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

Zhanbiao Li
University of Windsor

Follow this and additional works at: https://scholar.uwindsor.ca/etd

Recommended Citation
https://scholar.uwindsor.ca/etd/1694

This online database contains the full-text of PhD dissertations and Masters' theses of University of Windsor students from 1954 forward. These documents are made available for personal study and research purposes only, in accordance with the Canadian Copyright Act and the Creative Commons license—CC BY-NC-ND (Attribution, Non-Commercial, No Derivative Works). Under this license, works must always be attributed to the copyright holder (original author), cannot be used for any commercial purposes, and may not be altered. Any other use would require the permission of the copyright holder. Students may inquire about withdrawing their dissertation and/or thesis from this database. For additional inquiries, please contact the repository administrator via email (scholarship@uwindsor.ca) or by telephone at 519-253-3000 ext. 3208.
Development and Analysis of A Prototype Car-Carrier Structure Using the Finite Element Method

By

Zhanbiao Li

A Thesis submitted to the
Faculty of Graduate Studies and Research
through the
Department of Mechanical, Automotive and Materials Engineering
in Partial Fulfillment of the Requirements for the
Degree of Master of Applied Science
at the University of Windsor

Windsor, Ontario, Canada

2003

© Zhanbiao Li, 2003
The author has granted a non-exclusive licence allowing the National Library of Canada to reproduce, loan, distribute or sell copies of this thesis in microform, paper or electronic formats.

The author retains ownership of the copyright in this thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without the author’s permission.

L’auteur a accordé une licence non exclusive permettant à la Bibliothèque nationale du Canada de reproduire, prêter, distribuer ou vendre des copies de cette thèse sous la forme de microfiche/film, de reproduction sur papier ou sur format électronique.

L’auteur conserve la propriété du droit d’auteur qui protège cette thèse. Ni la thèse ni des extraits substantiels de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation.
ABSTRACT

A very common mode for transporting vehicles over land in North America is the use of truck car-carriers. This thesis describes a computer-based modeling, simulation, and validation effort to aid in the development, testing, and design improvements of the structure of a prototype car-carrier. Car-carrier structures often crack and break during service due to fatigue stress application. In addition, the large deflections on the top decks of the trailer of a car-carrier may cause considerable damage to the carrier’s structure and to the loaded vehicles due to possible collisions with the carrier as well as collisions with other loaded vehicles. In this research, numerical and experimental methods are used to investigate, analyze, and improve a prototype car-carrier’s structure. A finite element model of the trailer of the prototype car-carrier was developed and model validation was completed with experimental measurements taken from field tests using a truck car-carrier when negotiating a typical speed bump. Numerical computations were conducted using LS-DYNA and the simulation results were examined in LS-POST to compare the simulation results with the field test data. After an experimental/numerical comparison, local and global geometric modifications were investigated to minimize lateral structural deflections and stress/strain concentrations. It was found that the globally modified model significantly decreased stress concentrations and lateral deflections of the trailer. The investigation shows that the finite element method can provide a powerful tool for design optimization of truck car-carriers and other dynamically loaded vehicle structures.
ACKNOWLEDGEMENT

The author would like to express his appreciation to the many individuals and organizations that contributed to this project. First, the assistance of Drs. William Altenhof and Peter Frise is gratefully acknowledged. With the aid of these two fine professors the author has grown intellectually. Thank you for your time, efforts, guidance, patience, recommendations, and encouragement during this research.

The author would also like to express his thanks to Dr. Robert Gaspar for providing the test instruments and helping conduct the field tests. In addition, the extensive help and in-kind contributions from Allied System (Canada) Company, StarTrans Company, Dalor Industries Inc., and the City of Windsor Traffic Engineering Department are gratefully acknowledged.

Last but not least, the financial support from Materials and Manufacturing Ontario (MMO) is also greatly appreciated.
# TABLE OF CONTENTS

ABSTRACT ......................................................................................................................... iii
ACKNOWLEDGEMENTS ........................................................................................................ iv
TABLE OF CONTENTS ......................................................................................................... v
LIST OF FIGURES ............................................................................................................... x
LIST OF TABLES ............................................................................................................... xv
GLOSSARY OF ABBREVIATIONS ....................................................................................... xvi

1. INTRODUCTION .............................................................................................................. 1
   1.1 Background ............................................................................................................... 1
   1.2 Fatigue Failure Theory ............................................................................................. 6
   1.3 Problems With the Prototype Car-carrier Structures Under Investigation ............. 8
   1.4 Objective .................................................................................................................. 10

2. LITERATURE REVIEW .................................................................................................... 12
   2.1 Documentation on Vehicle Dynamic Analysis ....................................................... 12
   2.2 Documentation on Field Test Methods ................................................................... 15
   2.3 Documentation on Structure Optimization ............................................................ 16
   2.4 Summary on Literature Review ............................................................................. 17
   2.5 Focus Of Research Work ......................................................................................... 18

3. INTRODUCTION TO THE FINITE ELEMENT METHOD FOR DESIGN APPLICATIONS .............................................................................................................. 19
   3.1 Features of the Finite Element Method .................................................................... 19
   3.2 Steps Working with the Finite Elements Method ................................................... 19
      3.2.1 Finite Element Model Development ................................................................ 20
      3.2.2 Computer Calculations .................................................................................... 20
      3.2.3 Visualization and Analysis of Simulation Results ........................................... 20
   3.3 LS-DYNA Overview ................................................................................................. 21
# TABLE OF CONTENTS - CONTINUED

4. FE MODEL DEVELOPMENT ................................................................. 22
   4.1 Geometry Model ........................................................................ 23
      4.1.1 Element Type and Geometry ............................................. 24
      4.1.2 Element Formulations ....................................................... 27
      4.1.3 Element Size ..................................................................... 28
      4.1.4 Accounting for Shell Thickness ......................................... 29
      4.1.5 Structure Simplifications and Omissions ......................... 30
      4.1.6 Numerical Modeling of Welding Connections .................... 33
      4.1.7 Part Name Definitions ...................................................... 36
      4.1.8 Units ................................................................................ 38
      4.1.9 Geometry Model Summary .............................................. 38
   4.2 Material Model ........................................................................... 39
      4.2.1 Material Models for Shell Elements .................................... 39
      4.2.1.1 *MAT_PLASTIC_KINEMATIC Model ............................... 39
      4.2.1.2 *MAT_RIGID ............................................................. 41
      4.2.2 Material models for Solid Elements .................................... 43
   4.3 Element Section Properties ....................................................... 44
      4.3.1 Shell Element Property ..................................................... 44
      4.3.2 Solid Element Property .................................................... 44
   4.4 Contact Algorithms .................................................................... 44
      4.4.1 Joint Pin Contact ............................................................... 46
      4.4.2 Plate Edge to Pin Contact ................................................. 47
      4.4.3 Contacts Between Two Rectangular Tubes With One in Another .............................. 50
      4.4.4 Contacts of C-Tubes inside Rectangular Tubes .................... 51
      4.4.5 Other Potential Contacts .................................................... 51
      4.4.6 Contacts for a Typically Joint Mechanism ......................... 53
   4.5 Boundary Conditions ................................................................. 54
      4.5.1 Define Velocity .................................................................. 54
      4.5.2 Apply Gravity .................................................................... 55
TABLE OF CONTENTS - CONTINUED

4.5.3 Prescribed Motion ................................................................. 55
4.6 Output Database ................................................................. 55
4.6.1 Binary Output ................................................................. 56
4.6.2 ASCII Output ................................................................. 56
  4.6.2.1 Nodout ................................................................. 56
  4.6.2.2 Glstate ................................................................. 56
4.7 LS-DYNA Miscellaneous ....................................................... 57
  4.7.1 Simulation Duration ...................................................... 57
  4.7.2 Energy Control ............................................................. 57

5. EXPERIMENTAL SET-UP AND TEST SCENARIOS ......................... 59
  5.1 Experimental Set-up .......................................................... 59
  5.2 Test Scenarios ................................................................. 63
    5.2.1 Test Scenario 1 ......................................................... 63
    5.2.2 Test Scenario 2 ......................................................... 63
    5.2.3 Test Scenario 3 ......................................................... 64
  5.3 Data Processing ............................................................... 65
    5.3.1 Calibration Factor Calculation ..................................... 65
    5.3.2 Test Data Processing .................................................. 66

6. COMPARISONS OF RESULTS BETWEEN SIMULATION AND TEST DATA .................................................. 69
  6.1 Simulation Validation ........................................................ 69
  6.2 Comparisons Between the Simulation Results and Test Data ........ 69
    6.2.1 Qualitative Comparisons ............................................. 70
    6.2.2 Quantitative Comparisons ......................................... 70
      6.2.2.1 Boundary Condition Differences ............................... 72
      6.2.2.2 Structural Configuration Difference ............................ 73
      6.2.2.3 Material Mechanical Property Discrepancies .................. 73
      6.2.2.4 Finite Element Method’s Limitations ............................ 73

vi
TABLE OF CONTENTS – CONTINUED

7. FULLY LOADED TRAILER MODELING .............................................. 75
   7.1 Loaded Vehicle model .................................................. 75
   7.2 Tie-down Chain Model .............................................. 77
   7.3 Fully Loaded Trailer Model ......................................... 80

8. ANALYSIS OF SIMULATION RESULTS ........................................ 81
   8.1 Stress Distribution Analysis ....................................... 81
   8.2 Lateral Deflection Analysis ........................................ 84

