Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 213 (2018) 539–548

Procedia Engineering

www.elsevier.com/locate/procedia

7th International Conference on Fatigue Design, Fatigue Design 2017, 29-30 November 2017, Senlis, France

# Enhanced fatigue structural stress analysis of a heavy vehicle seam welded steel chassis frame: FEA model preparation, weld model description, fatigue stress calculation and correlation with 10 year operating experience.

Jean Abry[a], Christophe Mittelhaeuser[a], Sébastien Wolf[a], Didier Turlier[a]*

[a] *LOHR Industrie, ZI de la Concorde, Duppigheim 67120, France*

**Abstract**

A FE seam weld model has been built for advanced fatigue structural stress analysis. The shell element model allows both weld toe and weld root fatigue assessment. In order to validate the model, this project deals with the complete analysis of a last decade manufactured vehicle for which experience feedback is possible. The vehicle structure is mainly built with steel sheets and the assembly is performed by seam welding. Shell element model is appropriate for FEA. The entire FEM preparation is presented hereafter and each step is detailed: 3D CAD weld building, mid-surface idealization, weld leg imprint creation, weld mesh connection and load application. Fatigue analysis is performed based on a fatigue limit approach and IIW FAT data. For every structure location a Haigh diagram is built. Different diagrams are presented for plain metal, cut edge, weld toe and weld root. The FEA results are post processed and each appropriate tensor component is taken into account. For plain metal, stress variation along the worst principal stress orientation is calculated. In welds, for both toe and root locations, the structural stress computation based on grid point forces extraction is described. Longitudinal shear stress in weld and at weld toe as well as weld longitudinal normal stress calculations are detailed. Fatigue results are displayed for each location over the whole chassis model. Finally, vehicles built during the last decade are inspected back from the customers experience field. Observed crack types and locations are compared to fatigue analysis predictions.


Peer-review under responsibility of the scientific committee of the 7th International Conference on Fatigue Design.

*Keywords:* Fatigue cracks, Fatigue strength, Stress analysis, Structural analysis, Finite element analysis, Fillet welds.

* Corresponding author. Tel.: +33 3 88 38 98 00 ; fax: +33 3 88 49 14 69 .
*E-mail address:* virtual.testing@lohr.fr

1877-7058 
Peer-review under responsibility of the scientific committee of the 7th International Conference on Fatigue Design.
10.1016/j.proeng.2018.02.050

## 1. Introduction

LOHR Industrie products car carrier vehicles, railway semi-trailer transport systems and urban electric people transportation solutions. All products are steel made and assembled by welding. For vehicle structure analysis, LOHR Industrie has developed FEA models for assemblies relative to seam weld and bolt [1]. Seam weld fatigue assessment model has been described in numerous publications [2, 3, 4] and validation based on simple assembly specimens has been detailed.

A project has been launched in order to verify the proposed FEA model and the associated fatigue behaviour prediction by comparison with long mileage vehicle structure inspections. An old generation car carrier trailer model is built. The fatigue analysis based on advanced structural stress approach is performed.

## 2. Trailer FEA model

The vehicle which has been proposed is the Eurolohr 100, shown in Fig. 1. Its concept was unique and revolutionary: a shorter body and a longer and lower trailer provide for optimal load capacity. The physical dissociation between tractor and body, leads to a great flexibility of use. The trailer, thanks to its maximum dimensions (12 m), provides a greater load volume than a trailer that has been traditionally configured (carrier + trailer) and the loading angles are more favourable.

Fig. 1. Eurolohr100 car carrier convoy.

The trailer FEM may be built based on detailed 3D CAD data. All seam welds have been modelled with the 3D construction tool embedded in the CAD solution. Then a company software called "seam sim suite" with several modules is used to generate the idealized sheet bodies before meshing.

In order to reduce preparation time and because of the vehicle design symmetry only one longitudinal half of the trailer structure is used, as shown in Fig.2. The global definition is performed in a part assembly context and a tool allows importing of all selected assembly parts in a single top part to reduce part file manipulations.

