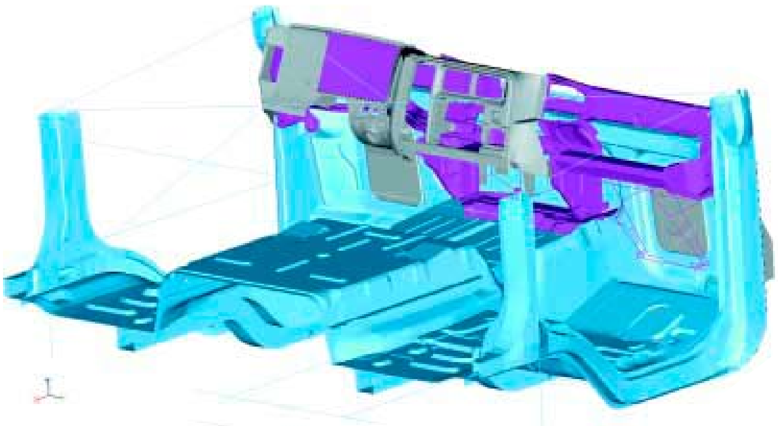
Improving Durability Engineering at Renault Vehicules Industriels

LMS
ENGINEERING INNOVATION

One of the exercises in the research project “FATINAMYCS” was to optimize the fatigue lifetime of a truck-cabin dashboard support. LMS’ partner in the project, Renault Vehicules Industriels (RVI), used LMS FALANCS to predict the fatigue lifetime using numerical techniques, validated the result experimentally using LMS Time Waveform Replication software.



Current design practice tends to rely upon purely test-based approaches, by which time design flexibility has been lost and the only resort is to use expensive palliative cures. By the use of a numerical approach, the design cycle could be dramatically accelerated, while also improving the overall design. The predictive approach implemented a numerical fatigue analysis, based on modal response, and was proven in its efficiency and accuracy in finding hotspots and comparing different designs.

In a first stage, a prototype of the original design was tested on a multiaxial rig and found for the specified fatigue requirements: a pattern of events (bumps and potholes) that need to be reproduced on the rig a given number of times. Failure of spot-welds was identified at two locations: at the connection between the dashboard support and central panel (first hot spot); and at the connection between the dashboard support and side panel (second hotspot). No failure occurred in the sheet metal itself. Following a modal analysis it became clear that the first mode, which is excited in the most severe way, has an opening and closing behavior between the dashboard support and central panel at the first critical spot weld. A re-enforcement was proposed to constrain the opening behavior of the two parts. This reduced the forces in the spot weld, and thus increased the fatigue life of that connection. Next, a second prototype was built and tested. Crack initiation was found at the side panel only towards the end of the test. This proved not to be critical, and thus the design could be signed off.

The second phase of the project was to reproduce the previous result by numerical prediction. First, a FE-model was created from which a dynamic and fatigue analysis was to be performed. As mentioned above, the structural modes are excited and thus modal superposition needed to be used to obtain the dynamic stress/force behavior.

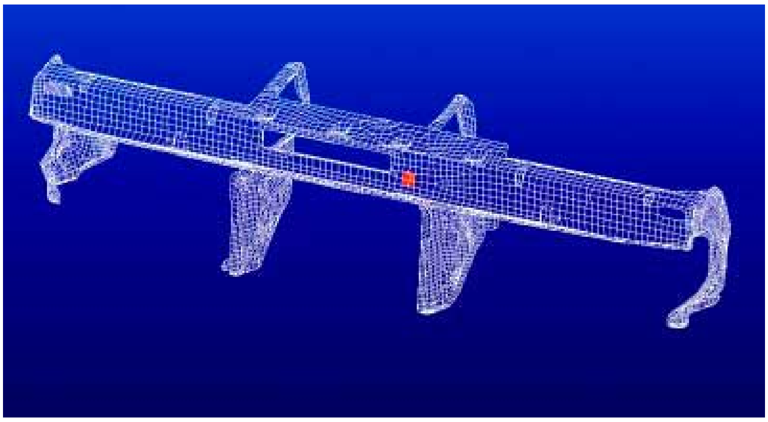
Specific to the simulation are the excitations being displacements, which requires a special attention when using modal superposition. In order to drastically reduce the calculation time, the Craig-Bampton reduction method was used. This means that the fixed interface normal modes and the constraint modes need to be calculated. Once this analysis is done, the time-histories are connected to the model and the contribution over time for each mode calculated. From the mode shapes (stresses/forces) and the modal contribution factors, the stress/force-histories can easily be calculated by means of linear superposition.