9. LOCAL STRUCTURAL MODIFICATIONS ......................................... 86
   9.1 Structural Modifications ........................................... 86
   9.2 Reduction in Stress Concentration .................................. 87
   9.3 Reduction in Lateral Deflections ................................... 88

10. GLOBAL STRUCTURAL MODIFICATIONS ...................................... 90
   10.1 Global Geometric Modifications .................................. 90
   10.2 Reduction in Stress Concentration .................................. 96
   10.3 Reduction in Lateral Deflection .................................... 98

11. CONCLUSIONS AND RECOMMENDATIONS .................................... 101
   11.1 Summaries and Conclusions ....................................... 101
       11.1.1 Conclusions Regarding Field Testing ...................... 101
       11.1.2 Conclusions on FE model Development and Validation .. 102
       11.1.3 Conclusions on Structural Improvements .................. 102
   11.2 Recommendations .................................................... 103
       11.2.1 Recommendations on the Field Testing .................... 103
       11.2.2 Recommendations on FE model Development and Validation 104
       11.2.3 Recommendations on Structural Improvements .......... 105
# TABLE OF CONTENTS – CONTINUED

REFERENCE..................................................................................................................106

APPENDIX A – Parameter Calculations In MathCAD For ASTM A 36 And
Aluminium 5052 H32........................................................................................................108

APPENDIX B – Calibration Factor Calculation Process...............................................110

APPENDIX C – Average and RMS Value Calculations
Using MathCAD For Condition 3..............................................................................113

APPENDIX D – Portion of the LS-DYNA Input File......................................................115

VITA AUCTORIS.............................................................................................................122
LIST OF FIGURES

Figure 1 - Car relocation using truck car-carriers..................................................1
Figure 2 - Configuration of commonly used car-carriers............................................2
Figure 3 - The loading/unloading of vehicles on/off the top decks............................2
Figure 4 - Fatigue fracture and breaks at the top of a car-carrier...............................4
Figure 5 - Fatigue fracture on a vertical tube............................................................4
Figure 6 - Fatigue fracture on a horizontal base tube...............................................5
Figure 7 - Generalized S-N curve for steel...............................................................6
Figure 8 - Schematic $\Delta \varepsilon - N_f$ curves showing the influence of surface cracks........7
Figure 9 - Car-carrier trailer under investigation....................................................8
Figure 10 - Three-stacked railway car-carriers.........................................................11
Figure 11 - Field test videos vs. numerical animations of the high mobility trailer.........12
Figure 12 - Air-ride suspension system.................................................................13
Figure 13 - Experimental Set-up..............................................................................14
Figure 14 - Instrumentation Set-up...........................................................................15
Figure 15 - Accelerometer locations in trailer.........................................................15
Figure 16 - Side assembly of trailer........................................................................23
Figure 17 - Comparison between solid mesh and shell mesh....................................25
Figure 18 - Triangular meshes introduced around the holes of the steel tubes............26
Figure 19 - The mesh of latch plate using triangles...............................................26
Figure 20 - Modified round bar or pin end surface mesh.........................................27
Figure 21 - Mesh of a rectangular steel tube..........................................................28
LIST OF FIGURES - CONTINUED

Figure 22 - Mesh of a small structure.................................................................29
Figure 23 - Contact surface is based on mid-surface normal projection vectors........29
Figure 24 - The 3-D drawing of tie-down device..................................................30
Figure 25 - The simplified mesh of the tie-down device........................................31
Figure 26 - Flip decks were not included in the FE model.....................................31
Figure 27 - Two contacting tubes of the trailer.....................................................32
Figure 28 - The merge of the two slide tubes......................................................32
Figure 29 - Nodes around the latch holes are constrained to the pins......................34
Figure 30 - The hinge pin constraint...................................................................34
Figure 31 - The round bar is welded to one side of the tube..................................35
Figure 32 - The flat bar constraint........................................................................35
Figure 33 - Lock System of adjustable tubes.......................................................36
Figure 34 - The FE model of the whole trailer......................................................38
Figure 35 - The end region of the tube was assigned rigid material.......................42
Figure 36 - The structure around the hole was assigned rigid material...............42
Figure 37 - The whole plate was assigned rigid material.......................................43
Figure 38 - Hinge pin connection between two decks...........................................47
Figure 39 - Selection of slave nodes for defining contact algorithms....................48
Figure 40 - Penetration in the left contact.............................................................48
Figure 41 - Contact between the latch pins and tube holes...................................49
Figure 42 - Nodes around holes are constrained to the pins...................................50
LIST OF FIGURES - CONTINUED

Figure 43 - Contact between the tubes, one inside another.........................50
Figure 44 - Contact between the rectangular tube and the C-tube....................51
Figure 45 - Potential contacts exist between the C-plate and the tube ..............52
Figure 46 - Potential contact exists between the C-section plate and tube...........52
Figure 47 - Automatic-single-surface contact was defined between these plates....53
Figure 48 - The hinge connection structures...........................................54
Figure 49 - The test car-carrier................................................................59
Figure 50 - Measured points on the trailer and the speed bump used in the field tests...60
Figure 51 - Instrumental and data-transferring set-up..................................61
Figure 52 - Data conditioner and cables wire-up...........................................62
Figure 53 - Data tape recorder....................................................................62
Figure 54 - Test scenario 1........................................................................63
Figure 55 - Test scenario 2........................................................................64
Figure 56 - Test scenario 3........................................................................64
Figure 57 - Calibration signal curves for the four channels..............................66
Figure 58 - Test data curves for test scenario 1 (both-side impact)....................67
Figure 59 - Test data curves for test scenario 2 (driver's side impact)..............67
Figure 60 - Test data curves for test scenario 3 (diagonal direction impact)........68
Figure 61 - Vertical accelerations of point (5) for test condition 1....................71
Figure 62 - Lateral accelerations of point (4) for test condition 1.....................71
Figure 63 - Lateral accelerations of point (4) for test condition 2.....................72
LIST OF FIGURES - CONTINUED

Figure 64 - Lateral accelerations of point (4) for test condition 3........................................72
Figure 65 - The FE model of a loaded vehicle.................................................................76
Figure 66 - Bottom view of Ratchet tube and chain.........................................................78
Figure 67 - The tie-down chain model.............................................................................78
Figure 68 - Unrealistic deformation due to stress concentration......................................79
Figure 69 - Isometric view of the FE model of the loaded trailer..................................80
Figure 70 - Side view of the FE model of the loaded trailer..........................................80
Figure 71 - Illustration of stress and/or strain concentration regions in the FE model....82
Figure 72 - Stress concentration at the front tow bar region..........................................82
Figure 73 - Stress concentration around the hole of the suspension frame.......................83
Figure 74 - Stress concentration on the support adjustable tube of the top decks........83
Figure 75 - The maximum lateral deflections for the three simulation conditions.........84
Figure 76 - Back view of the deformation of slender C-plate..........................................85
Figure 77 - Local geometric modification locations in the trailer..................................86
Figure 78 - Bottom views of the modifications at the front tow bar region.....................87
Figure 79 - Side view of the globally modified model.....................................................91
Figure 80 - Isotropic view of the globally modified model.............................................91
Figure 81 - Structural improvements for the suspension frame......................................92
Figure 82 - Support configuration of the front top deck...............................................93
Figure 83 - The front support mechanism of the top front deck....................................94
Figure 84 - The rear support scheme of the top front deck..........................................95
LIST OF FIGURES - CONTINUED

Figure 85 - The front support mechanism of the rear top deck.........................95
Figure 86 - The maximum lateral deflection curves for Simulation Condition 1........98
Figure 87 - The maximum lateral deflection curves for Simulation Condition 2........99
Figure 88 - The maximum lateral deflection curves for Simulation Condition 3........99
# LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 1</td>
<td>The Mass and Dimensions of the Trailer</td>
<td>22</td>
</tr>
<tr>
<td>Table 2</td>
<td>Part Name and Corresponding Properties For Shell Element Parts</td>
<td>37</td>
</tr>
<tr>
<td>Table 3</td>
<td>Mechanical Properties for Plates or tubes</td>
<td>41</td>
</tr>
<tr>
<td>Table 4</td>
<td>Comparison of average and RMS values of acceleration</td>
<td>74</td>
</tr>
<tr>
<td>Table 5</td>
<td>The maximum effective plastic strain/effective stress at stress/strain concentration regions for different simulation conditions</td>
<td>84</td>
</tr>
<tr>
<td>Table 6</td>
<td>Maximum effective plastic strain/effective stress comparison for simulation condition 1, before and after local modifications</td>
<td>87</td>
</tr>
<tr>
<td>Table 7</td>
<td>Maximum effective plastic strain/effective stress comparison for simulation condition 2, before and after local modifications</td>
<td>88</td>
</tr>
<tr>
<td>Table 8</td>
<td>Maximum effective plastic strain/effective stress comparison for simulation condition 3, before and after local modifications</td>
<td>88</td>
</tr>
<tr>
<td>Table 9</td>
<td>Maximum lateral deflection comparisons between original model and locally modified model</td>
<td>89</td>
</tr>
<tr>
<td>Table 10</td>
<td>Maximum effective plastic strain and/or maximum effective stress comparisons among the original model, locally modified model, and globally modified model, for simulation condition 1</td>
<td>96</td>
</tr>
<tr>
<td>Table 11</td>
<td>Maximum effective plastic strain and/or maximum effective stress comparisons among the original model, locally modified model, and globally modified model, for simulation condition 2</td>
<td>97</td>
</tr>
<tr>
<td>Table 12</td>
<td>Maximum effective plastic strain and/or maximum effective stress comparisons among the original model, locally modified model, and globally modified model, for simulation condition 3</td>
<td>97</td>
</tr>
<tr>
<td>Table 13</td>
<td>Maximum lateral deflection comparisons among the original model, locally modified model, and globally modified model</td>
<td>100</td>
</tr>
</tbody>
</table>
GLOSSARY OF ABBREVIATIONS