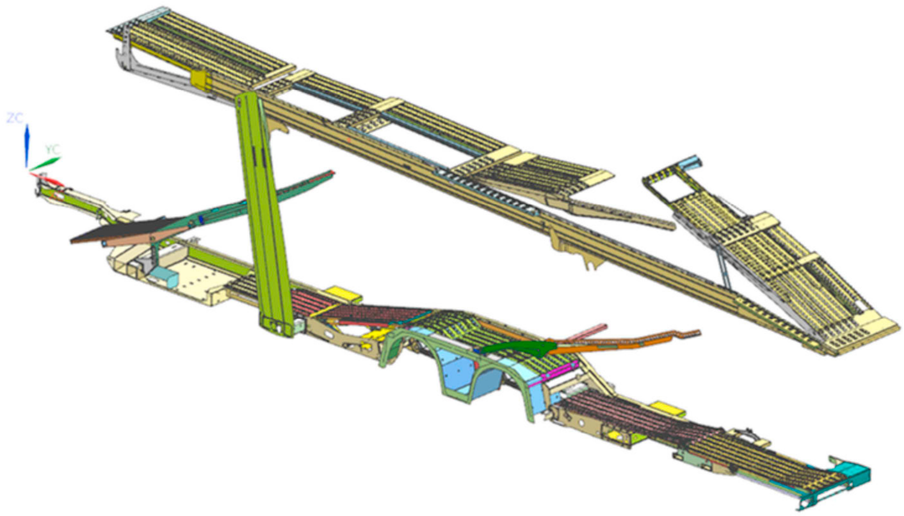


Fig. 2. 3D top part linked bodies half structure.

A first routine eases the transfer of all components geometry from assembly level to the top part level by creating linked bodies. If weld construction has been performed between assembly components then specific attributes are set on bodies and weld bodies that hold weld associativity with bodies using unique identifiers as shown in Fig.3.

When solid welds construction has been performed, the material characteristic is assigned to each solid body. The material xml library provides the material list; a local copy of the material characteristics which allows for each material collection to be individually parametrized with specific fatigue characteristics for instance. For welds a filler metal category has been introduced which will allow both weld zone recognition and specific weld fatigue limit assignment.

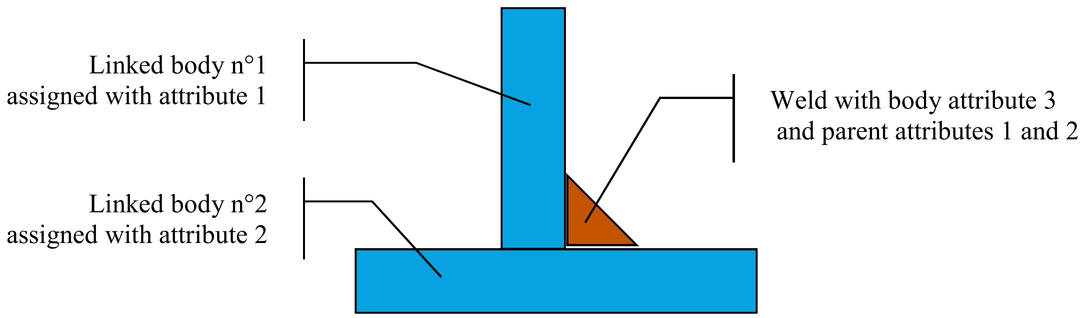



Fig. 3. Associativity attributes assignment

The structure is built using mainly metal sheets, their slenderness allowing shell meshing on surfaces. Each sheet body is transformed into a mid-height surface in order to preserve eccentricity at each weld assembly. A specific program is used in order to generate mid-surfaces automatically. All bodies that are fit for mid-surface degeneration are processed and initial body attributes are transferred. The others are processed through manual mid-surface generation. For welds, the idealized surface corresponds to the weld throat mid-height section. A routine is then used to generate weld contours that will be used to imprint the weld toe on the plate assembly as shown in Fig.4. Different configurations are taken into account: usually plate edge to plate face weld connection type for T joint fillet weld assemblies and plate surface edge over a plate face connection type for plate overlap welds assemblies. Intermittent welding may be used and weld extremities are also delimited in order to process weld toe at weld ends.

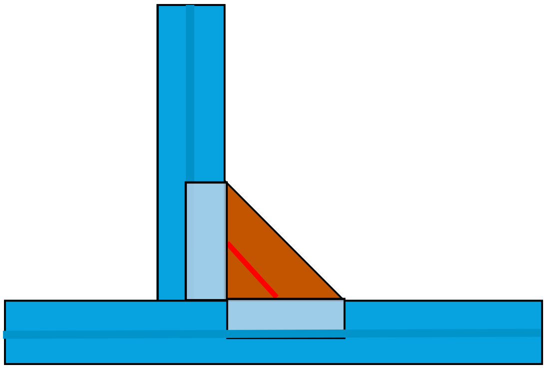


Fig. 4. Weld contour generation

Once the geometry idealization with weld contour imprint is performed, a specific tool is used to generate the structure meshes with the appropriate connections.

