Analysis of Longitudinal Beam Damage of a Container Chassis Semi-Trailer Using Finite Element Method – Case Study

Kazimierz Drozd¹, Aleksander Nieoczym²

¹Department of Materials Engineering, Faculty of Mechanical Engineering, Lublin University of Technology, Lublin, Poland
²Department of Machine Design, Faculty of Mechanical Engineering, Lublin University of Technology, Lublin, Poland

k.drozd@pollub.pl

Abstract. This study investigates the damage of a container chassis loaded with a container having a capacity of 22 thousand litres. The chassis was damaged during full-load operation. Based on the observed effects of damage induced in a longitudinal beam (longeron) of the container chassis, a hypothetical course of events was elaborated. An attempt was made to recreate the actual course of events by eliminating the reasons due to which the effects of damage significantly differed from those observed post factums. To this end, numerical finite element method (FEM) simulations were performed using the Abaqus software. The damaged longitudinal beam of the container chassis was modelled using three-dimensional (3D) solid elements. It was assumed that the beam had the same profile cross-section over its entire length. In the numerical model, the container chassis had a pinned support on the fifth wheel. Elastic elements were used to model the axle-supported regions of the container chassis with the wheels. The following three events were identified as the potential causes of container chassis damage: the container chassis was supported by the bumper on the ground when the wheels on the right side were in contact with the bottom of the road surface cavity; the container chassis was supported by the bumper on the ground when the wheels on the right side were not in contact with the bottom of the road surface cavity; the bumper was subjected to point impact during vehicle reversal. The numerical results demonstrated that for each of the proposed damage hypothesis, the highest reduced stresses were located in the very region where the container chassis damage was actually observed. It was found that the observed damage was most probably induced when the container chassis was supported on the bumper and there was no contact between the wheels and the road surface due to the presence of a cavity in the road surface. Moreover, analytical calculation results demonstrated that the force acting on the structural members of the container chassis in the analysed load case exceeded the force resulting from the static load of the structure by more than 2.5 times. The load-induced stress in the beam material significantly exceeded the strength of the structure.

1. Introduction

The frame is the main structural member of a semi-trailer, and it transfers both loads from the transported cargo weight and road surface reactions. The correct vehicle design and its execution in compliance with requirements specified in the vehicle approval are key factors ensuring the durability and reliability of the semi-trailer. Even if these requirements are satisfied, there are cases when a
vehicle is excluded from further use due to its structural damage. Potential damage – whatever its cause – is modelled and evaluated via simulations. Nowadays technical objects are modelled by the finite element method (FEM) in order to achieve basically two aims. One is related to the analysis of stresses in the current design of a semi-trailer or its design optimization. The other concerns the identification and verification of potential causes of damage.

In FEM strength calculations, the main task is to calculate load forces and torques and to simulate road conditions in such a way that the numerically modelled conditions would reflect the real road conditions. The FEM analysis begins with calculating strength under static load from the weight of the cargo and the structure itself. The cargo load can be evenly distributed over the top surface of the longitudinal beam as shown by Han et al. [1] or Zhang et al. [2]. As a result, it is possible to determine bending stresses. If the model does not consider the support simulating reactions of the left front wheel, the load causes the longitudinal beam to undergo bending and torsion at the same time. The semi-trailer’s motion on an uneven road is simulated by eliminating support in the parts of the structure in which the road wheels are mounted [2]. In the simulations, the container chassis was loaded with dynamic forces for the following drive modes: driving on a rectilinear track with constant speed; driving on a curved track; acceleration; and braking [3]. The dynamic force required to apply load was calculated as the product of static load and dynamic coefficient.

The aim of the modal analysis of semi-trailer frame vibration is to investigate the effect of vibration frequency on the structural deformation mode. Zhang et al. [4] determined the ranges of vibration frequencies and their impact on the deformation of a semi-trailer frame resulting from bending and torsion of specified structural members. The problem of transport system vibration, in combination with the determination of stiffness and damping coefficients of its structural members, was solved by Abdelkareem et al. [5]. The authors of studies [4, 5] also proposed important frame design modifications by the use of additional stiffening elements or local increase in the cross-sectional area of the frame profile.