ASTM – American Society for Testing and Materials
CPU – Central Processing Unit
DADS – Dynamic Analysis and Design System
DoF – Degrees of Freedom
FE – Finite Element
FEA – Finite Element Analysis
FEMB – Finite Element Model Builder
FM – Frequency Modulation
HMMWV – High Mobility Multipurpose Wheeled Vehicle
HMT – High Mobility Trailer
RMS – Root Mean Square
SAE – Society of Automotive Engineers
SUV – Sport Utility Vehicle
1. INTRODUCTION

1.1 Background

The delivery of new vehicles to automobile dealers is one major task for the automotive industry. Usually vehicle manufacturers distribute their vehicles to dealers by sea, rail or truck. Sometimes small carriers that haul only two or three vehicles are used. Very large vehicles, such as campers, boats, small dump trucks, and some very large vans are transported by flatbed trailers. Occasionally, antique or very high value vehicles are transported in enclosed carriers, which are generally much more expensive than other transit methods. The more effective and less expensive transporting mode for passenger vehicles is to use dedicated car-carriers. With the development of highway systems in North America, shipment by means of haul-away (truck) car-carriers is a very common method for transporting vehicles over land. Figure 1 illustrates the common transporting mode of vehicles using typical car-carriers.

Figure 1. Car relocation using a truck car-carrier.
These large carriers are designed specifically for transporting eight to eleven passenger vehicles, each on its own support deck. In order to maximize the number of loaded vehicles, car-carriers are usually built in the form shown in Figure 2, which depicts a headramp-trailer combination.

![Figure 2. Configuration of commonly used car-carriers.](image)

The structural members of a car-carrier undergo significant impacts and shocks during loading/unloading of vehicles, or when the vehicles are in transit. Figure 3 shows the loading/unloading process of vehicles.

![Figure 3. The loading/unloading of vehicles on/off the top decks.](image)
Usually shocks and vibrations originate from two sources in a truck-trailer system [1].

- External sources, such as road or surface irregularities, braking, and forward acceleration;
- Internal sources from the vehicle itself, such as engine vibration, drive mechanisms, and wheel imbalance.

For car-carriers, significant shocks and vibrations are induced by external sources due to the large weight of the loaded vehicles and harsh road conditions. The impact effects of the internal sources to the car-carrier structures are negligible compared to the external sources and will not be considered in this research.

In recent years in order to protect the loaded vehicles and to extend the service life of the car-carrier, air-ride suspension systems have replaced the conventional steel-leaf-spring suspension systems in the car-carrier to isolate the vehicle for shocks due to external sources. It is commonly thought that an air-ride suspension will give a smoother ride than a leaf-spring system. However, breaks and failures still occur in the car-carrier structures during normal service. Caruso and Silver [2] concluded that the worst ride occurred on interstate highways at high speeds. With more car-carriers transporting on highways at high speeds, car-carrier structural fractures or breaks are more common. Figure 4 through Figure 6 show a few examples of structural fracture.
Figure 4. Fatigue fracture and break at the top of a car-carrier.

Figure 5. Fatigue fracture on a vertical tube.
1.2 Theory of Fatigue Failure

Most failures in vehicles are due to time-varying loads compounded with the effects of corrosion. These failures typically occur at stress levels lower than the yield strengths of the materials. Fatigue failures always initiate at a crack site. The crack may have been present in the material since its manufacture, or it may have developed over time due to cyclic straining. Virtually all structural members contain discontinuities, ranging from microscopic (< 0.25 mm) to macroscopic introduced in the manufacturing or fabricating process. Fatigue cracks generally start at a notch or other stress/strain concentrations. As the stresses at the notch oscillate, local yielding may occur due to stress/strain concentration, even though the nominal stress in the section is well below the yield strength of the material. Thus, it is critical that dynamically loaded parts be designed to minimize stress/strain concentrations.
Generally, there are two main variables influencing fatigue failure: cyclic stress/strain and flaw or crack size. In addition, conditions may result in an interaction of the fatigue process with other time-dependent failure processes, such as corrosion and creep.

According to fatigue theory for steel structures, the life of a structure increases as the repeated applied stress/strain decreases, for stresses between the endurance limit and the ultimate strength [3]. Figure 7 illustrates a typical curve of $S_f/S_u$ versus the number of applied loading cycles to failure for steel alloys, in which $S_f$ represents the fatigue stress amplitude and $S_u$ represents the ultimate strength of the material.

![Figure 7. Generalized S-N curve for steel [3].](image)

Figure 8 illustrates the relationship between fatigue life and strain, and the sensitivity of fatigue life to initial crack size ($a_c$) [4]. In the low cycle fatigue region ($N_f < 10^3$ cycles) an increase in crack size has much less effect on life than in the high cycle fatigue region ($N_f > 10^5$ cycles).
Figure 8. Schematic $\Delta e - N_f$ curves showing the influence of surface cracks [4].

In considering fatigue, the designer usually adopts one of the two design philosophies: ‘safe life’ and ‘damage tolerance’ [4]. No matter which philosophy is selected, decreasing the severity of stress/strain concentrations is an effective approach to increasing the fatigue life of structures. In this investigation, the car-carrier structures’ durability will be investigated by evaluating stress and strain levels; pre-existing or propagating cracks and environmental conditions (corrosion, creep, temperature) will not be considered.

1.3 Concerns With the Prototype Car-Carrier Structures

This project investigated the dynamic behaviour of a prototype car-carrier structure. The mechanical drawings of the trailer of the prototype car-carrier were provided by StarTrans, a car-carrier manufacturing company located in Amherstburg, Ontario, Canada. Figure 9 shows the structure of the trailer of the car-carrier.
Figure 9. The trailer of a car-carrier under investigation.

When six heavy sport utility vehicles (SUVs) are loaded on the trailer, large lateral deflections will occur during harsh driving conditions, especially on the top decks. This loading condition may cause damage to the loaded vehicles, and premature breaks and failures occur on the trailer.

In order to minimize the impact on car-carriers’ structures and to avoid possible damage to loaded vehicles, shipping manuals have been prepared for guidance in loading/unloading of vehicles. One requirement is that vehicles must be driven at 8 km/h or less during loading and unloading [5]. Carefully abiding by the requirements can minimize the possible impact damage to carriers’ structures and to loaded vehicles. However, the impact due to harsh driving conditions cannot be avoided. Almost all the breaks or fractures of car-carriers’ structures arise during transit.
The rear trailer of the car-carrier experiences considerably more fractures and welded point breaks due to its inefficient structural configuration, fewer support tubes, and frequent loading/unloading impacts. Therefore, this research will focus on the trailer portion of the car-carrier.

1.4 Research Objective

Trucks just like cars and aircraft are designed for a limited lifetime. Truck manufacturers have to ensure that no essential components or parts will fail during normal service. Assuring this durability is a very complicated process, especially for trucks. For car-carrier design, an “experimental” optimization is not practical because of many possible car-carrier configurations due to the combinations of different components.

The overall goal of this project is to develop an accurate finite element (FE) model of the trailer and validate it using experimental tests so that stress risers can be identified and eliminated after geometric modifications. The experimental field tests will also contribute to a better understanding of the dynamic responses of the trailer structure. Lateral deflections of the trailer will also be investigated using the FE method. These modifications may help reach the maximum load capacity of the trailer and extend its lifetime. This project will provide new analytical procedures and also an experimental method to car-carrier design.
2. LITERATURE REVIEW

A thorough literature search has been conducted trying to find topics concerning vehicle dynamic analysis, fatigue failure analysis, vibration data collecting methods, structural optimization, and car-carrier design. The following sections summarize the related research.

2.1 Documentation On Vehicle Dynamic Analysis

The automotive industry uses several simulation methods for the description of the driving behaviour of vehicles. Although structural differences exist between the trailer under investigation in this research and the following cited structures, the methodologies used in the above-mentioned research are applicable.

Banthis et al. [6] developed a FE model of a three-stacked railroad hauling car carrier, as illustrated in Figure 10, in order to analyze the dynamic response of the structure. The finite element method was used for the strength, stability, and modal analysis of the system. The conceptual design of the containers and their components was used as the basis for constructing a finite element model.

A finite element model was developed using the configuration of three containers stacked on top of each other with three full size automobiles in each container. The frame members of the containers and the rails running along the length of the floor of the containers were modeled as two-node elastic beam elements. The automobiles were modelled as lumped masses. To distribute their weight over the floor, these lumped masses were attached to the floor through a pyramidal structure of very stiff beam
elements. Wall, roof, and floor composite panels were modelled as uniformly thick shell elements with an overall stiffness equivalent to that of the actual ribbed panels. To verify the analysis results and to gather field data, a test program was conducted. In the test program, two fully triple-stacked container systems were loaded with automobiles. One of the stacked systems was instrumented with nine accelerometers placed in different locations. The test and analytical results were compared to verify the finite element model.

Figure 10. Three-stacked railway car-carriers [6].