Each metal sheet midsurface and weld mid-height throat section surface is meshed using shell elements. The weld element thickness is the weld throat size, then the weld section area is fitted. The weld connections to sheet plates are made using 1D elements. Each node of the weld mid-height section edge is copied using projection on the associated sheet metal and a rigid 1D element connection is built. Then the projected node is connected to the plate face or the plate edge in the case of overlap joints using 1D multi point constraint elements to the associated plate element nodes.

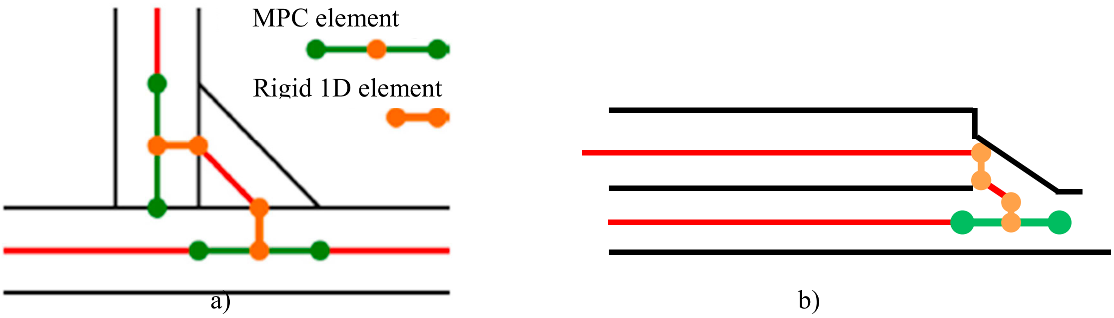



Fig. 5. Weld connectors a) T-joint b) overlap joint

Thanks to the material attributes the different assembly zones may be processed using adapted criteria. The assembly zone options are: plain metal areas, cut edges of metal sheets, weld toe contours and welds.

The entire trailer meshing is then obtained through symmetry transformation, as illustrated in Fig.6.

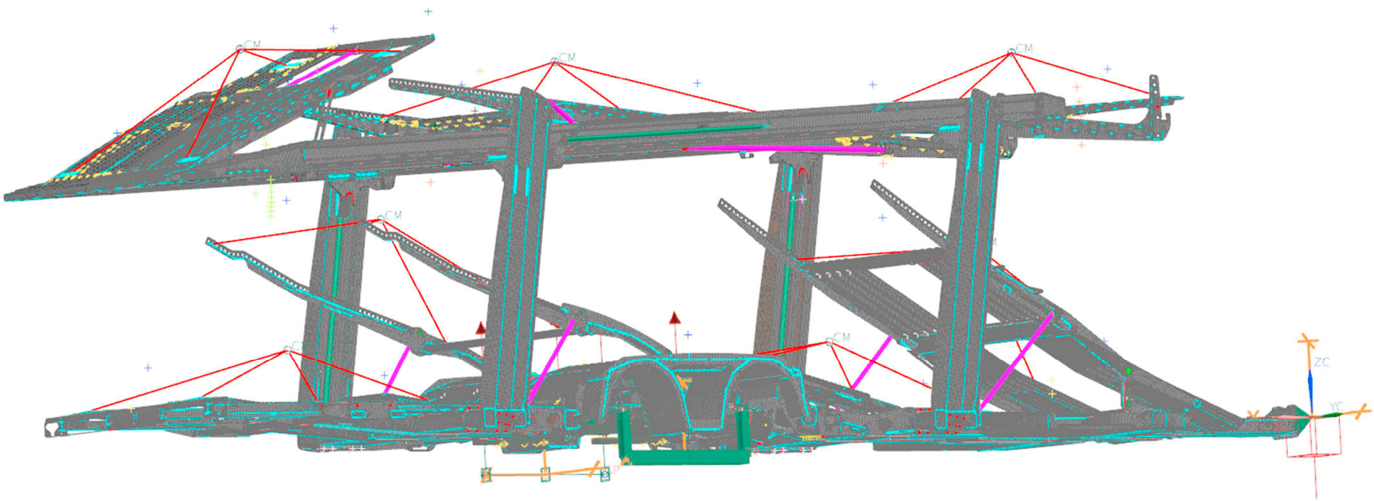


Fig. 6. Eurolohr 100 trailer structure FEA model

The boundary conditions are: fixed translation at the tractor coupling interface and at the wheel ground contact areas. The axle kinematic and suspension have been introduced using bush and beam elements.