Numerical results are sometimes verified via bench tests. Napierala [6] investigated the strength of welded joints connecting longitudinal and transverse beams in a semi-trailer frame. Numerical results showed good agreement with those obtained in the experiment that was conducted with the use of resistance strain gauges and displacement sensors mounted on the frame elements. In [7] Dzialak and Karliński described the procedure for measuring the deflection of a lifted semi-trailer by the photogrammetric method. The method based on the use of the MTS road simulator, described by Kaczor et al. [8], involves fatigue verification of entire vehicles and their individual parts. Load distribution on several vehicle axles can be measured on a test stand equipped with a deformable beam [9].

2. Object of the study and research methods
The object of a FEM numerical analysis was a damaged three-axle semi-trailer for transporting standard 20’ (feet) containers (figure 1a). Loaded with a liquid cargo tank, this semi-trailer having an actual weight of 22 tons took part in two events. One was a collision at a stop situation, which resulted in rear bumper beam damage. The other event took place when the right wheels of the semi-trailer hit a pothole, which led to the right side of the rear bumper resting on the road surface. The analysed object had visible fractures of the right-side longeron located in relief holes (figures 1b-c) and visible deformation of the semi-trailer structural members (figures 1d-f).
Figure 1. Semi-trailer under analysis – green arrows show the place and direction of photographing: a) general view, b) – c) longeron damage location, f) deformed rear bumper, 1 – fracture initiation location.

The analysis has shown the presence of a 20 mm deep indentation located on the outer surface of the rear bumper (force in figure 1f). The lower plane of the bumper is deformed too (figure 1f); however, the arch-shaped element connecting the bumper with the semi-trailer frame is undamaged. The observed mode of deformation of the lower bumper shell indicates that the deformation could have occurred as a result of the bumper’s resting on the road surface.

The semi-trailer frame longitudinal beam (figure 1e) has a total length of approximately 8,300 mm. This I-beam has the overall cross-sectional dimensions of 360x100 mm and a wall thickness of 11 mm. In the rear part of the frame, only in the web, six triangular holes were factory-made (figure 1e). The frame damage took the form of material cracking of different lengths on the web between the triangular (relief) holes (figures 1b-e). As a result of the damage, the web fragments were separated by a distance of up to 25 mm (figure 1c). Given the deformation mode of the beam elements, it was concluded that the fracture initiated at Point 1 (figures 1b-e), where the greatest distance between the separated fragments could be observed. It was also observed that the fracture always initiated at the transition between the arch-shaped profile and the rectilinear edge of the triangular hole in the beam web.

Given that apart from the fracture of the right-side longeron and the rear bumper, no other structural deformation was observed, the model was simplified in terms of geometry and kinematics.
The geometric simplification consisted in analysing a single beam model, with its profile and transverse dimensions in the front (undamaged) end being unchanged, without the support and the rear bumper. The kinematic simplification was related to considering only the kinematic connection between the rear bumper and the beam end. This meant treating the bumper support as a non-deformable element. The above simplifications made it possible to investigate the quasi-static problem in a linear-elastic range and to shorten the simulation time considerably. This applied simplifications also resulted from the fact that – given an unknown driving speed – an attempt at simulating dynamic phenomena would have to involve optimization of the simulation parameters for several variants that are described later in the manuscript.

To determine the cause of semi-trailer frame damage, FEM strength simulations were carried out to verify three hypotheses of the potential course of events. In two of these hypotheses, the semi-trailer drove over a pothole, which resulted in the bumper’s resting on the road with the right side wheels of the vehicle having (hypothesis 1) or losing (hypothesis 2) contact with the bottom of the road defect. Under hypothesis 3, it was assumed that the semi-trailer damage was caused by point impact to the rear bumper of the semi-trailer.

Numerical analyses were carried out for three hypotheses, and for each of these hypotheses two variants (A and B) of beam loading were tested. The scope of the numerical simulations was extended to include these variants because the literature of the subject does not provide any recommendations with respect to loading a semi-trailer with a container mounted thereon. In the A variant, the beam with only continuous/distributed load (5 in figure 2) was modelled for half of the container weight of 110 kN. The other variant (B) involved modelling the beam with concentrated load (6 in figure 2) that was only applied to the ends of the container with a length of 5853 mm, i.e. where the container is attached to the frame. To each of these two points a load of 55 kN was applied, corresponding to a fourth of the weight of the entire container.

