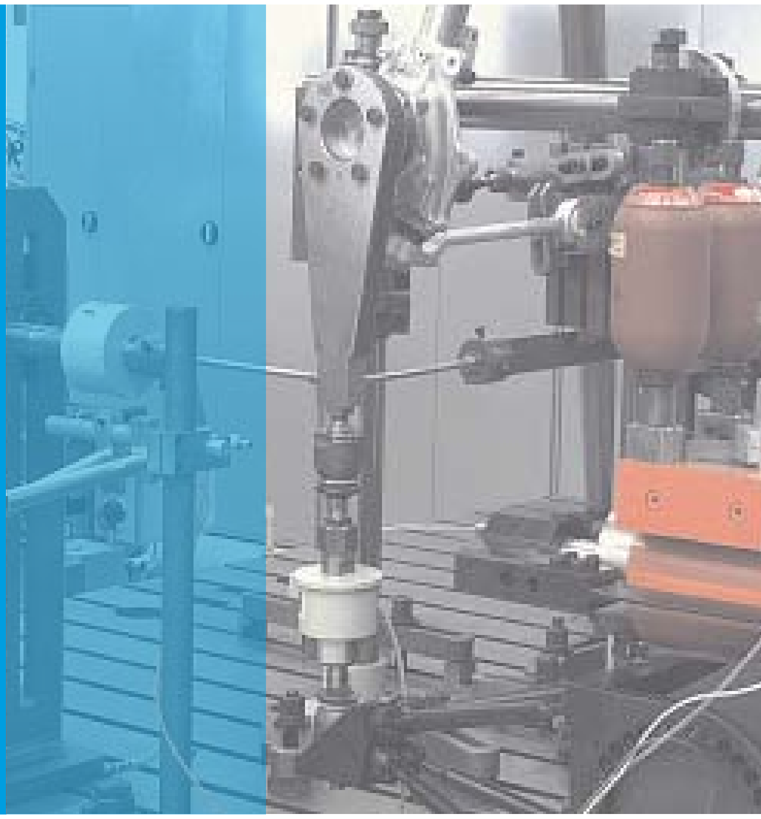
FEM based Durability Analysis of the Knuckle of the 5 Series BMW



LMS ENGINEERING INNOVATION

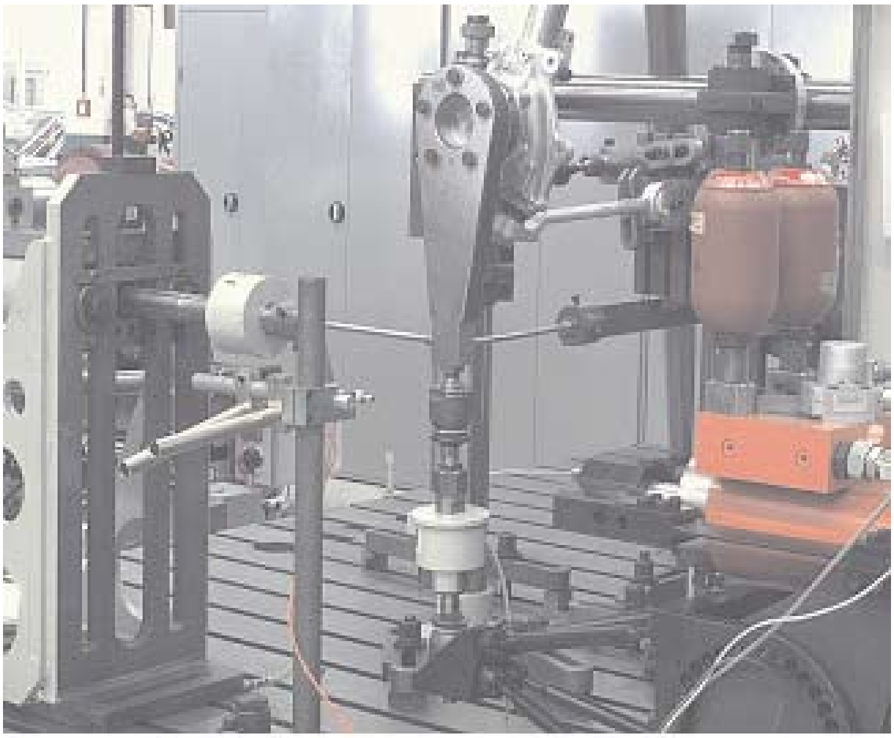


Fig 1: Test rig for the knuckle

Traditionally, durability of vehicle components and subsystems has been optimized and verified by testing physical prototypes. Today however, the objective is to validate the design as part of Computer Aided Engineering (CAE) and to physically test only validated designs in final durability proof tests for ultimate product release.

Objectives

In a joint project by BMW AG, P+Z Engineering GmbH and LMS numerical fatigue analyses were performed for the knuckle of the 5 series BMW and compared to physical fatigue tests. Since fatigue tests were started after finishing the numerical analyses, their results could not affect the predictions. The tool used for numerical analysis was LMS FALANCS, which accounts for external multiaxial loading and for local multiaxial stress states and can find critical locations (hot spots) of a component within reasonable time. As part of the cooperation, BMW performed the road load data acquisition and the fatigue tests, P+Z generated the finite element models and LMS performed the numerical fatigue analyses.