Car loading is introduced using lumped masses connected to the trailer platforms at the wheel contact areas. Fatigue loads correspond to in service road conditions and the dynamic loads are applied as global accelerations with vertical, transverse and longitudinal variations.

## 3. Fatigue enhanced structural stress analysis

FEA solving is performed using linear static solution and fatigue analysis is performed using a specific post-treatment code: POSTAL, acronym for POst-processing STructural Assessment Laboratory. A fatigue endurance limit approach is used, Haigh diagrams are built for each zone type: plain metal, cut edge, weld toe and weld root.

Structural stress is computed at the weld toe using through-thickness linearization, as mentioned in [5]. The shell element normal stress distribution is linear through the plate thickness and it is the sum of a membrane stress and a bending stress. In order to reduce element size sensitivity and element shape quality dependency, the structural stress is not directly extracted from finite element solving but it is calculated from the element nodal forces and moments.

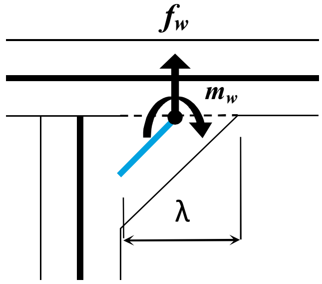
Grid point forces F or moments are summed at the element nodes along the weld toe contour and then distributed on adjacent element edges with regard to their respective lengths. The force and moment densities are computed using the inverse shape function matrix transformation as given below for a first order element and two element edge nodes 1 and 2, l being the element edge size.

$$\begin{bmatrix} f_1 \\ f_2 \end{bmatrix} = \frac{1}{l}\begin{bmatrix} 4 & -2 \\ -2 & 4 \end{bmatrix} \cdot \begin{bmatrix} F_1 \\ F_2 \end{bmatrix} \qquad (1)$$

Similar structural stress is calculated at the weld root through the weld leg section as defined in [6, 7] where weld solid meshing is used. Here because shell elements are used structural stress is calculated by nodal forces and moments extraction and is the summation at the weld root of the membrane stress (2) and the bending stress (3) as given in [4, 7].

$$\sigma_{m,w} = \frac{f_w}{\lambda} \quad (2)$$

$$\sigma_{b,w} = \frac{6.m_w}{\lambda^2} \quad (3)$$




For weld toe, the structural fatigue stress limit is based on FAT 90 for load carrying fillet welds and FAT 100 for non-load carrying fillet welds and butt welds: S/N curves are given in [5]. FAT values correspond to admissible stress ranges for 2 million cycles with a survival probability of 97.5%. For weld root, the FAT curve for stress range is FAT 80 [4, 6, 7].

To perform such calculations, weld element edge shall coincide with weld leg mid-height contour and a local coordinate system is used for correct force and moment component extraction.

For the vehicle structure fatigue assessment, the endurance limit approach is used. The structural stress is calculated using the worst stress range, which is the envelop of the stress variations from all the fatigue load cases.

The results are expressed as a percentage of the stress range in relation with the fatigue limit by comparing the value obtained to the maximum admissible value on Haigh diagram for the same mean value.

Examples of Haigh diagrams are shown in figure 7.

