

# Modal Stress Recovery in SIMPACK's PostProcessor

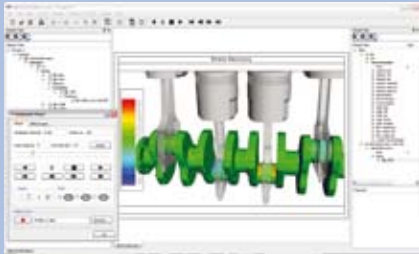


Fig. 1: Stress Contouring of a SIMPACK Crankshaft Model



Fig. 2: Export to FEMFAT from the PostProcessor

The new SIMPACK version 8.9 includes the first version of modal stress recovery. SIMPACK Stress is available in SIMPACK's PostProcessor. The stresses will be represented as contour plots vs. time.

This will help the user to:

- assess the mechanical strength of a component,
- detect the weak spots of a component under many different realistic loading conditions and
- obtain hints to improve the design.

In addition, these assessments can be achieved without transferring the results back to a finite element solver or to a finite element PostProcessor, for example by identifying critical loads or critically stressed spots in SIMPACK's PostProcessor, see Fig. 1 and Fig. 3.

When the critical loading conditions have been identified, the corresponding data may be exported to the durability software FEMFAT-MAX where the safety factors and the damage factors of the component are to be calculated. The export to FEMFAT is available either in the PostProcessor, see Fig. 2, or in the case of automated calculations as a command line call.

Since stress recovery in SIMPACK is based on a modal approach, the PostProcessor requires modal stresses that are to be calculated by a finite element code.

The input data for the calculation of the modal stresses is automatically generated in SIMPACK prior to the time integration in the measurement configuration. SIMPACK currently supports the finite element codes ABAQUS, ANSYS, NX NASTRAN and MSC.NASTRAN. When the calculations of the modal stresses are finished, SIMPACK's time integration and measurements should be started.

As a result the user obtains the time series of the scaling factors of the modal stresses. These are stored in an extra SIMPACK binary result file for stress recovery.

This file also includes references to the results of the finite element analyses. Before the stress recovery sbr-file is written, the normal modes used for the modal stress calculations are compared with the corresponding normal modes in the current SID file by the modal assurance criterion (MAC matrix). If necessary, the time series of the modal stresses are adjusted automatically. To see the contour plots of the stresses, load the stress recovery sbr-file into the PostProcessor, drag the body into the animation page and select stress contouring and the stress component to be displayed in the property dialog of the animation page.

The first part of the functionality includes the generation of data for stress recovery in combination with the connected FE-tools, stress contouring in the PostProcessor as well as 2D plots. The second part of the stress recovery connects the PostProcessor to durability tools such as FEMFAT-MAX.

Fig. 3: Customer Model, Komai Wind Turbine