Letherwood and Wehage [7] published a paper describing a computer-based modelling, simulation, and validation effort to aid in the development, testing and
procurement of a lightweight, High Mobility Trailer (HMT) towed by the High Mobility Multipurpose Wheeled Vehicle (HMMWV), which is used by the military. The analyses employed the Dynamic Analysis and Design System (DADS) computer-based methodology to predict vehicle and system performance. Simulations were performed on obstacle avoidance (lane change) manoeuvres, bumps, potholes, and country courses. In the study, instrumented field tests were conducted and the data was used to develop and validate the models. Figure 11 shows the comparison of the field test videos and the computer animations when the trailer negotiated a pothole and a speed bump.

![Numerical Animation Images](image1.png) ![Experimental Test Images](image2.png)

Figure 11. Field test videos versus numerical animations of the high mobility trailer [7].

### 2.2 Documentation On Field Test Methods

The suspension system of the trailer under investigation is an air-ride suspension system as shown in Figure 12. An air ride system possesses some advantages over a leaf spring suspension system such as a smooth ride, high roll stability and articulation, equal
weight distribution, low suspension maintenance, longer equipment life, and decreased cargo damage. The energy transmitted by the air suspension is about 48% less than that transmitted by leaf springs and the vibration amplitudes for the air suspension are approximately 60% less than that of a leaf spring system [8].

![Diagram of air-ride suspension system](image)

**Figure 12. Air-ride suspension system.**

Pierce, Paul and Burgess [9] concluded that conventional air-ride suspension showed an oscillatory response near 2 Hz. At frequencies above 50 Hz, measured structure responses were very small and had little effect in damage to the structures. Suspension geometry and air spring design work together to produce a natural frequency below 2 Hz. Typical mechanical suspensions have natural frequencies that range from 2-5 Hz, depending on payload.

The vibrations of an air ride suspension depend on a number of parameters, which include load, velocity, spring stiffness, damping coefficient, and road conditions. Using a theoretical approach, it is very difficult to accurately evaluate the vibration of an air ride suspension system.
There has been significant work conducted in the past not only on the vibration level comparison between leaf spring suspension and air-ride suspension, but also about the data collecting methodologies.

Lozano-Guzman and Tapia-Armas [8] conducted a comparison of the vibration levels between a leaf spring and an air-ride. Data was acquired from accelerometers and stored through a microprocessor-based data-logger. After processing the signals, a set of plots representing the vibration amplitude and frequency as a function of time was obtained. Figure 13 shows the experimental set-up. The researchers concluded that the vibration amplitudes for the air suspension are about 60% less than for that of leaf spring system.

![Experimental Set-up](image)

Figure 13. Experimental Set-up [8].

Singh and Marcondes [10] studied the vibration levels in commercial truck shipments as a function of suspension and payload. To monitor the vibration levels in the different trailer systems, piezoelectric accelerometers were attached to the floor of the trailer. A total of six trailers (four with leaf spring, one with air-ride suspension, and one
panel van) were monitored for a total of over 16,000 km. Various locations were instrumented in both the front and the rear of each trailer. In addition, accelerometers also measured the lateral and longitudinal vibrations in these trailers. Microdot cables were used to connect the accelerometers to signal conditioners. A high performance frequency modulation (FM) recorder was used to record the output from the signal conditioners. Figure 14 shows the instrumentation set-up. Figure 15 illustrates the positioning of accelerometers in a typical trailer.

![Diagram](image1)

Figure 14. Instrumentation Set-up [10].

![Diagram](image2)

Figure 15. Accelerometer locations in a trailer [10].
2.3 Documentation On Structural Optimization

Nigel [11] investigated a combination of testing and analysis of a waist rail of a bus body that had experienced premature failure. A finite element model of the complete vehicle was used as the basis for the correlation between the experimental test and numerical model analysis. A good correlation was obtained with certain types of manoeuvres and this was then used to predict a life for the vehicle based upon structural failure. A detailed FE model of the joint area of the rail showed a high stress concentration effect, with the highest stress being at the position where the failure occurred. A revised joint was developed which reduced stress concentration effects.

Langenhove and Maasdam [12] used the global/local model concept to improve the durability of large complicated vehicle structures. The concept of global/local models allows for the efficient calculation of the effect of the dynamic response of a system to external inputs (loading conditions) and consequently to the effect of this loading on local stress/strain at critical areas. Firstly, local geometric modifications can be made to reduce stress concentrations without changing the overall structural behaviour. A second type of modification can include the modification on the global model: this procedure can modify the overall dynamic behaviour of the vehicle.

The global/local model methods predict not only the overall dynamic behaviour of the structure, which can be described by its modal behaviour, but also gives an accurate prediction of the local dynamic stress/strain levels.
2.4 Summary of Past Research

Based upon the literature reviewed, several observations are made in the investigation of the trailer of a prototype car-carrier.

1. The experimental test methods mentioned in the above literature will be referred to in this research. Accelerometers and other previously mentioned instruments will be used to collect vibration data at critical positions of the trailer. Speed bumps or deep potholes will be used to induce manageable vibrations.

2. As the air ride suspension system’s vibration frequency is as low as 0.5 Hz, accelerometers with sensitivity as low as 0.5 Hz have to be chosen for the field tests in order to capture the vibration data.

3. Dynamic finite element analysis (FEA) and experimental methods are excellent approaches to improving structure in reducing stress concentrations and extending fatigue life. In model development process, lumped massed will be used to model the loaded vehicles on the trailer.

4. The previously cited global/local structural modification methods are very informative and will be utilized in this research to improve the trailer structure.

5. Little or no research has been found in FE car-carrier structural analysis and enhancement.

The need to fully understand how the car-carrier structure will perform under driving conditions very much exists. This will enable engineers to check the stress/strain concentration regions of the trailer and find the appropriate method to strengthen the structure. 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.
2.5 Focus of Research Work

The work associated with this thesis focuses on the following areas:

1. Only the trailer portion of the prototype car-carryer will be investigated and only the external sources to the structural impact will be considered.

2. A detailed FE model of the trailer under investigation will be developed.

3. Experimental field tests, using a trailer with a similar configuration to the prototype trailer under investigation, will be conducted to gather vibration data for the inputs to the FE models and as means of comparison of dynamic responses between the test trailer and the FE model of the unloaded trailer under investigation. The experimental tests will also provide knowledge of vibration over car-carriers when subjected to harsh driving conditions.

4. After comparison between the experimental and numerical responses, the FE model of the trailer will be updated with loaded vehicles and simulations will be conducted under identical test conditions. The numerical analyses will focus on the stress/strain profile and lateral deflections of the trailer.

5. Local and global structural modifications will be considered to decrease the severities of stress/strain concentrations and lateral deflections of the trailer.
3. INTRODUCTION TO THE FINITE ELEMENT METHOD FOR DESIGN APPLICATIONS

The finite element method is designed to approximate the solutions of partial differential equations with boundary conditions. Together with the increasing availability of large capacity, high-speed digital computers, the FE techniques have been widely applied to the prediction of structural performances.

3.1 Features of the Finite Element Method

Computer-aided analysis of structures using the FE method can predict how a prototype will respond to real-world events, thus minimizing the time spent in design and the money spent on experimental testing.

It is impossible to obtain the detailed stress data in a specific area of a structure by experimental testing. By using computer-aided FE simulation, detailed stress values in a specific region of the structure can be obtained from the simulation results, thus providing more information for structural design.

3.2 Steps Involved in Working with the Finite Element Method

Employing the FE method requires that three steps be followed: model development, calculations, and visualization and analysis of simulation results. Here only a brief overview is provided.
3.2.1 Finite Element Model Development

This is done using a pre-processor. First, a model of the object under study, describing the geometry, needs to be created. This can be done in a variety of ways, but is typically completed using a CAD system. During the second step, the geometric model is meshed or divided into small discrete elements. The user needs to provide information on how to divide the model into a mesh. The degree of discretization depends generally on the required accuracy of the results. A finer mesh can yield better results, but will also incur more computing time. In the third step additional characteristics of the model are defined. Material properties, additional constraints, prescribed motions, and external loads are prescribed to the model. In this project, all pre-processing and mesh development was completed using Finite Element Model Builder (FEMB) version 27.

3.2.2 Computer Calculations

After the model is developed, the input file is generated from the pre-processor. A computer package solver takes care of all computations of the partial differential equations. In this research, LS-DYNA version 960 (revision 1488) was used to do all the calculations.

3.2.3 Visualization and Analysis of Simulation Results

Finally, during the last stage, the results of the calculations can be visualized and analyzed. This is normally done by manually operating a separate application (post-processor), which can visualize the results from the calculations in several ways. This
analysis can trigger an iterative process where one or more steps are refined further. In this project, all post-processing was completed using LS-POST version 2.0.

3.3 LS-DYNA Review

There are quite a few commercial finite element method packages in the market, such as ANSYS, MSC/NASTRAN, ABAQUS, LS-DYNA, I-DEAS, CATIA, and ALGOR. LS-DYNA, developed by a team lead by Hallquist (1976), is a general-purpose, explicit finite element program used to analyze the non-linear dynamic response of three-dimensional structures. The main solution methodology is based on explicit time integration, which is especially suitable for the simulation of a short time period and dynamic non-linear structural response [13]. This explicit method is very efficient and accurate to simulate complex dynamic non-linear responses.

LS-DYNA includes large material and element libraries, many contact algorithms, and a wide range of applications. It is capable of simulating complex real world problems, and is widely accepted as the premier analysis software package for today’s most challenging engineering applications. Using LS-DYNA, automotive companies and their suppliers can test vehicle designs without having to tool or experimentally test a prototype, thus saving time and expense.
4. FINITE ELEMENT MODEL DEVELOPMENT

The trailer under investigation can load seven passenger cars (at a maximum) with eight adjustable loading decks. The mass and dimensions of the trailer under investigation is listed in Table 1.