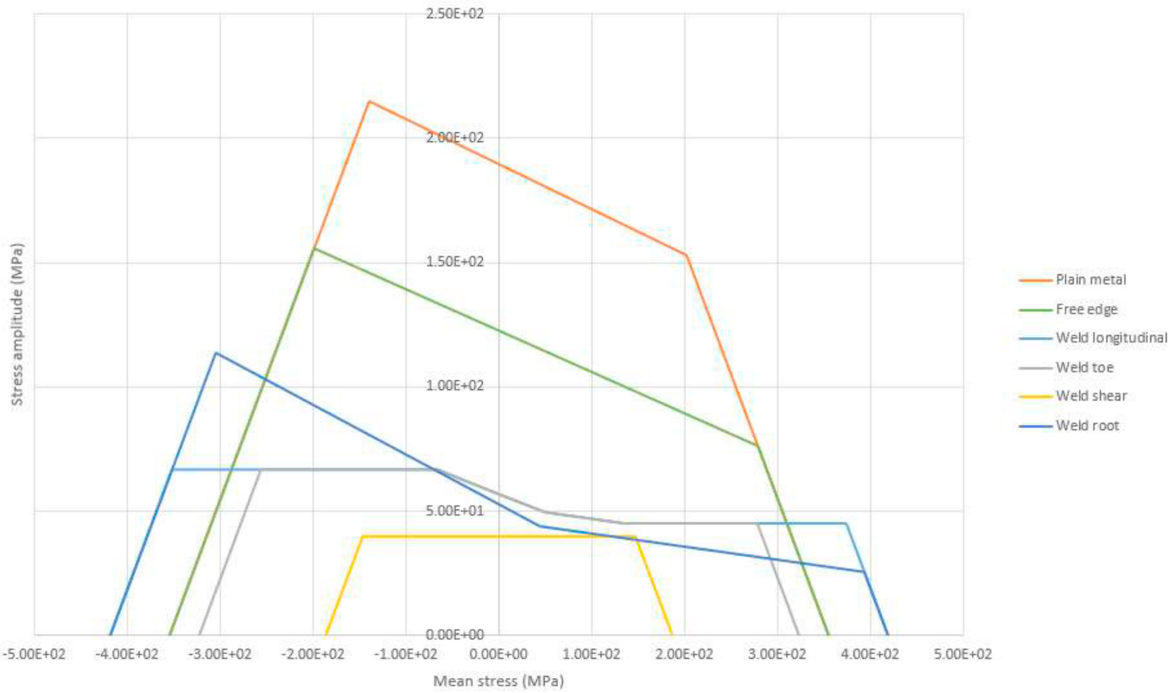



Fig. 7. Haigh diagrams according to the different zones of the model.

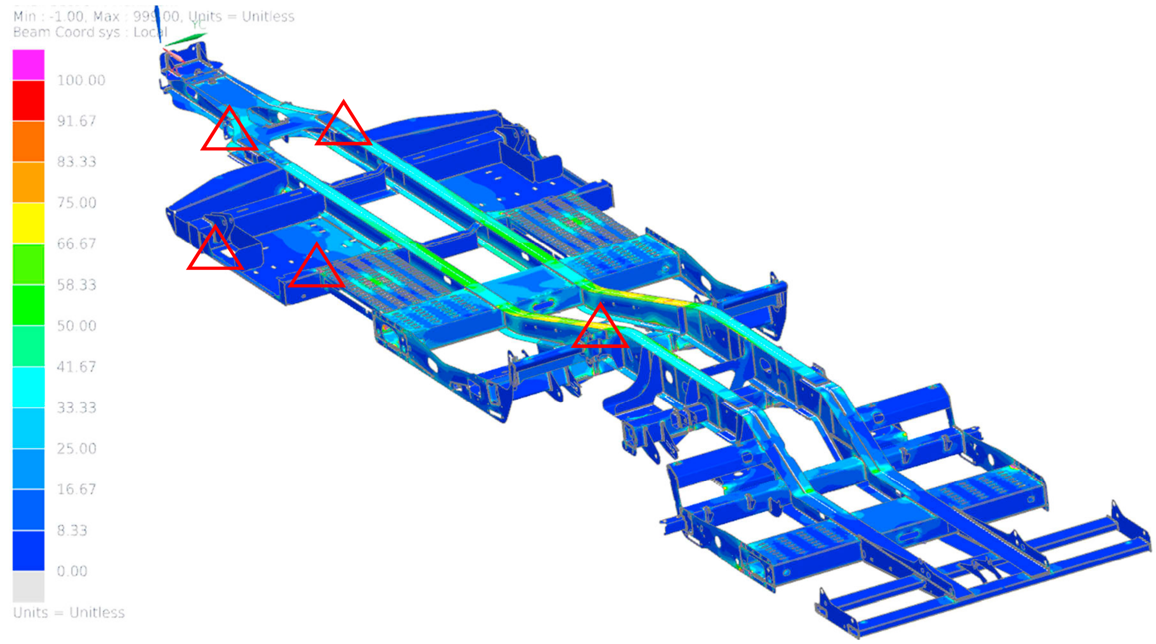



Fig.8. Trailer chassis fatigue results (%/fatigue limit)

## 4. Vehicles inspection and comparison

Vehicles in second hand market and with mileage between 1 and 1.5 million kilometers have been accessible for trailer structure inspection. Some cracks have been located, but no long or fatal crack has been observed. Comparison between crack locations and fatigue results is performed:

- In the drawbar area, on the side of the main beam at the weld connection of a support plate a crack has been observed at the weld toe, see Fig. 9. The weld toe crack is starting at the lower flange plate edge. At this location, on the corresponding element at the lower flange plate free edge the fatigue result is 152 %:

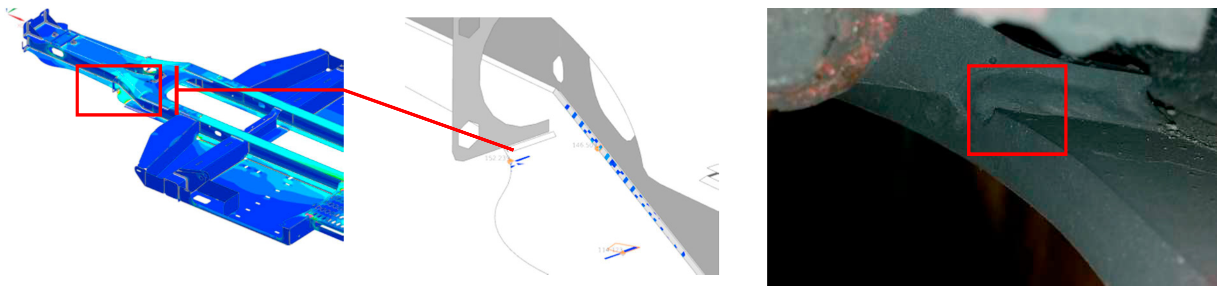


Fig.9. Drawbar support plate weld toe

- At the front side cross-member, at the edge radius before the folded metal sheet, a crack has been observed, see Fig.10. It is not a welded area but a steel plate cut edge with a high variation of shape. The fatigue result at this location is 189%:

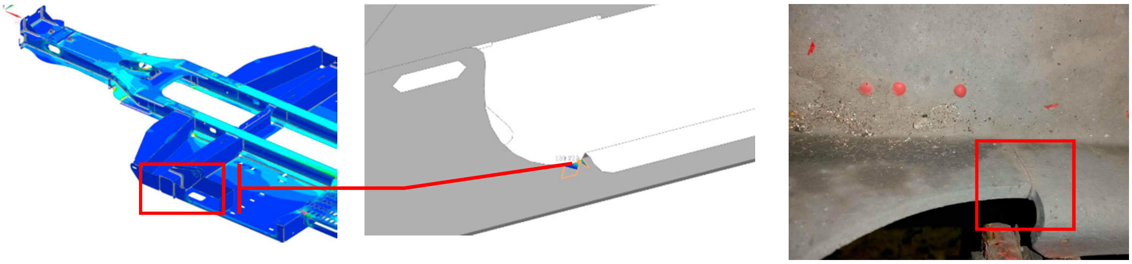


Fig.10. Front side fold

- At the front of the chassis frame, a crack has been observed at the cover plate lap joint weld, see Fig.11. The crack path is through the weld itself, even with the effective weld length much higher than the design size. The fatigue result at this location is 159%:

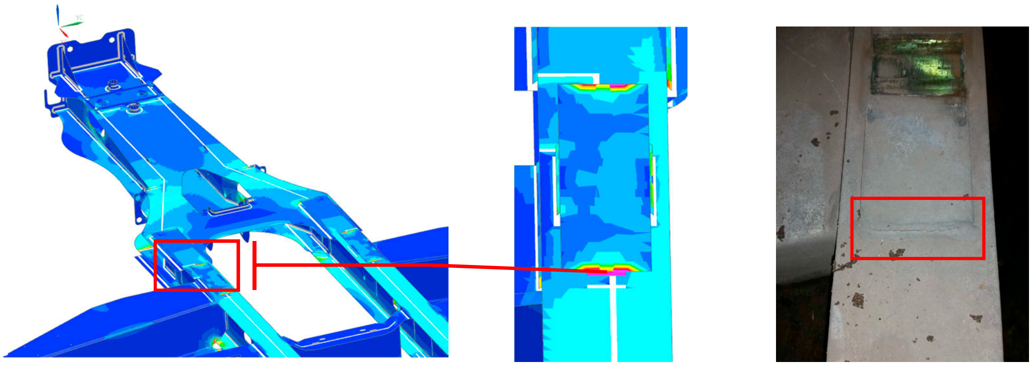


Fig.11. Chassis front cover plate

- At the front platform, a crack has been observed. It initiated at the weld end termination toe and propagated through the perforated plate, see Fig.12. At this location, stress range is high even at the basis of the fold and fatigue result at the weld termination end is 166%:

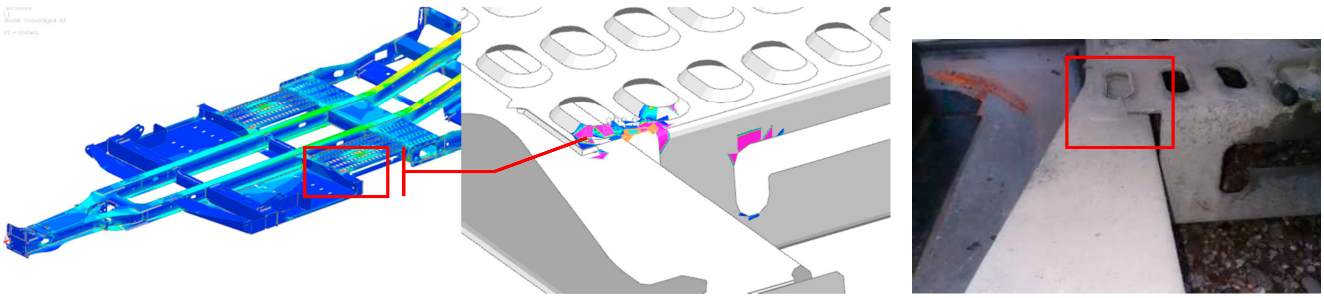


Fig.12. Chassis front platform

- In the central part of the chassis frame, a crack has been observed at weld termination toe. FEA result indicates stress exceeding the fatigue limit at the weld itself but not at the weld toe in the plate edge.

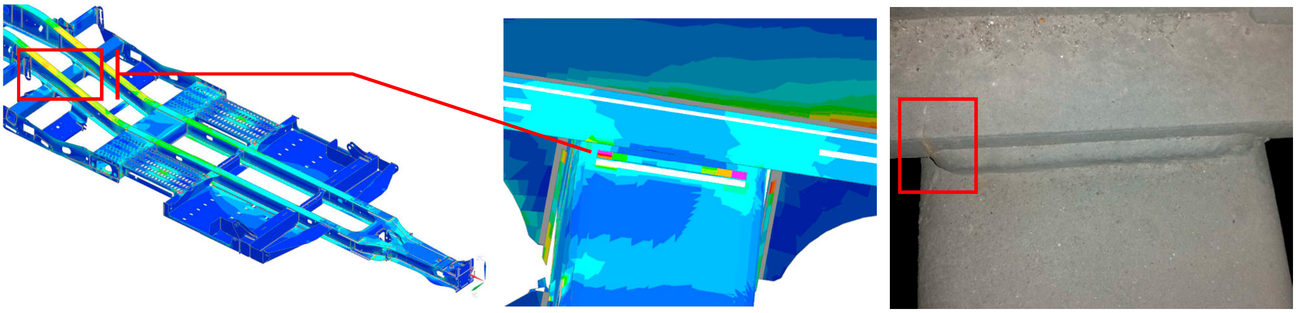


Fig.13. Cross member connection

Globally cracks have been observed in areas where the stress ranges exceed fatigue limits which have been determined for different initiation types, namely weld toe, weld termination toe and through the weld. All weld toe cracks have initiated in plate cut edge faces. For the last observed crack, the location of initiation is not appropriate and a more detailed investigation has to be performed.

Back in the FEA model, zone indications for which correspond specific fatigue criteria are plot at the cross member connection. It is noticeable that the element edge adjacent to the weld termination has a plate cut edge attribute. In fact this allocation is not correct for an edge node in connection with the weld.

For this specific node the attribute shall be shifted to weld toe and structural stress local axis shall be oriented tangential to the cut edge and perpendicular to the weld termination edge. When handling corrections as in Fig.14 and taking into account the correct stress variation components, the fatigue results reach 115% which exceeds the fatigue limit at weld toe.