The beam model was assigned the properties of a linear-elastic material, i.e. S235 unalloyed steel having the following mechanical properties: Young’s modulus $E = 210$ GPa, Poisson's ratio $\nu = 0.30$, yield stress $R_p = 235$ MPa, ultimate tensile strength $F_{tu} = 360$ MPa [10]. In all numerical calculations, maps of reduced stresses were created according to the Huber-Misses hypothesis. Some of these maps are presented later in this manuscript. Conclusions regarding the probable course of events were drawn by comparing the calculated stresses with the parameters of the material used in the beam structure.

2.1. Model for testing hypotheses 1 and 2 assuming that the bumper is resting on the road surface

The beam was modelled as a solid spatial element having the real cross-sectional dimensions and the same cross-section of the profile over the entire length (figure 2). Strength tests were carried out on a fragment of the beam located above the axles of the wheels of a semi-trailer 1. The beam support was modelled using elastic elements having a stiffness of 11.2 MNm$^{-1}$ at points corresponding to the position of each of the three axles of the semi-trailer, which reflected the semi-trailer suspension and wheel stiffness [5]. At a distance of 565 mm from the front end, the beam was articulated in order to model support against a saddle 2. On the other end of the beam, at a distance corresponding to the position of the bumper relative to the structure, a reference point 3 was created and kinematically coupled with an end 4 of the support beam.

Every hypothesis was tested for two variants of load distribution on the semi-trailer beam. The first variant (A) involved uniform load distribution over the beam length between cargo securing points 5 (figure 2). The other variant (B) only took into account the distribution of load in container mounting points 6.
Figure 2. Model of longitudinal beam: 1 – spring support, 2 – articulated support on the saddle (fifth wheel), 3 – reference point on the bumper, 4 – rear end of the beam i, 5 – uniformly distributed load, 6 – concentrated load, 7 – force acting on the bumper.

The model was discretized using C3D10 tetragonal solid elements, which are 10-node finite elements with second-order shape function. The finite element mesh was additionally remeshed in the notches (where the highest stresses and strains were expected to occur) in order to locally increase computational accuracy.

2.2. Model for testing hypothesis 3 assuming that the bumper is subjected to point impact

The hypothesis assuming that the semi-trailer frame damage was caused by impact to the bumper was tested using the same geometric model as for hypotheses 1 and 2. However, the model had to be expanded to include force at reference point 3 (figure 2), which was to reflect the impact on the rear bumper. Only by analysing this force (its value being unknown) was it feasible to determine the effect of the impact to the rear bumper on the formation of stresses in the loaded semi-trailer frame.

Figure 3. Impact force calculation: a) – rear bumper model and the distribution of stresses used for calculating rear impact force: 1 – fixed support, 2 – direction of indenter motion, b) – depth of deformation vs. force, c) bumper section deformed by the indenter.

Force (7 in figure 2) was determined by numerical analysis for a geometric model of the buffer shown in figure 3a. According to measurements of the real object, the bumper was a closed rectangular section with the dimensions of 120x80 mm and a wall thickness of 5 mm. It was attached to supports 1. It was assumed that the supports were constrained from moving in a direction 2.

The bumper model (figure 3a) was discretized using the same type of finite elements and the same material properties as those used for modelling the beam. The simulation consisted of establishing the relationship between the force and the depth of deformation caused by a non-deformable rounded
indenter having 15 mm in diameter, pressing on the rear wall of the bumper in the location of the observed deformation 2. This relationship is illustrated in figure 3b. Based on the calculated deformation depth amounting to about 20 mm (figures 1f and 3b,c), the force acting on the bumper was estimated at 57 kN.

Having established that the force acting on the rear bumper is equal to 57 kN, the action of this force 7 was modelled for the entire beam, at the point marked as 3 (figure 2). This point was kinematically coupled with the beam end 4. The direction of this force was parallel to the longitudinal beam axis and its sense complied with that shown in the diagram.

