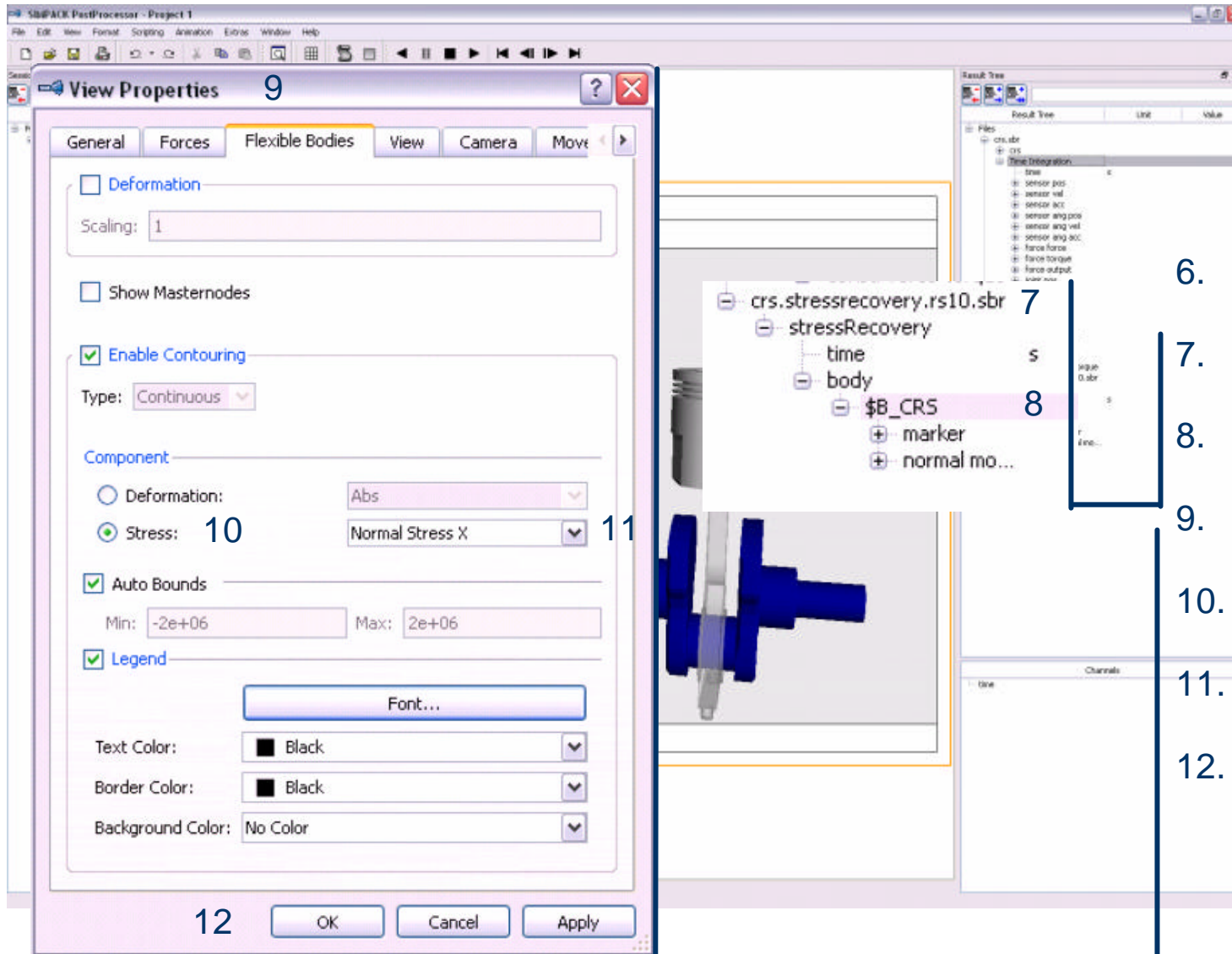
Graphical Representation of Stresses in SIMPACK's Post Processor

