Development and analysis of a prototype car-carrier structure using the finite element method

Z. Li, W. Altenhof, R. Gaspar & P. Frise
Department of Mechanical, Automotive, and Materials Engineering, University of Windsor, Canada.

Abstract

This paper describes a computer-based modeling and simulation effort to aid in the development and design improvements of a prototype car-carrier structure. The most common approach for transporting vehicles over land in North America is the use of truck auto-carriers. The auto-carrier’s structures and weld points often crack and break during service due to fatigue stresses. In addition, the large deflections on the top decks of the auto-carrier contribute considerable damage to the carrier structures and to the loaded vehicles. In this research, a finite element model of a prototype car-carrier structure was developed and numerical computations were conducted in LS-DYNA. The simulation results correlated well with responses observed from actual car-carriers. Stress distribution and deflections of the structures were investigated when the trailer negotiates speed bumps. Structural modifications were considered to minimize deflections and stress concentrations.

1 Introduction

The delivery of vehicles to dealers is one of the major tasks in the vehicle production industry. With the development of highway systems in North America, shipment by haul-away (truck) auto-carriers is the most common method for transporting vehicles over land. Most often vehicles will be transported on a large carrier that is designed specifically for transporting 8 to 11 passenger vehicles, each on its own support deck. Car-carriers are usually built in the form shown in Figure 1, which is a headramp-trailer combination.
Significant shock and vibration are induced within a car-carrier during travel over harsh roads. With more car-carriers in service at highway driving conditions, structural fractures or breaks are more common. Most of these failures are caused by repeated stress application. These fractures or breaks typically initiate at stress concentration areas, although the overall stress levels may be lower than the yield strength of the structure. According to fatigue theory for steel structures, the life of the structure increases as the repeatedly applied stress decreases, for stresses between the endurance limit and the ultimate strength [1]. Figure 2 illustrates a typical $S_f/S_u$ versus number of cycles to failure curve for steel alloys, in which $S_f$ represents the fatigue stress and $S_u$ represents the ultimate strength.

![Figure 2: Generalized S-N curve for steel.](image)

Also, due to the high mass centre of the loaded car-carrier and its slender structure configurations, the large deflections (especially lateral and vertical deflections) of the structural members may cause damage to the loaded vehicles.

This paper will investigate the deflection and stress distribution profile over a prototype trailer structure, as illustrated in Figure 3, designed by StarTrans, an auto-carrier manufacturer located in Ontario, Canada. The results of the numerical simulations are used to improve the structural design, decrease deflections, and reduce stress concentrations to improve fatigue life.
Banthia et al. [2] developed a finite element (FE) model of a three-stacked railroad hauling car carrier in order to analyse the dynamic response of the structure. In the numerical model, the loaded vehicles were simulated as lumped masses. Letherwood and Wehage [3] published a paper describing computer-based modelling, simulation, and validation to aid in the development, testing, and procurement of a lightweight, High Mobility Trailer (HMT) towed by the High Mobility Multipurpose Wheeled Vehicle (HMMWV). Simulations were performed on obstacle avoidance manoeuvres, bumps, potholes, and country courses. Nigel [4] used the FE method to investigate design modifications to a whole bus structure which experienced premature failure due to fatigue loading conditions. The results of the research led to a significant increase in the life of the bus structure. In a similar study by Orringer and Tong [5] gusset plates and beams with round cross-sections were used to enhance the fatigue strength of a bus structure.

To the best of the authors’ knowledge, no one has ever developed a FE model of a car-carrier and analyzed the associated stress distribution or deformation by the car-carrier during transit under harsh driving conditions. This research represents the first attempt made in the field of dynamic finite element analysis to aid in the design and development of car-carriers.

2 Finite element model development

The trailer can hold 7 passenger cars with 8 adjustable loading decks. The trailer’s mass is approximately 6,000 kg with dimensions of 14.5m length, 2.6 m width, and 2.6 m height, excluding the loaded vehicles.

2.1 Trailer geometry

The main structural members used to construct the trailer are rectangular hollow structure section tubes, plates, and round pins. All structures of the prototype trailer were created and meshed in Finite Element Model Builder (FEMB) version 27.
2.1.1 Model discretization
Due to the size of the vehicle carrier an emphasis was placed on developing a numerical model which minimized the number of finite elements while ensuring acceptable simulation results. Hence a guideline, which is outlined below, was utilized in the development of the FE model of the car-carrier.

2.1.1.1 Element type and geometry Tubes and plates were modeled using either triangular or quadrilateral shell elements. Pins and round bars were modeled using either hexahedral or wedge elements. For reasons of better accuracy and efficiency, quadrilateral elements were preferred for two-dimensional meshes and hexahedral elements for three-dimensional meshes [6]. However in some areas, in order to decrease the number of elements, triangular elements (2.4% of the total shell elements) and wedge elements (4.5% of the total solid elements) were introduced. Figure 4 illustrates the FE mesh distribution for tubular and round bar geometries.