Table 1. The Mass and Dimensions of the Trailer.

<table>
<thead>
<tr>
<th>Trailer Mass (Unloaded)</th>
<th>Length</th>
<th>Width</th>
<th>Height</th>
</tr>
</thead>
<tbody>
<tr>
<td>6,000 kg</td>
<td>14.5 m</td>
<td>2.6 m</td>
<td>2.6 m</td>
</tr>
</tbody>
</table>

The whole trailer, as illustrated in Figure 9, can be decomposed into thirteen separate parts:

- The side assemblies on both sides;
- The rear main assembly;
- The hitch assembly;
- The axle mount assembly;
- The front bottom deck;
- The middle bottom deck 1;
- The middle bottom deck 2;
- The rear bottom deck;
- The front top deck;
- The middle top deck;
- The rear top deck;
- The front middle deck.
Figure 16 shows the driver’s (left) side assembly of the trailer. There are two sides of this assembly: the driver’s side assembly and the passenger’s side assembly. The structure of both sides of the trailer is basically identical.

![Figure 16. Side assembly of the trailer.](image)

The draft design of a prototype car-carrier has been completed by StarTrans. The mechanical drawings of the trailer part, both hard copy and electronic copy were used as reference in this investigation. Initially, an attempt was made to convert the electronic CAD data of the trailer assembly to IGES format in AutoCAD R14, then to import the IGES format data into FEMB in order to discretize the original CAD trailer structure. Errors in the electronic CAD data did not permit successful import of the trailer structure into FEMB. Hence, the complete trailer geometry was created using CAD geometry commands within FEMB.

### 4.1 Geometry Model Development

The main structures used to construct the trailer are rectangular hollow structural section tubes, plates, and round pins. The geometry model of the prototype trailer was
created and meshed in FEMB according to the dimensions specified in the provided drawings.

An appropriate mesh is critical for reliable simulation results. While a mesh that is too coarse will not capture the realistic information, a mesh that is too fine will unnecessarily increase the CPU cost. Due to the large size of the vehicle carrier, an emphasis during the model development was placed on creating a numerical model that minimized the number of elements while ensuring acceptable simulation results. Hence a guideline, which is outlined below, was utilized in the development of the FE model of the trailer.

4.1.1 Element Type and Geometry

Choosing appropriate element types to build the model was essential for reducing the number of elements. Shell elements were used to mesh tubes and plates, and solid elements were used to mesh round pins and bars.

The tubes and plates could have been modeled using eight-node brick elements. Since brick elements have only one integration point, they would have to be layered at least three deep to capture a stress distribution due to bending, thus substantially increasing the number of elements needed. Another consideration is the ratio of maximum to minimum lengths of element sides for a brick element. This aspect ratio is best kept less than four for reliable accuracy. Using three elements through the thickness for a given plate thickness will severely reduce the in-plane dimensions of the element, thereby requiring a very large number of small elements to be used. The formulation of the shell element does not constrain the in-plane dimensions of the element regardless of the thickness. In this model, the aspect ratio of the shell element is as large as eight while
reliable accuracy is still ensured. Therefore, the shell element will permit fewer elements to be used when compared to the brick element [14]. The comparison between shell mesh and solid mesh is shown in Figure 17.

![Diagram showing comparison between solid mesh and shell mesh](image)

**Figure 17.** Comparison between solid mesh and shell mesh.

Another advantage of the shell element is the time step computed by LS-DYNA. For the brick element, the time step has a linear dependence on the minimum side length, which in the present case would be the thickness of the structure. The time step captured for the shell element has a much weaker dependence on the thickness, thus allowing large time steps to be used for a given element thickness. If wave propagation through the thickness of the structure is not of major concern, then the shell element can be used with greater efficiency and substantial savings in cost over a comparable model with brick elements.

To improve accuracy and efficiency, quadrilateral elements were preferred for two-dimensional meshes and hexahedral elements for three-dimensional meshes. This preference is clear in structural analysis and seems to also hold for other engineering
disciplines [15]. However, it is also generally accepted that triangular elements can provide acceptable accuracy and convergence characteristics [16]. In the mesh process, an attempt was made to use quadrilateral and hexahedral meshing as much as possible, as they yield better simulation results [16]. In order to reduce the number of elements, triangular elements were also used in some structures as illustrated in Figures 18 and 19. It is worth noting that only 2.4% of the total number of shell elements was triangular.

Figure 18. Triangular meshes introduced around the holes of the steel tubes.

Figure 19. The mesh of the latch plate using triangles.
For some round bars or pins, fine mesh along the cylindrical surface is needed due to contact with other items in the trailer. Otherwise, a coarse mesh will make cylindrical surfaces very rough and cannot emulate the actual interaction. In this case, 32 elements were used along the circular surface to make the cylindrical surface smoother. In order to decrease the number of solid elements, some wedge elements were utilized as shown in Figure 20. Less than 4.5% of the total number of solid elements was wedge elements.

![Modified end surface mesh of the round bar or pin.](image)

Figure 20. Modified end surface mesh of the round bar or pin.

### 4.1.2 Element Formulation

In order to further decrease the number of elements, fully integrated shell elements with $2\times2$ gaussian quadrature were used for all shell elements (Hughes-Liu shell element formulation in LS-DYNA). The Belytschko-Tsay shell element (under-integrated) might have been used to model the rectangular tubes. In this case, four elements would have to be used to capture the stress distribution in the transverse direction of the rectangular tubes. For fully integrated (Hughes-Liu) shell element, two shell elements in the transverse direction will ensure acceptable simulation results. In this project, most of the rectangular tubes were meshed with two elements in the transverse direction as shown in
Figure 21. Although fully integrated shell element formulation 16 works faster in general cases, somehow in this investigation, more simulation time was needed for formulation 16 than formulation 1. A constant stress solid element formulation (formulation #1 in LS-DYNA) was used for all solid elements.

![Mesh of a rectangular steel tube](image)

Figure 21. Mesh of a rectangular steel tube.

4.1.3 Element Size

For some small structures, excessively large elements will not accurately represent the real structure, such as an entity shown in Figure 22. There is a hole on the rectangular tube. A minimum of two elements in the dimensional direction is required in order to capture bending stress. The minimum length of these elements is only 4.75 mm. In this investigation, all shell elements whose minimum side was less than 5 mm were assigned rigid material properties in order to decrease computer calculation time.
4.1.4 Accounting for the Shell Thickness

When creating shell elements, the shell thickness effects have to be taken into account. In the treatment of thickness, LS-DYNA projects both the slave and master surfaces based on the mid-surface normal projection vectors as shown in Figure 23. The surface, therefore, must be offset by an amount equal to \( \frac{1}{2} \) its total thickness. This approach allows LS-DYNA to check the node numbering of the segments automatically to ensure that the shells are properly oriented [17]. In this FE model, all shell elements were meshed on the physical mid-surface of the structures. Thus, the initial contact interpenetrations will be minimized.

Figure 23. Contact surface is based on mid-surface normal projection vectors [17].
4.1.5 Structure Simplifications

Some auxiliary parts of the trailer were not included in the model since these parts do not experience much force when the trailer is in transit. Examples include the hydraulic cylinders, control panels, unused support decks, and springs.

The tie-down device as illustrated in Figure 24 was simplified to decrease model size. The pin for constraining the chain, the round bars on the ratchet tube, and the ratchet wheel were omitted. Figure 25 illustrates the simplified model of the tie-down device. The ratchet tube was fastened to the support plates using spotwelds, as when the vehicles are tied down on the deck, the tube does not rotate. The top plate (contacting with vehicle tire) was meshed as a flat plate, neglecting the non-skid holes on the plate. In the whole model of the trailer, all non-skid plates were meshed as flat plates.

Figure 24. The 3-D drawing of a tie-down device.
Figure 25. The simplified model of a tie-down device.

The flip decks as shown in Figure 26, the attached black pipe, and pins were not created and meshed in the FE model as these decks are used as passages only when vehicles are driven from one support deck to another. When vehicles are tied down, these decks are not used to support the vehicles. If these decks were created in the FE model, the number of elements would be greatly increased.

Figure 26. Flip decks were not included in the FE model.
Some items of the trailer were combined to simplify the FE model. For example, Figure 27 illustrates the CAD geometry of a slide tube and a constraint tube. These entities are combined within the FE model, as illustrated in Figure 28 for model simplifications.

![Slide tube and constraint tube](image)

Figure 27. Two contacting tubes of the trailer.

![Mesh model](image)

Figure 28. The merge of the two tubes in the FE model.
Another significant simplification was that chamfers and fillets of the structures were not considered in the FE model. All corners of the trailer structures were treated as sharp angles in the FE model. This simplification will expedite the model building process and greatly reduce the number of elements. The stress concentration effects due to this simplification would be small since these chamfers or fillets are very small compared to the geometries of the structures.

4.1.6 Numerical Modelling of Welded Points

Welding is used to fasten together different components of the trailer. In the FE model, spotwelds were used to represent the welding connections between shell element structures. In LS-DYNA, *CONSTRANGED_SPOTWELD is used to define massless spot welds between non-contiguous nodal pairs. The spot weld is a rigid beam that connects the nodal points of the nodal pairs; thus nodal rotations and displacements are coupled. No failure criterion was specified in the LS-DYNA algorithm.