3. Results and discussion

3.1. Numerical simulation of the case when the bumper is resting on the road surface

First of all, hypothesis 1 was tested, under which the bumper was resting on the road surface (3 in figure 2). The wheels of the semi-trailer got into the road defect and yet were in contact with the bottom of the cavity, and at the same time the semi-trailer was supported on the rear bumper. In the simulation of variant A, with the load uniformly distributed over the beam (5 in figure 2), the stresses did not exceed 225 MPa (figure 4). It was found that the highest stresses were located in the area with relief holes (figures 4b,c). The stress field of up to about 150 MPa extended from the upper flange of the longeron to the line marked by the upper edge of the triangular notches (1 in figures 4a,b). The greatest stress accumulation was observed at two points in the upper part of each of the triangular notches (2 in figure 4b). These points were located where the curvature of the notch changed, i.e. where the straight section of the notch ended and the arch-shaped region began. The simulated damage location is marked as 1 in figure 1b. Stresses calculated for the lower parts of the triangular notches are shown in figure 4c. It can be observed that the stresses exceeding 205 MPa only occur in the arch-shaped region, in the lower part of notch 3 located above the first axle of the semi-trailer wheels. A similar damage location obtained for the real object is marked as 1 in figure 1d.

![Figure 4](image_url)

**Figure 4.** Map of stresses obtained by verification of hypothesis 1 (wheels are in contact with the road cavity bottom) for variant A (frame loaded over the entire container length): a) – general view, b) – view of the upper edge of the holes, c) – view of the bottom of the holes.

In the simulation of load variant B, it was assumed that the weight of the container is distributed in half only at two points (6 in figure 2) of the beam. The highest stresses amount to 170 MPa (figure 5) and also occur at points 1 where the curvature of the side of the triangular hole made in the web changes (increases). These points are located in the upper part of all triangular holes made in the beam web. Moreover, one can observe relatively high stresses at points 2, between the double hole. The stress field at the top of the beam, above the line joining the upper edges of the triangular holes, is quite uniform in terms of value, and the distribution of stresses is similar to that in figure 4b.
Next, simulations were performed to verify hypothesis 2, under which it was assumed that the semi-trailer was supported on the rear bumper while the wheels were not in contact with the bottom of the road surface cavity. For load variant A (uniform loading of frame), the maximum stresses are equal to 239 MPa (figure 6). They are located in the upper part of the beam, where the curvature of the triangular notches changes, and at the bottom of notch 1. Stresses of this magnitude could cause plastic deformation of the longeron material. In addition to that, stresses amounting to at least 150 MPa are observed in the beam regions 2 and 3 where the triangular notches are adjacent to each other. An equivalent of this location is marked as 1 in figure 1e. It can be observed that in terms of value and distribution, the simulated stresses are similar to those obtained in the simulation verifying hypothesis 1 for load model A (figure 4).

For load variant B, the highest stresses induced in the longeron material due to the semi-trailer’s resting on the rear bumper amount to only 170 MPa. The stress values and maps obtained for the two-point load model (6 in figure 2) are exactly the same as those obtained for the model presented in figure 5.

3.2. Simulation of the case when the bumper is subjected to a point impact
To verify hypothesis 3, under which the rear bumper was subjected to impact that could cause local deformation, it was assumed that at point 3 (figure 2) reflecting the real damage location, a force of 57 kN (figure 3) was applied. For variant A with a uniform loading of the longeron along the entire length of the 20-foot container, the maximum stresses were located in the lower part of the longeron, in the area where it was supported by the third axis. However, these stresses were low and amounted to 70 MPa.
Stresses obtained in this case are the highest for all tested models. For the case of point load in variant B, these stresses are equal to 241 MPa (figure 7). These values are sufficient for inducing plastic deformation of the material. It should be noted that the simulated stress distribution in the longeron is more symmetrical in the area with the cut-out holes. As a result of this symmetry, the stresses on the lower edge of the notches are higher, e.g. at the location marked as 2. Moreover, the maximum stresses concentrated on the upper edge of the triangular notches have different ranges (areas) and different maximum values than those observed at point 1. These relations are in total agreement with the damage shown in figure 1b. Also, the stresses between the vertices of the triangles denoted by 3 are significantly higher than those observed for other maps, which means that they could cause the damage observed in figure 1e.