![Mesh of tubes and cross section mesh of round bars.](image)

2.1.1.2 Element formulation Fully integrated shell elements with 2×2 gaussian quadrature were used for all shell elements (Hughes-Liu shell element formulation in LS-DYNA). A constant stress solid element formulation (formulation #1 in LS-DYNA) was used for all solid elements. For all beam elements, formulation #6 with tubular cross section type was utilized.

2.1.1.3 Element size Although larger element sizes result in lower CPU times, large elements cannot accurately represent small detailed structures. In the FE model, the minimum shell element side length is 4.75mm and the minimum solid element side length is 2.2 mm.

2.1.1.4 Structure simplifications and omissions Some auxiliary parts of the trailer were not included in the model. These parts do not experience significant force or deformation when the vehicles are in transit, such as hydraulic cylinders, control panels, unused supporting decks, and springs. These entities have been excluded from the numerical model.

2.1.2 Numerical modelling of weld points
In the trailer structure welding is used to fasten different entities together. In the FE model, *CONSTRAINED.SPOTWELD was used to represent the welded
points between shell elements structures. For welding points between solid elements and shell element structures, spotwelds could not be used as solid elements do not have rotational degrees of freedom. In this case, *CONSTRAINED_EXTRA_NODES was used to constrain nodes on the shell parts to the rigid solid parts.

2.1.3 Summary of the complete trailer model
The positions of the loading decks in the trailer are changeable. In the FE model, the positions of the upper decks were adjusted to the limit height (4.6 m overall) in order to simulate the worst loading condition. The final FE model of the trailer contains 124,465 nodes and 105,390 elements (6,895 spotwelds, 78,955 quadrilateral elements, 1,904 triangle elements, 16,820 hexahedral elements, 792 wedges, and 24 beam elements). Figure 5 illustrates the FE model of the complete trailer.

![Figure 5: Illustration of the FE model of the trailer.](image)

2.2 Loaded vehicle model

This prototype car-carrier was designed to carry cars, pick-up trucks, minivans and sport utility vehicles (SUVs), among which SUVs are the heaviest. In the FE model, the worst loading condition was simulated; six SUVs were chosen as the loaded vehicles on the trailer. The SUVs chosen in the model have an average base curb weight of 3,175 kg and mass center height of 0.8 mm. Possible interactions (contacts) between the SUVs along the longitudinal direction were not considered in this investigation.

The SUV model was simplified as one rigid block and four deformable blocks. The rigid block with a mass of 3,175 kg represents the whole weight of the SUV. The four identical deformable blocks were used to simulate the tires.

After the vehicle is driven on the support deck, it is tied down using four steel chains. The chain’s hook is inserted into the tie-down slot on the base frame of the vehicle and the other end of the chain is wound around a ratchet tube. An initial tensile force of approximate 900 N is generated when the vehicle is tied
down. The chain is modeled as a beam element with one end on the vehicle and another on the ratchet tube of the trailer.

In order to model the four tires as one entity to represent the entire SUV, all top nodes of the tires and the four detached nodes (representing the tie-down slots) were constrained to the rigid mass (see Figure 6) using the command *CONSTRANGED_EXTRA_NODES.

![Figure 6: The FE model of one loaded vehicle.](image)

### 2.3 Material models

According to the performance and loading conditions of the trailer structures in service, appropriate material models were chosen from the LS-DYNA material library. Shell elements with side length less than 5.0 mm and all solid elements were assigned a rigid material model (material type 20 in LS-DYNA). This will allow a large saving in CPU time with the potential of not significantly altering accuracy. Material type 12 (*MAT_ISO1ROPIC_ELASTIC PLASTIC) was chosen for all other shell elements. Material type 71 (*MAT CABLE_DISCRETE_BEAM) was used for the beam elements. This model permits the elastic cable to be modeled realistically without the cable providing any load bearing capacity if it is in compression.

### 2.4 Contact algorithms

Three different types of contact algorithms were used in the numerical model. *CONTACT_AUTOMATIC_NODES_TO_SURFACE, *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE, and *CONTACT_AUTOMATIC_SINGLE_SURFACE were used to investigate interactions between structural members within the car-carrier. Automatic type contact simplifies the problem definition and is generally more reliable. Surface-to-surface contact is fully symmetric; choice of slave and master surface is arbitrary. Nodes-to-surface contact is especially suited to nodes contacting rigid bodies. In total, 137 contact algorithms were defined in the model.
2.5 Boundary conditions

An initial velocity of 60 km/hr was prescribed for the trailer in its forward direction. Gravity is applied to all elements in the model.