For welding points between solid element and shell element structures, spotwelds could not be used, as all solid elements were defined rigid in the FE model and cannot transfer rotational displacements. The LS-DYNA code does not allow the spotweld to connect the shell to a rigid solid. In this case, *CONSTRANGED_EXTRA_NODES was used to constrain extra nodes from shell elements to rigid bodies. Extra nodes for rigid bodies may be placed anywhere, even outside the body, and they are assumed to be part of the rigid body. In the trailer, there are structures where pins or round bars are welded to plates around it. Figures 29 to 33 are a few examples where extra nodes in shell elements were constrained to rigid bodies. In the whole FE model, about eighty node sets were constrained to rigid pins.
Figure 29. Nodes around the latch holes are constrained to the pins.

Figure 30. The hinge pin constraint. Nodes around the pin, including all nodes on the round pipes, nodes on the top of the latch plate, and bottom nodes on one side of the rectangular tube, are constrained to the rigid pin.
Figure 31. Modelling of round bar to tube connection. The round bar is welded to one side of the tube. Extra nodes on the welded side of the tube were constrained to the rigid round bar in the FE model.

Figure 32. The flat bar constraint. The flat bar is connected to the main frame (rectangular tube) through spotwelds. The round bar is connected to the flat bar by constraining extra nodes on the flat bar to the round bar.
Figure 33. Lock system of adjustable tubes. On the left is a CAD image of the actual
lock system for the adjustable tubes. On the right is the FE model of the lock
system. Nodes around the holes of the rectangular tubes are constrained to the
pin. Therefore, the pin constrains the relative movement between the tubes.
This constraint method can simplify the model greatly.

4.1.7 Part Name Arrangement

The trailer is composed of approximately one thousand separate components. If
each entity were given a part name in the FE model, it would be very arduous to define
material and section properties for each part. In order to simplify the model building
process, plate or tube components with same element type, element property, and
material property, were given one part name. Table 2 illustrates the part name
arrangement with the corresponding material properties and element section properties
for tubes and plates. In the table, “LG” represents “large”, “HSS” represents “hollow
structural section”, “HRS” represents “hot-rolled steel”, and “SCH” represents
“schedule”.

As all solid elements were defined rigid, each pin or round bar has to be given a
unique part name (separate part ID) to move independently. This requirement arises from
the fact that LS-DYNA internally computes the six degrees-of-freedom for each rigid body (rigid material or set of merged materials), and if disjoint groups of rigid elements use the same part ID, the disjoint groups will move together as one rigid body [18]. It is also convenient to define contact or constraint algorithms using unique part names for different rigid parts. Approximately 200 of the 231 total parts within the trailer FE mode were assigned rigid material properties.

Table 2. Part Name and Corresponding Properties For Shell Element Parts.

<table>
<thead>
<tr>
<th>Part Name</th>
<th>Structure</th>
<th>Thickness (mm)</th>
<th>Material name given in the drawings</th>
<th>Corresponding ASTM material</th>
<th>Material used in the FE model</th>
</tr>
</thead>
<tbody>
<tr>
<td>3/16&gt;18</td>
<td>Rectangle Tube</td>
<td>4.67</td>
<td>LG HSS Tube</td>
<td>A 500 grade B &amp; C</td>
<td>A 500 grade B</td>
</tr>
<tr>
<td>1/4&gt;18</td>
<td>Rectangle Tube</td>
<td>6.35</td>
<td>LG HSS Tube</td>
<td>A 500 grade B &amp; C</td>
<td>A 500 grade B</td>
</tr>
<tr>
<td>1/2&gt;18</td>
<td>Rectangle Tube</td>
<td>12.70</td>
<td>LG HSS Tube</td>
<td>A 500 grade B &amp; C</td>
<td>A 500 grade B</td>
</tr>
<tr>
<td>3/8&gt;18</td>
<td>Rectangle Tube</td>
<td>9.53</td>
<td>LG HSS Tube</td>
<td>A 500 grade B &amp; C</td>
<td>A 500 grade B</td>
</tr>
<tr>
<td>180&gt;18</td>
<td>Rectangle Tube</td>
<td>4.57</td>
<td>LG HSS Tube</td>
<td>A 500 grade B &amp; C</td>
<td>A 500 grade B</td>
</tr>
<tr>
<td>179&lt;18</td>
<td>Rectangle Tube</td>
<td>4.55</td>
<td>LG HSS Tube</td>
<td>A 500 grade B &amp; C</td>
<td>A 500 grade C</td>
</tr>
<tr>
<td>134&lt;18</td>
<td>Rectangle Tube</td>
<td>3.40</td>
<td>LG HSS Tube</td>
<td>A 500 grade B &amp; C</td>
<td>A 500 grade C</td>
</tr>
<tr>
<td>1/8&lt;18</td>
<td>Rectangle Tube</td>
<td>3.18</td>
<td>LG HSS Tube</td>
<td>A 500 grade B &amp; C</td>
<td>A 500 grade C</td>
</tr>
<tr>
<td>P1/4</td>
<td>Plate</td>
<td>6.35</td>
<td>HRS Plate</td>
<td>A 36</td>
<td>A 36</td>
</tr>
<tr>
<td>P1/8</td>
<td>Plate</td>
<td>3.18</td>
<td>HRS Plate</td>
<td>A 36</td>
<td>A 36</td>
</tr>
<tr>
<td>P3/8</td>
<td>Plate</td>
<td>9.53</td>
<td>HRS Plate</td>
<td>A 36</td>
<td>A 36</td>
</tr>
<tr>
<td>P3/16</td>
<td>Plate</td>
<td>4.67</td>
<td>HRS Plate</td>
<td>A 36</td>
<td>A 36</td>
</tr>
<tr>
<td>P134</td>
<td>Plate</td>
<td>3.40</td>
<td>HRS Plate</td>
<td>A 36</td>
<td>A 36</td>
</tr>
<tr>
<td>P5/16</td>
<td>Flat Bar</td>
<td>7.94</td>
<td>HRS Plate</td>
<td>A 36</td>
<td>A 36</td>
</tr>
<tr>
<td>P1/2</td>
<td>Plate</td>
<td>12.70</td>
<td>HRS Plate</td>
<td>A 36</td>
<td>A 36</td>
</tr>
<tr>
<td>P75</td>
<td>Plate</td>
<td>19.05</td>
<td>HRS Plate</td>
<td>A 36</td>
<td>A 36</td>
</tr>
<tr>
<td>P1</td>
<td>Plate</td>
<td>25.40</td>
<td>HRS Plate</td>
<td>A 36</td>
<td>A 36</td>
</tr>
<tr>
<td>P120</td>
<td>Plate</td>
<td>3.05</td>
<td>HRS Plate</td>
<td>A 36</td>
<td>A 36</td>
</tr>
<tr>
<td>P2638</td>
<td>Plate</td>
<td>6.70</td>
<td>HRS Plate</td>
<td>A 36</td>
<td>A 36</td>
</tr>
<tr>
<td>3/161206</td>
<td>Round Tube</td>
<td>4.67</td>
<td>HRS Tube</td>
<td>A 519(1026)</td>
<td>A 519(1026) HRS</td>
</tr>
<tr>
<td>179BP</td>
<td>Round Tube</td>
<td>4.55</td>
<td>SCH80 Black Pipe</td>
<td>A53 F</td>
<td>A53 F</td>
</tr>
<tr>
<td>3/16BP</td>
<td>Round Tube</td>
<td>4.67</td>
<td>SCH80 Black Pipe</td>
<td>A53 F</td>
<td>A53 F</td>
</tr>
<tr>
<td>1/8BP</td>
<td>Round Tube</td>
<td>3.18</td>
<td>SCH80 Black Pipe</td>
<td>A53 F</td>
<td>A53 F</td>
</tr>
<tr>
<td>1/4-1026</td>
<td>Round Tube</td>
<td>6.35</td>
<td>HRS Tube</td>
<td>A 519(1026)</td>
<td>A 519(1026) HRS</td>
</tr>
<tr>
<td>1/4ALU</td>
<td>Plate</td>
<td>6.35</td>
<td>LG Aluminum Plate</td>
<td>Aluminum 5060</td>
<td>5052 H32</td>
</tr>
<tr>
<td>3/16ALU</td>
<td>Plate</td>
<td>4.67</td>
<td>LG Aluminum Plate</td>
<td>Aluminum 5060</td>
<td>5052 H32</td>
</tr>
<tr>
<td>1/8WASH</td>
<td>Washer</td>
<td>3.18</td>
<td>Flat Washer Z.P</td>
<td>A 519(1026)</td>
<td>HRS</td>
</tr>
</tbody>
</table>
4.1.8 Units

The basic units used in the FE model were: millimetre (mm) for length, kilogram (kg) for mass, and millisecond (ms) for time. Other units are derived from the above three basic units. Therefore, the unit for force, velocity, and acceleration is kg-mm/ms\(^2\), mm/ms, and mm/ms\(^2\) respectively.

4.1.9 Geometry Model Summary

The positions of the support decks in the trailer are changeable. In the FE model, the decks' positions were adjusted to the similar positions as the test trailer. The complete FE model of the trailer contains 123,492 nodes and 105,027 elements (6,796 spotwelds, 78,955 quadrilateral elements, 1,904 triangular elements, 16,580 hexahedral elements, and 792 pentahedral elements). Figure 34 illustrates the FE model of the trailer.