The Rupp model was used to perform the spot weld fatigue calculation. This requires that the spot welds be modeled as beams, such that the element forces correspond to the forces on the spot weld. These forces are then used to calculate the radial stress in the sheet metal around the spot weld, and the stress in the nugget itself. A ‘Critical Plane’ approach is used, which means that the stresses are calculated for 18 directions around the weld. Once these calculations are done, the damage calculation is performed using stress-life approach, as proposed by Rupp.

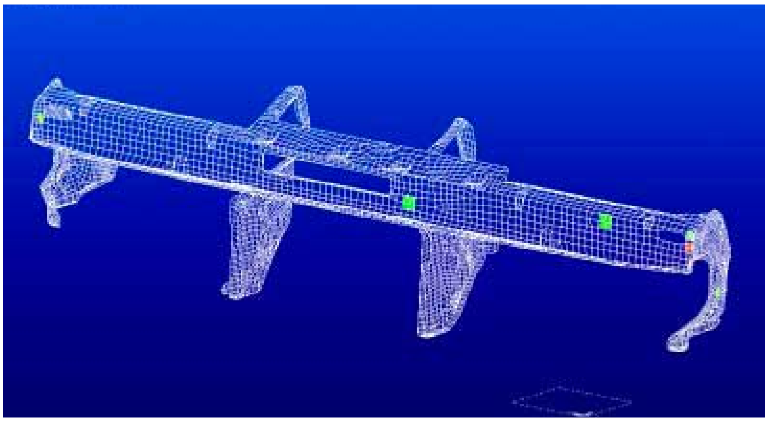
For the sheet metal, the strain-life approach was used. From the elastic tensor obtained by modal superposition, a projection to a potential critical plane is first made, then repeated for the 18 planes. Once the projection is done a transfer of elastic to elastic-plastic stress-strain behavior is performed using Neuber-rule (uniaxial plasticity model). The damage can then be calculated for each direction using the Smith-Watson-Topper and Miners rules. Doing so for each element would take a huge amount of calculation time, so to reduce this a special filtering technique was used. This filtering technique is based on rainflow projection and is applied to the contribution factors where it can be applied directly to the loading for a quasi-static approach.

Critical locations were found at the spot weld between the dashboard support and central panel (first hot spot) and at the spot weld between the dashboard support and side panel (second hotspot). These are the same locations in the same order as found by the rig-test. For the absolute lifetime there is a difference as to be expected due to the high variability on the experimental data from the Rupp-model. The prediction was on the conservative side. Successively a more detailed analysis on the forces in the spot weld was performed which showed a strong participation from the first mode. Closer investigation of the local deformation of the first mode showed an opening behavior between the dashboard support and the central panel. For the optimization, this opening aspect was reduced by reinforcement. Second fatigue analysis was performed on the modified model. This showed a critical location at the connection between the dashboard support and the side panel of the structure. Again, the same critical locations as during the rig-test were found.

It is clear that using an experimental approach and numerical approach for durability optimization leads to the same results. Specifically, in the case of mode excitation, the available technologies prove to be efficient and accurate in identifying the critical regions and the assessment of any proposed modifications. The advantage of using the numerical approach is that less time is needed, and fewer prototypes need to be built and tested. Overall this reduces the cost and increases the efficiency of the durability design process, and results in a better, more competitive design. ■



A preliminary analysis in LMS FALANCS showed potential problems at the spot weld between the dashboard support and central panel and at the spot weld between the dashboard support



After modification, the design lifetime had been improved to acceptable levels

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