Component Tests

A knuckle connects the wheel of a car to the rest of the vehicle suspension such as the strut, tie rod and control arms, Fig. 2. It is loaded by longitudinal, lateral and vertical spindle forces, which change non-proportio-nally with time during different maneuvers. These forces were measured with a wheel force transducer on a test rig and are typical for a handling course, in which high lateral loads occur. The material used for the BMW 5 series knuckle is a cast AlSi alloy.

Analysis Methods

The numerical fatigue analysis is based on four basic steps:

1. Stress analysis of the component for static unit load cases based on finite element analysis. If natural frequencies are excited, modal stress shapes have to be determined.

2. Quasi-static superposition of the calculated elastic stresses in the time domain for all nodes of the FE mesh. To determine crack initiation locations in a component the time history of the local pseudo stress tensor $es_{ij}(t)$ has to be determined. For n load components acting on the structure this can be realized by quasi-static superposition:
$es_{ij}(t) = S_{m=1,n}\, c_{ij,m}\, L_m(t)$
Here, the stress influence coefficient $c_{ij,m}$ is the stress resulting from unit load case m, and $L_m(t)$ is the m-th internal force as a function of time.

3. Application of a local multiaxial stress-life or strain-life analysis based on these pseudo stress histories. Both approaches are used in this project and both work with the critical plane approach which takes into account that the direction of the maximum principle stress locally may change over time when multiaxial non-proportional loading is acting on a component

4. Determination of critical locations based on a reduction of the loading history and an elimination of less damaged FE nodes. The basic idea of this approach is to eliminate those finite element nodes early on in the processing, which are only slightly damaged, instead of analyzing each node with the full loading history. Thus, an automated, conservative approach to strongly speeding up the analysis is used, which is designed and proven not to miss any critical location and, at the critical locations, to produce exactly the same numerical result as without increased analysis speed.

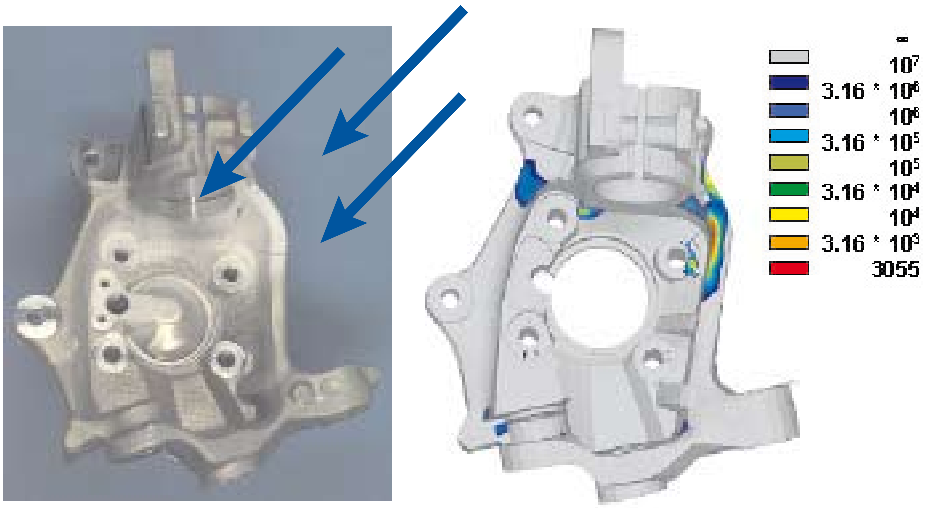


Fig 2: Comparison testing results via numerical results

Results

Fig. 2 shows that the predicted locations of crack initiation correspond to the experimental result. Fatigue life predicted for the most damaged location was 3055 repeats of the loading history. The mean of four physical tests gave a life of 2400 repeats. Such a predictive accuracy lies well inside of the scatter band of the tests.

The shown process of durability simulation enables car manufacturers such as BMW to analyze new designs as ‘virtual prototypes’ with the intent to perform tests on physical prototypes only after numerical design verification. ■

LMS INTERNATIONAL
Researchpark Z1, Interleuvenlaan 68
B-3001 Leuven [Belgium]
T +32 16 384 200 | F +32 16 384 350
info@lmsintl.com | www.lmsintl.com

Worldwide For the address of your local representative, please visit www.lmsintl.com/lmsworldwide

LMS is an engineering innovation partner for companies in the automotive, aerospace and other advanced manufacturing industries. LMS enables its customers to get better products faster to market, and to turn superior process efficiency to their strategic competitive advantage. LMS offers a unique combination of virtual simulation software, testing systems and engineering services.

LMS is focused on the mission critical performance attributes in key manufacturing industries, including structural integrity, system dynamics, handling, safety, reliability, comfort and sound quality. Through our technology, people and over 25 years of experience, LMS has become the partner of choice for most of the leading discrete manufacturing companies worldwide.

LMS is certified to ISO9001:2000 quality standards and operates through a network of subsidiaries and representatives in key locations around the world.

LMS®
ENGINEERING INNOVATION