![Figure 34. The FE model of the trailer.](image)
4.2 Material Models

According to the performances and loading conditions of trailer structures in the actual situations, appropriate material models were chosen from the LS-DYNA material library.

The material types of different structures are specified in the mechanical drawings provided by StarTrans (also see Table 1). These material names are commercial names used by Central Steel & Wire Company [19], who provides structural steels to StarTrans for constructing trailers. From the handbook compiled by Central Steel & Wire Company, the corresponding ASTM (American Society for Testing and Materials) or SAE (Society of Automotive Engineers) names were found. The detailed mechanical properties of each material were then found from the appropriate “ASTM Standards” or “SAE Handbook”.

4.2.1 Material Models for Shell Elements

4.2.1.1 *MAT_PLASTIC_KINEMATIC Model

The materials composed of the trailer are structural steels and aluminum, which mostly undergo elastic deformation when the trailer is in operation. Material type 12 *MAT_ISOTROPIC_ELASTIC_PLASTIC might have been used to model the shell elements. This is a very low cost isotropic plasticity model for three-dimensional solids. This material does not work properly for shell elements when the stress exceeds the yield stress of the material [20]. In the FE model of the trailer, due to limited number of spotweld connections and the severe loading conditions used in the model, plastic deformation may occur during the simulations. Therefore material type 3
*MAT_PLASTIC_KINEMATIC* was used for most shell elements in the FE model. For this material model, the plastic behaviour is simplified as a straight line connecting the yielding point and the ultimate tensile point in the stress-strain curve. In LS-DYNA, material type 3 requires the following information for input:

1) The density of the material;
2) Young's Modulus of the material;
3) Poisson's ratio for the material;
4) The yield stress of the material;
5) Plastic hardening modulus of the material;
6) Hardening parameter of the material;
7) Two parameters needed for strain rate considerations (if these two pieces of information are not provided, strain rate effects are not considered);
8) Other optional parameters, such as strain rate parameter C and P, Failure strain, and formulating for rate effects.

Young's modulus is the slope of the stress-strain curve up to the proportional limit. It is a measure of the stiffness of the material in its elastic range and has the unit of stress. Most metals exhibit this linear stiffness behaviour and have elastic moduli that vary insignificantly with heat treatment or with the addition of alloying elements [21].

In this investigation, the material properties for plate or tube structures are listed in Table 3. In the *MAT_PLASTIC_KINEMATIC* material model, the hardening parameter (β) was set equal to 1 (isotropic hardening); other parameters regard strain rate effects were not considered. Other parameters needed to define the material model were calculated as shown in Appendix A.
Table 3. Mechanical properties for plate or tube structures.

<table>
<thead>
<tr>
<th>Material</th>
<th>Density (kg/m³)</th>
<th>Young’s Modulus (MPa)</th>
<th>Poisson’s Ratio</th>
<th>Yield Stress (MPa)</th>
<th>Ultimate Stress (MPa)</th>
<th>Elongation (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>ASTM A 36</td>
<td>7,830</td>
<td>$210 \times 10^6$</td>
<td>0.28</td>
<td>$2.482 \times 10^5$</td>
<td>$4.413 \times 10^5$</td>
<td>21</td>
</tr>
<tr>
<td>ASTM A 500 grade B</td>
<td>7,830</td>
<td>$210 \times 10^6$</td>
<td>0.28</td>
<td>$3.172 \times 10^5$</td>
<td>$3.999 \times 10^5$</td>
<td>23</td>
</tr>
<tr>
<td>ASTM A 500 grade C</td>
<td>7,830</td>
<td>$210 \times 10^6$</td>
<td>0.28</td>
<td>$3.447 \times 10^5$</td>
<td>$4.275 \times 10^5$</td>
<td>21</td>
</tr>
<tr>
<td>ASTM A 519 (SAE 1026) HRS</td>
<td>7,830</td>
<td>$210 \times 10^6$</td>
<td>0.28</td>
<td>$2.413 \times 10^5$</td>
<td>$4.413 \times 10^5$</td>
<td>24</td>
</tr>
<tr>
<td>ASTM A 53 F</td>
<td>7,830</td>
<td>$210 \times 10^6$</td>
<td>0.28</td>
<td>$2.068 \times 10^5$</td>
<td>$3.309 \times 10^5$</td>
<td>25</td>
</tr>
<tr>
<td>Aluminium 5050 H32</td>
<td>2,711</td>
<td>$71 \times 10^6$</td>
<td>0.33</td>
<td>$1.950 \times 10^5$</td>
<td>$2.300 \times 10^5$</td>
<td>25</td>
</tr>
</tbody>
</table>

4.2.1.2 *MAT_RIGID Model

For some shell element structures with contact holes as shown from Figure 35 to Figure 37, the structures around the holes or the whole entities were assigned rigid material properties, as these structures’ deformations are negligible in the actual service. In trial simulations a non-rigid material model was assigned and severe element distortions occurred in the vicinity of the holes. This resulted in a decrease of the simulation time steps and a significant increase in CPU time. To eliminate this difficulty, a rigid material model was assigned to these regions of shell elements.

Three basic parameters are needed for defining the rigid material: mass density, Young’s modulus, and Poisson’s ratio. The Young’s modulus and Poisson’s ratio of steel material were assigned to the rigid material. The Young’s modulus is used for determining sliding interface parameters if the rigid body interacts in a contact definition. Realistic values for these constants should be defined since unrealistic values may contribute to numerical problems in contact [18].
Figure 35. The end region of the tube was assigned rigid material properties.

Figure 36. The structure around the hole was assigned rigid material properties.
4.2.2 Material Model for Solid Elements

For the pins and bars in the trailer, the deformations were very small, therefore a rigid material model was used to eliminate the cost of deformed solid element computations. Approximating a deformable body as rigid is a preferred modeling technique in many real world applications. Rigid elements are bypassed in the element processing and no storage is allocated for storing history variables; consequently, the rigid material type is very cost efficient. The number, size, and aspect ratio of elements for a rigid material model do not contribute the cost of the simulation [19]. The definition of the rigid solid material model was exactly the same as the definition of the rigid shell elements.
4.3 Element Section Properties

This section describes how the element formulation, integration rule, nodal thickness, and cross section properties were defined. Element section properties were defined by using *SECTION command in LS-DYNA.

4.3.1 Shell Element Property

As stated in section 4.1.2, the shell element formulation option 1 (Hughes-Liu) was chosen in this model. Different shell parts can share one element property definition card in LS-DYNA as long as they have the identical shell thickness. In this investigation, the purpose to define shell element properties is to select an appropriate integration rule and to define shell thicknesses.

4.3.2 Solid Element Property

All solid elements were given the element formulation option 1 (constant stress solid element) in this investigation. This is an under-integrated solid element that makes computation very effective.

4.4 Contact Algorithms

In dynamic finite element analysis, deformations and deflections are often larger than those typically seen in static analyses. One must describe the contact between different parts of the model as well as intra-part contact when a part buckles upon itself. In the trailer FE model, if contact algorithms were not defined, the interacting
components will penetrate each other without resistance. The keyword *CONTACT provides a way of treating interacting between disjoint parts. Different types of contact may be defined according to the interacting performances. In this project, four different types of contact algorithms were used in the numerical model: *CONTACT_AUTOMATIC_NODES_TO_SURFACE, *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE, *CONTACT_SINGLE_EDGE, and *CONTACT_AUTOMATIC_SINGLE_SURFACE.

Non-automatic contacts were tried in this research, however, the simulations terminated unexpectedly without providing any warning or error information. Another reason for choosing automatic type contact is that the orientation of element segments is not necessary. Automatic contact was chosen because this input simplifies the problem definition and is generally more reliable [22]. Automatic contact definitions are commonly used in many automotive structural modeling investigating.

Surface-to-surface contact is fully symmetric — choice of slave and master surfaces is arbitrary. Generally, this type of contact algorithm is defined for two deformable entities in possible contact. Segment sets were used to choose the possible contact regions for both the master and slave sides in order to save CPU time.

Nodes-to-surface contact is especially suited to nodes contacting rigid bodies [22]. In this contact algorithm, slave sides are defined by individual nodes (nodes set) instead of segments. In this research, all contacts between shells and rigid bodies were defined using node to surface contact algorithm and the rigid bodies were made the master sides. This type of contact is computationally efficient.
For all contacts investigated in this project, static and dynamic coefficients of friction were defined as 0.3 and 0.2 respectively. In total, 148 contact algorithms, using the penalty method, were defined in the model. This method consists of placing normal interface springs between all penetrating nodes and contact surfaces [23]. The penalty scale factor was set to unity for contact definitions that experienced little or no excessive nodal penetrations. If excessive penetrations were observed, the penalty scale factor was increased to 10. After initial trial simulation, it was observed that the penalty scale factors had to be increased in 45 (out of 148) contact algorithms.

The difficulty in defining contact is to minimize the penetrations. Generally, increasing scale factor for penalty stiffness can reduce penetrations greatly. For some shell edge to pin contacts that approach did not work well, other methods were introduced to eliminate penetrations. The following are the illustration for a few contact examples.

4.4.1 Hinge Pin Contact

In the trailer, the connections between the front hinge assembly and the bottom front deck, as well as between the bottom front deck and bottom middle deck, are hinge pin connecting mechanism as shown in Figure 38. There are six black pipes and one long pin on each side of the connecting decks. Three round black pipes are welded to one rectangular tube belonging to one deck and the other three round black pipes are welded to another rectangular tube belonging to the other deck. The long pin is inserted into these black pipes. The two decks can rotate relatively to each other along the pin. In order to model this mechanism in the FE model, the black round pipes were assigned a rigid material model. *CONSTRAINED_EXTRA_NODES were used to constrain the
round pipes to the two rectangular tubes on both sides separately. *CONTACT_AUTOMATIC_NODES_TO_SURFACE was used to define the contact between the black round pipes and the pin. *CONTACT_SINGLE_EDGE was used to define the contacts between the pipes themselves along the axial direction of the pipes.