During the carrier’s motion in operating conditions, the trailer is supported in three positions; the hitch pin and the suspension frames on both sides of the trailer. The impacts on the trailer, due to harsh driving, are transmitted to the trailer structure through these three locations. The vibration data (displacement) of the three locations were prescribed in the FE model. The left side (driver’s side) of the trailer negotiates a speed bump which initiates vertical motion in both the left and right side suspension locations. Immediately afterward, the right side of the trailer suspension location impact another bump which again initiates motion in both of the suspension locations. The vertical displacement versus time curves representing the prescribed motions of the left and right suspension locations are illustrated in Figure 7. Since the profiles are almost sinusoidal, the stresses thereby caused on the trailer structures are repeated fatigue stresses. In the model *BOUNDARY_PRESERVED_MOTION_SET was used to define the motions.

![Figure 7: Vertical movement versus time curves for suspension frame locations.](image)

2.6 Simulation validations

All numerical simulations were conducted using LS-DYNA version 960 on a personal computer with 512 megabytes of RAM employing dual AMD Athlon 1.8 GHz processors. The simulation duration is 0.5 seconds and the approximate CPU time was 53 hours for each simulation.

Both hourglass and sliding interface energies were calculated during the simulations. For all simulations, both the hourglass and sliding interface energies were less than 5% of the total energy.
3 Analysis of simulation results

Examination of the effective stress distribution over the duration of the simulation showed five regions of high stress concentrations: front hitch pin region, front base corner region, middle base corner region, rear base corner region, and suspension vertical tube region. Most fractures and cracks were also found at those locations in service of a similar trailer. Figure 8 illustrates the effective stress contours of two of these regions. Large lateral deflections in the rear top deck of the model were also observed. The results of the simulation were analyzed by engineers from StarTrans and an acceptable engineering correlation was observed between the FE model and actual loading conditions. These observations imply that the FE model can be used to predict the response of the trailer under dynamic loading.

Figure 8: Two typical stress concentration regions.

4 Structural modifications

Local geometry modifications were made to decrease stress concentrations for the specific regions. For the hitch pin area, a stiffening plate was extended and two gusset plates were added with the anticipation of strengthening the hitch pin area. In the front, middle, and rear base stress concentration regions, the original plates were enlarged. These enlarged plates also contributed to reduce the lateral deflection of the top decks. In addition, supporting tubes were added in the front and the middle to reduce the stress concentrations on the base tubes and to reduce lateral deflections in the top. Figure 9 shows the global locations of the stress concentration regions and modifications to these regions.

4.1 Reduction in lateral deflections

Analysis of the output data from the simulation results reveals that the maximum lateral deflections occur at the rear top deck. The structural modifications reduced
the maximum lateral deflection at the left side of the trailer by 27% (from 150 mm to 110 mm) and the maximum lateral deflection at the right side by 25% (from 140 mm to 105 mm). Figure 10 illustrates the maximum lateral deflection comparison of the trailer at the left side, before and after modifications.

Figure 10: The maximum deflection comparison on the left side of top rear deck.

4.2 Reduction in the maximum effective stresses

By comparing the maximum effective stress in each region of concern, before and after the modifications, the influences of the structural changes can be observed. Table 1 summarizes the maximum stresses observed in each region of concern, before and after structural modifications. Modification in vertical suspension tube region is most effective, reducing the stress magnitude by 53%; modification in rear base corner is least effective, reducing stress by 12%, although modification
in this region contributed a significant reduction in the lateral deflection for the rear top deck.

Table 1. Maximum stress comparison. (Unit: MPa)

<table>
<thead>
<tr>
<th></th>
<th>Front hitch pin</th>
<th>Front base corner</th>
<th>Middle base corner</th>
<th>Rear base corner</th>
<th>Suspension vertical tubes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Before Modifications</td>
<td>320</td>
<td>120</td>
<td>240</td>
<td>250</td>
<td>320</td>
</tr>
<tr>
<td>After Modifications</td>
<td>250</td>
<td>90</td>
<td>200</td>
<td>220</td>
<td>150</td>
</tr>
<tr>
<td>Reduction</td>
<td>22%</td>
<td>25%</td>
<td>17%</td>
<td>12%</td>
<td>53%</td>
</tr>
</tbody>
</table>

5 Conclusions

The simulations have shown that the finite element method can predict the actual responses of the trailer under dynamic loading which results in cyclic stresses within the structural members. Modification to the local geometry and addition of structural supports can decrease lateral deflections and local stress concentrations in the carrier. In order to achieve further reduction in the lateral deflections on the rear top areas of the trailer, global modifications will have to be considered. This investigation and research will be conducted in future work.

References