*Roland Zettler, Thomas Ille*
*MAN Nutzfahrzeuge AG*

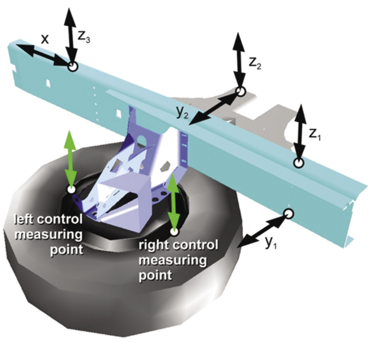
# Fatigue Analysis on a Virtual Test Rig using LOADS Durability

**In collaboration with MAN Nutzfahrzeuge AG, INTEC has developed the new SIMPACK-Interface LOADS Durability. The beta version was tested both within the scope of a diploma thesis as well as in a commercial application. It has also been successfully implemented in the simulation-based analysis of truck-frame components. LOADS Durability, in combination with the FEA-solver ANSYS and the fatigue-software FEMFAT, enables fatigue life predictions based on a MBS simulation.**

*Spare wheel mount on truck frame*

## MOTIVATION

The numerical computation of component stresses as a static load case is considered to be state-of-the-art industrial engineering practice. However, when quasi-static component stresses are calculated, the dynamic loading is not considered, important for fatigue analyses. INTEC, in developing the post-processor LOADS Durability, has managed to bridge this gap and provide an effective fatigue analysis tool.

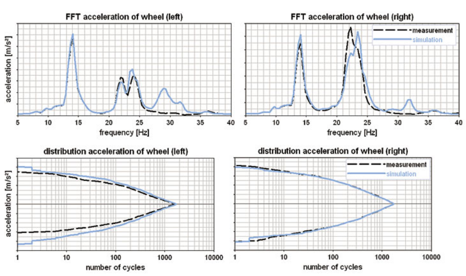



*SIMPACK MBS-model with sensors and excitation points*

## TEST RIG SIMULATION

The fundamental feature of LOADS Durability is that the fatigue analysis not only takes into account the component stresses as a result of internal forces, but also the local natural vibrations, a necessity within light structures (e.g. bus chassis or truck frame). To incorporate structural free vibrations, the use of a flexible body is a pre-condition in the SIMPACK model. However, stiff bodies do not require a flexible body representation

In the case under consideration, a test rig, investigating the frame add-on components, was simulated with particular attention paid to the fatigue life of the spare wheel carrier. A section of the test rig with the spare wheel carrier as the central component was modelled. The loads due to the accelerations, which were measured directly along with the integrated state variables, were applied to the multi-body-system and a time integration was performed.




*Correspondence of measured and simulated acceleration signal*

## POST-PROCESSING

The reaction forces at the mounting points of the flexible bodies are an important part of the component loading. The elastic behaviour of the body is described using the minimum number of modal co-ordinates; these are defined in the FEMBS-pre-processing. The LOADS Durability output file set contains the time dependent reaction forces and modal co-ordinates, the input files for calculating the unit stresses in the FEA-system and a FEMFAT input file for automatically assigning the time dependent output with the unit stresses.

## FATIGUE ANALYSIS

The fatigue analysis is carried out using the FEMFAT software. The LOADS Durability-output data allows a transient stress-distribution to be calculated, a precondition for a fatigue analysis. FEMFAT calculates the damage that occurs at each individual node, described by the FE elastic body. Damage computation takes place internally in the FEMFAT software by generating local fatigue limit curves. Additional material parameters such as the definition of welding seams, surface treatment and material type are also considered in FEMFAT, which offers a further improvement over a standard FE analysis.

## RESULTS

At the end of the process chain, a fatigue life value for every FEA-

*Roland Zettler, Thomas Ille*
*MAN Nutzfahrzeuge AG*

node of the considered component is predicted; these fatigue lives can then be visualised in the FEA-post-processor. Contour plots allow critical areas on the component to be identified and the damage to be described quantitatively. Extrapolated values of the fatigue life can be calculated by considering the collective load values and the time simulated.

A critical welding seam was identified on the component looked at, which correlated well with the damage location found on the measured physical component.

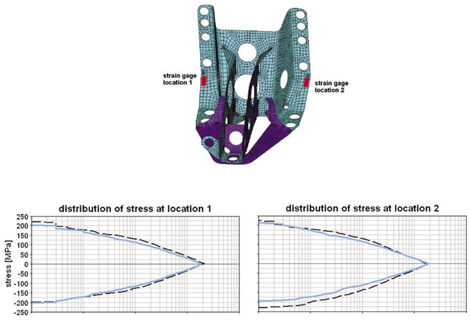
## ADVANTAGES OF THE PROCESS

The inclusion of a fatigue analysis in a MBS simulation offers new perspectives in the CAE and development process. Virtual prototypes can be tested on a SIMPACK test rig or in an entire vehicle model before going into production. This process will allow component development variation and optimisation considerations to be performed easily and quickly, with endurance predictions for each new variant simply produced. In the case of the spare wheel carrier, the improvement of the critical component could be forecast; this was achieved by the relocation of a welding seam and was verified by comparison with the physical test rig.

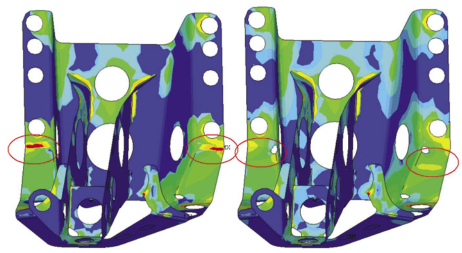
## THE FUTURE

This method saves both an enormous amount of computation time and also resources when compared with a transient FE-analysis. Although component stress calculation are what are being investigated, computation time is kept down due to the use of reaction forces and the few modal co-ordinates, mentioned above. After the first testing period, INTEC's development team has managed to cut down LOADS Durability computation times, so that large components can be calculated over even longer simulation periods. MAN Nutzfahrzeuge AG will apply this process chain in the future to optimise components with a dynamically heavy loading.

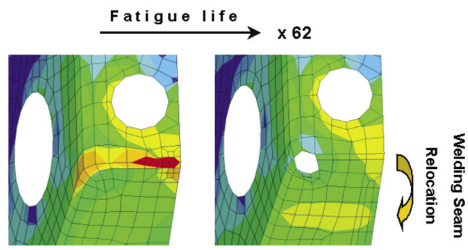
At present the analysis's focus clearly lies with the determination of the relative fatigue life. Achieving predictions for the absolute fatigue life will remain difficult, despite increased modelling quality and experience. Nevertheless, this is the goal which MAN Nutzfahrzeuge AG, with continuing new developments, hopes to achieve.




*Correspondence of measured and simulated stress*



*Damage contour plot of spare wheel carrier (old version left, improved version right)*




*Improved fatigue life as result of relocated weld*