

# Loads Stress&Strain and Loads Durability

A complete redesign of Loads Stress&Strain and Loads Durability has been implemented in SIMPACK version 9.2. While retaining the reliability of stress recovery from SIMPACK Classic, the main focus for 9.2 was easier handling, higher accuracy and the ability to process even larger FE-models.

## PRODUCTS & LICENSING

The module Loads Stress&Strain enables the user to set-up and perform modal stress recovery in SIMPACK with the FE-codes: ABAQUS, ANSYS, MSC NASTRAN and NX NASTRAN. As with SIMPACK Classic, stress can be displayed in SIMPACK Post as contour plots, Fig. 1, and 2D plots. In version 9.2, the functionality of Loads has been extended to enable stress analysis of beam structures created with the Rotorblade and Dynamic Spring modules. The interface to FEMFAT-MAX enables the computation of the structure's safety-against-failure based on the SIMPACK results with the aforementioned FE-programs. The use of this interface requires the module Loads Durability.

mation and one for the stress recovery. The multi-body model itself still uses a modal approach with the usual smaller number of degrees of freedom to accurately consider the flexible body deformations in the multi-body system. In the postprocessing steps, the stress recovery process is based on the required larger number of modes in order to obtain accurate stresses. The corresponding set of mode shapes for stress recovery contains all inertia relief modes and the active eigenmodes of the flexible body. As a consequence, stress recovery has no influence on the calculation speed of the SIMPACK model and, vice versa, the small number of degrees of freedom of the SIMPACK model does not reduce the accuracy of the computed stresses.

## NEW IN RELEASE 9.2

In comparison with SIMPACK Classic, the steps of stress recovery have been significantly simplified. Stress recovery can be activated/set-up in the Body properties dialog, see Fig. 2. After specifying the FE-model that is to be used for stress recovery, the further user inputs, shown in Fig. 2, are defined by defaults that can be accepted for the majority of applications. Unit conversions and further transformations are automatically done by SIMPACK. Whilst SIMPACK performs the Time Integration and Measurements, input decks for FE-stress analyses can be exported. After the corresponding stress analyses in the FE-codes have been completed, the FE-output files are converted using either the Actions menu or from the command line. A so called stress file is generated, which has been optimized for fast reading access. The size is usually smaller than the FE-output files because the stress file only contains the averaged stresses at the nodes with re-

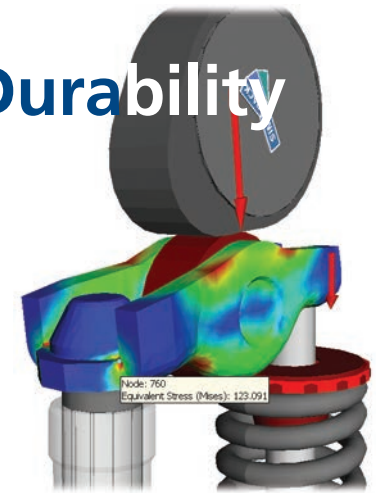


Fig. 1: Contour plot (equivalent stress Mises)

spect to the global reference system of the FE-model. For subsequent stress recovery, SIMPACK Post requires the sbr file, the mbf file and the stress file. For very detailed FE-models, the output of the FE-program can be limited to an element set, see Fig. 2. This reduces the amount of data to be processed during both the generation of the stress files and the display of the contour plots in the SIMPACK Post. Additionally, the properties dialog of the Flexible Body's Primitive in the SIMPACK Post enables the user to limit the amount of data to node sets that exist in the Flexible Body input file. With the stress exporter, 2D plots of the stress at selected nodes can be generated. The stress export is available via the GUI and scripting.

## SIMBEAM

For beam elements, stresses can be seen as the cross section forces and moments. These are the normal force due to the longitudinal deformation of the beam elements, the shear forces and bending moments due to bending, and the torsional torque. Basically, the process for SIMBEAM models is the same as described above for the FE-codes, apart from the computation/generation of the stress files and the displayed contour plots. The stress file is automatically updated when a SIMPACK solver is started, and the contour plots will always show element stress with respect to the element reference system, instead of averaged nodal stress. Enabling Loads in the Flexible Body properties dialogue activates the stress recovery for SIMBEAM models, see Fig. 3. The Loads default settings, see Fig. 3, can usually be accepted.

## INTERFACE TO FEMFAT

The input data to FEMFAT-MAX can be generated via the GUI and via scripting for the FE-Models. SIMPACK 9 supports all FEMFAT-MAX versions from 5.0 onwards.

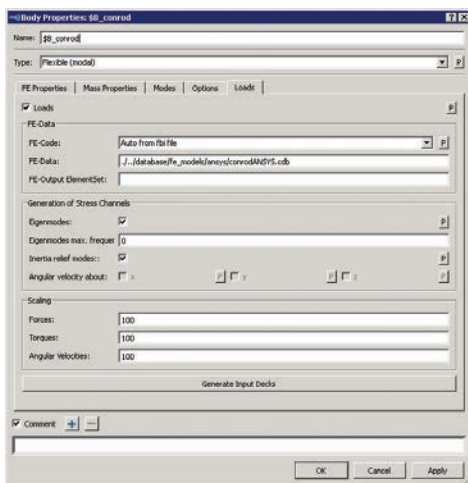


Fig. 2: Set-up of the stress recovery in the body properties dialog

## BASIC CONSIDERATIONS

An efficient modal stress recovery method is a requirement for computing stresses of a large number of load cases generated in a SIMPACK Measurements run. Generally, a larger number of mode shapes are required for an accurate stress recovery based on a modal approach than the number of mode shapes required for an accurate representation of the deformation in a multi-body model. SIMPACK uses two different model approaches: one for describing the defor-

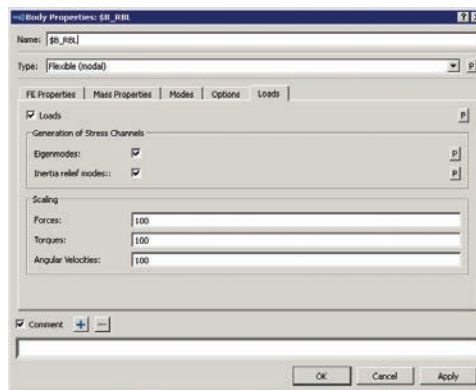


Fig. 3: Setting up the stress recovery for SIMBEAM models