![Figure 7. Map of stresses obtained by verification of hypothesis 3 (impact to rear bumper) for load variant B (point load on container edges).](image)

### 3.3. Discussion

Table 1 gives the results of maximum stress location in the simulations of semi-trailer frame loading. For the three simulated variants, the approximate maximum stresses were calculated to range from 225 to 241 MPa. These stresses could cause plastic deformation observed in the damaged parts of the semi-trailer. The locations of the maximum stresses in the longeron observed for the three simulated variants were almost the same. They corresponded to the damage location marked in figures 1b,e. The simulation results did not unequivocally show any stress concentration that could cause the material discontinuity observed in figure 1c or the cracks marked as 1 in this figure. However, if we consider a sequence of two road events, such a significant damage of the frame, with the yield stress exceeding and material strengthening during the first event, seems possible as a result of the second event.

| Hypothesis | Variant | Maximum stress, MPa | Probability of beam damage (Y)es/(N)o | Location of maximum stress in figure1 |
|------------|---------|---------------------|-------------------------------------|-------------------------------------|
| 1          | A       | 225                 | Y                                   | b,e                                 |
|            | B       | 170                 | N                                   | b                                   |
| 2          | A       | 239                 | Y                                   | b,e                                 |
|            | B       | 170                 | N                                   | b                                   |
| 3          | A       | 70                  | N                                   | -                                   |
|            | B       | 241                 | Y                                   | b,e                                 |

In the simulation, two points of stress concentration were determined on every horizontal edge of the triangular notches. In real conditions, only one damage of this type was induced (figure 1b). This discrepancy can be explained by the fact that the initiation of this damage type or other damage (in other triangular notches or between them) resulted in material unloading, and therefore no two cracks progressing from the same straight edge of the triangle were developed in any of the notches.
Due to the fact that the stresses simulated for longeron load variant A (for hypotheses 1 and 2) were higher than the yield stress of the material, it can be concluded that the correct model for simulating the performance of a fully operational semi-trailer is the one in which the loads are distributed uniformly over the entire length of the contact between the container and the semi-trailer frame.

In all tested load variants and all models used for verification of the damage hypotheses, the stress field was asymmetrical, which is marked in the stress maps (figures 4-7) as the large colourful area in the upper part of the beam. Although less dangerous compressive stresses are induced there under a normal load when the semi-trailer is moving on an even surface, under unusual loading conditions – such as those taken in the hypotheses tested in this study, these stresses may change their sign and, as a result, cause longeron damage.

Regardless of the load variant applied, the results of the strength tests show that in each of the three considered cases of structural performance, the highest reduced stresses are located in the triangular notches in the examined part of the longitudinal beam. Specifically, the highest reduced stresses are located in or near the rounded corners of the notches, which coincides with the location of cracks in the real structure.

In the analysis of the first two hypotheses of damage for the case of the semi-trailer’s resting on the bumper, the load values reflect static impact, which means that the dynamic impact on point 3 in figure 2 was ignored in the simulation. According to the description of the event, the wheels fell into a cavity in the road surface, which made the semi-trailer hit the ground with the bumper’s resting on the ground. By simplifying the model of this motion to free fall, it is possible to estimate the force $F$ during the rear bumper’s hitting on the ground. By comparing the exchange of potential energy of the semi-trailer when the bumper was at a height of $h = 300\text{ mm}$ (figure 1f) into kinetic energy of vertical motion with a gravitational acceleration $g = 9.81\text{ ms}^{-2}$, it is possible to calculate the change in vertical velocity $\Delta v = \sqrt{2 \cdot g \cdot h} = 0.767\text{ ms}^{-1}$. From Newton's second law we obtain the equality $m \cdot \Delta v = F \cdot \Delta t$, which can be used to calculate the force. After substituting $\Delta v$ and $\Delta t = 0.03\text{ s}$, which is the time of dynamic interaction between the bumper and the road, we determine the force to be equal to $F = 25.6 \cdot m$. This means that the dynamic interaction generated during the bumper’s fall, which was ignored in the models, may be more than 2.5 times greater than the considered static force resulting from the mass $m$ on the bumper if the longeron is supported by this bumper and the fifth wheel only. Taking into account the above dependence, the reduced stresses calculated for the first two hypotheses would be proportionally greater in the elastic deformation range (assuming linear deformability of the material), i.e. up to the yield stress.