Steps of the Process – Calculation of the Time Dependent Data



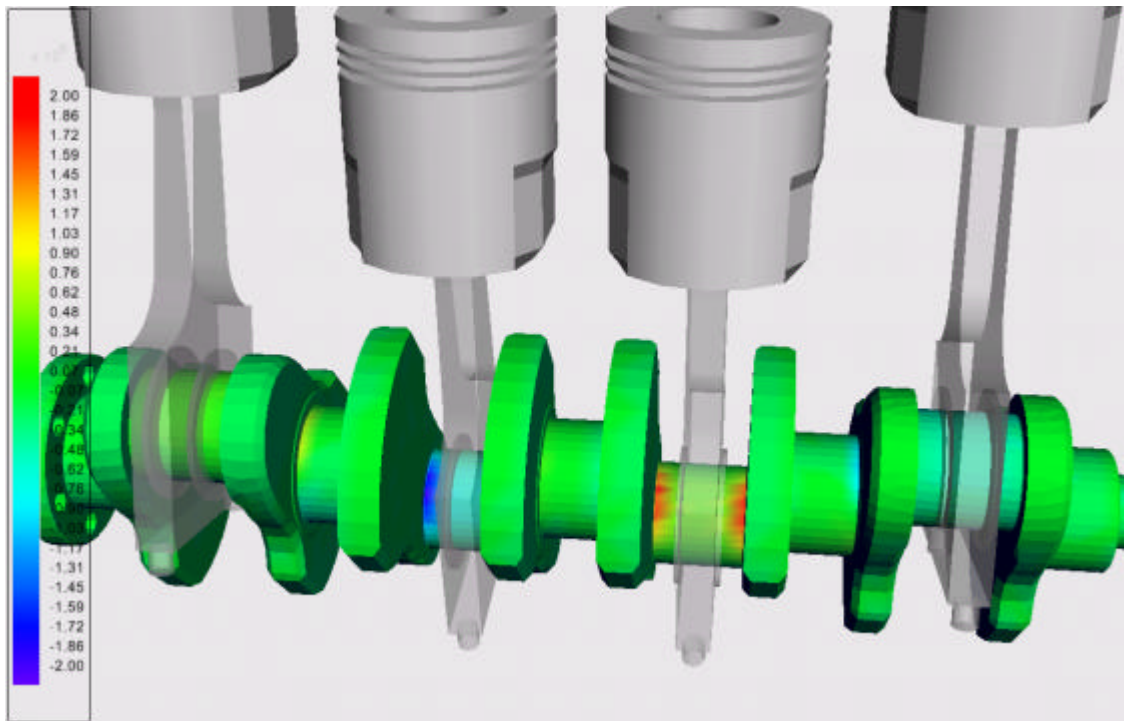
4. start the time integration and measurements
5. obtain the time dependent data as sbr file (new result type: modal recovery)

Steps of the Process – Post processing Superposition



6. drag the model into the animation page
7. load the stress recovery sbr file
8. drag the body into the animation page
9. edit the properties of the animation page
10. select stress contouring
11. select stress component
12. load FE-Results

Steps of the Process – Export to FEMFAT



13. export FEMFAT input decks

14. durability preview?



Graphical Representation of Stresses in SIMPACK's Post Processor

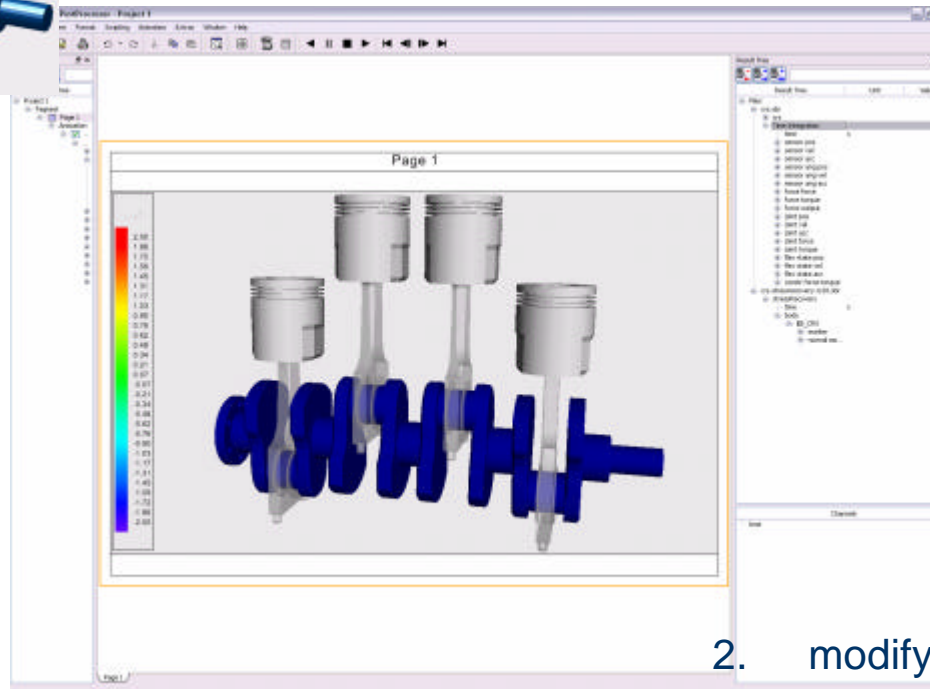
Release Version 8900

- stepped contouring
- editable colour bar
- 2D Plots for selected nodes (strain gauge)
- available in Version 8900 for NASTRAN, ANSYS and ABAQUS
- licensing: one license for all FE codes, separate licenses for export modules

Redesigned Import of FE-models



1. import the FE-model (mesh) into SIMPACK's pre processor



2. modify the FE-model
3. add markers (RBE)
4. define solver deck for reduction
5. start reduction
6. load reduced model into model set-up
7. select modes in the pre processor