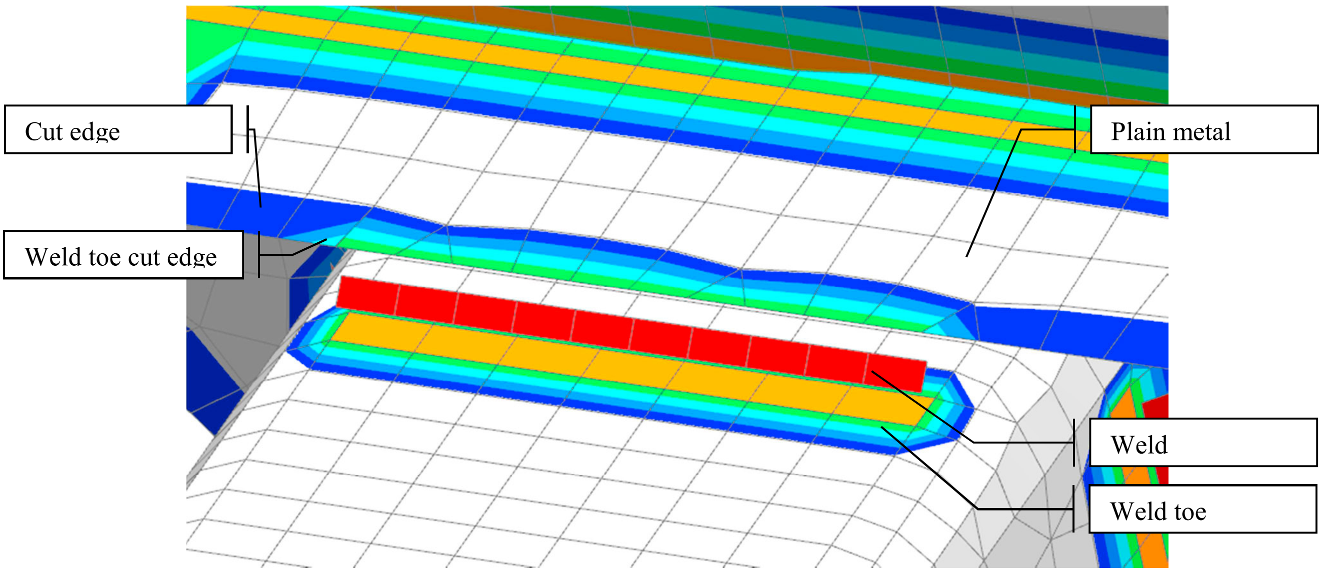



Fig.14. corrected area nodal indicator

## 5. Conclusion

LOHR Industrie has developed a shell element model for welded structure fatigue strength evaluation. The current project aims at verifying its assessments, making a global correlation on an old generation trailer design, for which end of life inspection is possible and to compare observed crack locations with fatigue structural stress results.

Each step of the model preparation is described. The enhanced structural stress calculation is performed, as is the comparison with the inspection results of four second hand market trailer structures, each having more than 1 million kilometers mileage: cracks have been observed at weld toes and at weld termination toes at cut edges of plates. One case is weld cracking from the weld root.

All of them correlated areas where calculation leads to stress levels exceeding the fatigue limit but one: the cross member connection crack has been observed despite model lack of prediction.

A detailed analysis of the model emphasized that in this specific case, the fatigue limit was not correct. The location of the crack has to fit two attributes: plate cut edge and weld termination toe. Only plate cut edge indication had been used for fatigue assessment and this has been corrected with the appropriate stress components in tangential direction along the plate edge.

Afterward, fatigue results were predictive, exceeding the fatigue limit. This improvement including the new weld toe indicator at plate cut edge allows POSTAL, the LOHR Industrie fatigue assessment tool, to be more robust.

## Acknowledgements

The major part of this investigational work has been performed by Selman Yildiz during his UTBM engineer school training period. Special thanks to him for building the trailer FEA model and for carrying out the analysis.

## References

[1] Turlier D, “Idéalisation des assemblages vissés et des soudures continues pour l’analyse MEF en éléments coques des structures de véhicules lourds soumis à des charges statiques et de fatigue », NAFEMS conference Modélisation et Simulation des Assemblages, Paris 30 mai 2013.

[2] Turlier D, Klein P, Wilhelm A, De Azevedo M (2010) “Seam Sim” method for seam weld structural assessment using shell element FEA model. UK 2010 NAFEMS conference proceedings. NAFEMS, Glasgow, pp 65–71 and NAFEMS benchmark magazine April 2010.

[3] Turlier D, Klein P and Bérard F (2010): “Seam Sim” method for seam weld structural assessment within a structure FEA. Proc. Int. Conf. AWST 2010, GEDIK Education foundation, Istanbul, pp 651–658

[4] Turlier, D., Klein, P. & Bérard, F., FEA shell element model for enhanced structural stress analysis of seam welds, Weld World (2014) 58: 511. doi:10.1007/s40194-014-0134-y

[5] Niemi E, Fricke W, Maddox SJ, Structural Hot-Spot Stress Approach to Fatigue Analysis of Welded Components - Designer’s Guide, Second Edition, Springer Nature Singapore Pte Ltd. 2018. doi:10.1007/978-981-10-5568-3

[6] Fricke, W., Kahl, A. & Paetzold, H., Fatigue Assessment of Root Cracking of Fillet Welds Subject to Throat Bending using the Structural Stress Approach, Weld World (2006) 50: 64. doi:10.1007/BF03266538

[7] Fricke W (2013) IIW guideline for the assessment of weld root fatigue. Weld World 57(6):753–791. doi:10.1007/s40194-013-0066-y