4. Conclusions
Based on the results of this study, it was concluded that the damage of the longitudinal beam of the semi-trailer frame could be caused or conducted to by the semi-trailer’s resting on the bumper. Ignored in modelling, the dynamic interaction generated during the bumper’s fall may be more than 2.5 times greater than the static force resulting from the mass $m$ on the bumper if the longeron was only supported by this bumper and the fifth wheel. One should reject the hypothesis that the damage of the longeron was caused by point impact to the rear bumper; however, this type of impact could cause deformation of the frame member. The correct model for simulating the loading of a semi-trailer in working order is that in which the load is distributed uniformly over the entire length of the contact between the container and the frame.

Acknowledgment
The project/research was financed in the framework of the project Lublin University of Technology-Regional Excellence Initiative, funded by the Polish Ministry of Science and Higher Education (contract no. 030/RID/2018/19).
References

[1] Q. F. Han, Q. B. Huang and S. D. Li, “Comparative Analysis of Static Strength of Traditional and Bionic Semi-trailer Frame,” Applied Mechanics and Materials, vol. 470, pp. 320-324, 2013. doi:10.4028/www.scientific.net/AMM.470.320

[2] W. C. Zhang, X. B. Du, B. B. Ma, B. H. Pei, J. Chen and Z. X. Liu, “The Finite Element Analysis and Topology Optimization of Semi-trailer Frame,” Advanced Materials Research, vol. 424-425, pp. 90-93, 2012. doi:10.4028/www.scientific.net/AMR.424-425.90

[3] W. Cheng, C. Kaiwu, Q. Fukang and X. Jin, “Study on Fatigue Strength and Life of Freight Car Frame after Making Holes,” MATEC Web of Conferences, vol. 175, pp. 1-4, 2018. doi:10.1051/matecconf/201817501035

[4] P. M. Zhang, W. Wang and W. C. Zhang, “Finite Element Modal Analysis of a Semi-trailer Frame,” Applied Mechanics and Materials, vol. 685, pp. 199-203, 2014. doi:10.4028/www.scientific.net/AMM.685.199

[5] M. A. Abdelkareem, M. M. Makrah, A. M. Abd-El-Tawwab, S. A. EL-Razaz, M. Kamal Ahmed Ali and M. M. Moheyeldeen, “An analytical study of the performance indices of articulated truck semi-trailer during three different cases to improve the driver comfort,” Proceedings of the Institution of Mechanical Engineers, Part K: Journal of Multi-body Dynamics, vol. 232, pp. 84-102, 2018. doi:10.1177/1464419317709895.

[6] R. Napierała, “Numerical-experimental static analysis of the effort of the crossbar-longeron in the semi-trailer frame,” AIP Conf. Proc., vol. 2078, pp. 1-6, 2018. doi:10.1063/1.5092101

[7] P. Działa and J. Karliński, “Comparative examination of the trailer frame in accordance with UIC 596-5,” Materials Today: Proceedings, vol. 12, pp. 416-422, 2019.

[8] M. Kaczer and M. Janusza, “Optimization of semi-trailer design as a result of using the MTS road simulator,” Mechanik, vol. 92, pp. 52-54, 2019. doi:10.17814/mechanik.2019.1.10

[9] A. Nieoczym, K. Drozd, A. Wójcik, “Geometric optimization of a beam detector for a WIM system”, Advances in Science and Technology Research Journal, vol. 12, pp. 233-241, 2018. doi:10.12913/22998624/97296

[10] EN 10025-2:2019, “Hot rolled products of structural steels - Part 2: Technical delivery conditions for non-alloy structural steels.”