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.