![Diagram of hinge pin connection between two decks](image)

Figure 38. Hinge pin connection between two decks.

4.4.2 Plate Edge to Pin Contact

In the trailer structure, a considerable amount of plate edge to pin contacts exist. In defining these contacts, care must be taken in choosing the slave nodes. Although only the nodes around the plate hole interact with the solid rigid pin, the slave nodes
chosen must include at least two layers of nodes in the radial direction; otherwise significant penetrations will occur when the interacting force is large.

Figure 39 shows an example of shell edge to pin contact. The two plates were given the same downward velocity and the two pins' ends were fully constrained from motion. For this example presented, the pins were assigned a rigid material model and the plates were elastic-plastic. All the simulation conditions are the same for the two contacts except for the selection of contacting slave nodes as shown in Figure 39. Figure 40 illustrates the large penetrations observed when slave nodes were incorrectly assigned.

![Single layer of slave nodes selected](image1)

![Multiple layers of slave nodes selected](image2)

Figure 39. Selection of slave nodes for defining contact algorithms.

![Penetration in the left contact](image3)

Figure 40. Penetration in the left contact.
One exception to this approach was the contact occurring between the latch pins and the holes of rectangular tubes as shown in Figure 41. No matter how many nodes were chosen around the holes for the slave side, even after SFS and SFC were increased to 300, a large penetration was still observed. Perhaps it was due to the number of holes within the rectangular tubes and a deficiency in the number of nodes between neighbouring holes.

![Figure 41. Contact between the latch pins and tube holes.](image-url)

In order to eliminate penetration for this kind of contact, another constraint algorithm had to be used: *CONSTRAINED_EXTRA_NODES*. Figure 42 shows that the nodes around the pins, including nodes on the tube and the latch plate, are chosen as the extra nodes of the rigid pins. Since there is little or no relative movement between the pins and the holes in actual dynamic conditions, this constraint method would cause negligible discrepancy with actual motions.
4.4.3 Internal Contact Within Rectangular Tubes

Figure 43 shows the contact between two rectangular tubes, one inside another. Since the mesh densities on the contact tubes were different, a nodes to surface contact algorithm was selected to model contact between the two entities. In order to decrease the CPU cost, only the nodes and segments in the contact region were chosen for contact.
4.4.4 Contacts Between C–Tubes and Rectangular Tubes

In Figure 44, the rectangular tube can slide inside along the C-section tube. A nodes to surface contact algorithm was used to define the contact between the C-tubes and rectangular tubes.

Figure 44. Contact between the rectangular tube and the C–tube.

4.4.5 Other Potential Contacts

During static conditions, some neighbouring structures do not interact. When vehicles are in transit, some entities will experience contact due to the deformations associated with dynamic conditions. In the FE model, some potential contacts were defined if it was unsure as to whether or not neighbouring structures would interact. Figures 45 and 46 show two examples that potential contacts may exist.
Figure 45. Potential contacts exist between the support tube and the outside plate as well as between the support tube and the inside short tube.

Figure 46. Potential contact exists between the C-section plate and tube.

Figure 47 shows the rear bottom deck configuration. Several layers of plates rest on top of one another. Complex contacts will occur between these plates under dynamic loading condition. In this case, *CONTACT_AUTOMATIC_SINGLE_SURFACE was used to define the contacts between these plates.
4.4.6 Contacts For a Typical Mechanism

Since the trailer structure is very complicated, a considerable number of contact algorithms would be needed in the FE model in order to represent the trailer’s functioning mechanisms. For example, in Figure 48 around the hinge area, four contact algorithms were used to ensure the same functions achieved within the numerical model as in the actual situation. In the present configuration, the small flat bar is welded to the rectangular tube, the pin is welded to the small flat bar, the round flat plate is welded together with the pin, and the large flat bar can rotate around the pin. In the FE model, the small flat bar was constrained to the rectangular tube by spotwelds. The pin and round flat plate was constrained to the small flat bar using *CONSTRAINED_EXTRA_NODES_SET, in which the pin was defined as rigid, the node set included nodes in the small flat bar and the round flat plate. The contact between the large flat bar and the pin was defined by a nodes to surface contact algorithm. In addition, it is possible that the large flat bar contact with the round flat
plate and the small flat bar on both sides. Therefore, another nodes to surface contact algorithm was needed to define this possible contact.

![Diagram of hinge connection structures](image)

Figure 48. The hinge connection structures.

### 4.5 Boundary Conditions

#### 4.5.1 Velocity Definition

An initial velocity of 50 km/h was prescribed for the trailer in its forward direction (negative X direction in the numerical model), which represents the same velocity used in the field tests. *INITIAL VELOCITY was used to define initial nodal velocity.
4.5.2 Gravity Definition

In the FE model, gravity is applied to all elements using the *LOAD_BODY_Z and *DEFINE_CURVE commands to simulate the weight of the trailer.

4.5.3 Prescribed Motion Definition

In driving conditions, the trailer is supported at the hitch pin and the suspension frames on both rear sides of the trailer. The impacts on the trailer due to harsh driving conditions are transmitted to the trailer structures through these three locations. The vibration data (acceleration or velocity) of the three locations were collected in the field test and prescribed in the FE model. In the model *BOUNDARY_PRESCRIBED_MOTION_SET was used to define the motions of nodes in the proximities of the support locations. The test data was taken immediately before the trailer’s wheels hit the speed bump and lasted for 0.75 seconds. The data taken in this period was used as the inputs for the simulations, as the impact during this period was the most significant. Only the vertical motions were prescribed as these data were recorded experimentally and represent the major loading mode.

4.5 Output Database

During the computation of the FE model, the output data are indispensable to obtain output files containing result information.
4.6.1 Binary Output.

In the FE model, the binary file D3PLOT was output, which contains plotting information to analyze data over the three dimensional geometry of the model. These databases can be plotted using LS-POST to form dynamic images of the simulation (animation) deformation and stresses.

4.6.2 ASCII Output

These data can graphically interpret the same results as the binary output. It allows the user to visualize time history results (X-Y data) gathered from LS-DYNA simulations.

4.6.2.1 Nodout File — Nodal Point Data

Thirteen nodes were chosen from the model to output the nodal information, including displacements, velocities and accelerations in three directions and the resultants. These nodes are located at the front top, middle top, and rear top of the trailer, as the deflection on the top is generally larger than other regions. These nodal points are chosen in order to compare the trailer simulation results with the field test results. In order to get the node output data comparable to the test data, the output interval of the node information should be identical to the test data-sampling rate. The test data-sampling rate was 3000 Hz, so the output interval was set to 0.25 milliseconds during the 0.75 seconds simulations.

4.6.2.2 Glstat File — Global Data Statistics

Energy of the whole model during the simulations. The energy data is to be used to validate the simulations.

4.7 Other Considerations in the Finite Element Model

4.7.1 Simulation Duration

The simulation duration was set to 0.75 seconds in the *CONTROL_TERMINATION command as the trailer required approximately 0.5 seconds to negotiate the speed bump at a driving velocity of 50 km/h. The input test data from field tests were also taken for a period of 0.75 seconds.

4.7.2 Energy Control

In order to validate the simulation, hourglass energy and sliding interface energy were calculated and output. In the input file, the command *CONTROL_ENERGY was used to request output of both hourglass and sliding interface energies.

The hourglass energy gives an indication of the amount of hourglassing that is occurring during the numerical simulation. This mode of deformation for the finite elements is due to the single integration point selected for evaluating the stresses and strains. The finite elements deform according to hourglass shape. It represents an unrealistic mode of deformation. In order to control these purely numerical deformations, hourglass resisting forces are added for cases when they are excited. Then, in the mechanical energy balance, it appears an hourglass energy that is linked to the hourglass resisting forces against deformation of hourglass modes.
In the absence of friction, the sliding interface energy for any simulation should be zero as the sliding interface energy gives an indication of the work done to overcome frictional forces and nodal displacements at the contact interface. Large negative values of the sliding interface energy can indicate that excessive nodal penetrations are occurring on the contact surface. Large negative values of the sliding interface energy imply that the validity of the FE model is questionable.
5. EXPERIMENTAL SET-UP AND TEST SCENARIOS

Field tests were conducted to investigate dynamic responses of the trailer, gather load spectrum and vibration data as inputs to the numerical model, and as a means of verification and/or comparison of the response of numerical model. A trailer with geometry similar to that of the numerical model under investigation was used for the field tests. Figure 49 shows the test car-carrier.

![Figure 49. The test car-carrier.](image)

5.1 Experimental Set-up

For the experimental field tests there were no vehicles loaded onto the trailer. In order to induce manageable vibrations, the data was collected when the test trailer was negotiating a typical parking lot speed bump. All tests were conducted at a truck speed of 50 km/h. An illustration of the test trailer, instrumentation positions, and the speed bump used in the tests is shown in Figure 50. The instrumentation locations are listed below:

- Point (1) is near the hitch pin;
- Point (2) is located at point on the rear suspension frame of the driver’s side;
• Point (3) represents a point on the rear suspension frame of the passenger’s side;
• Point (4) is on the top region of a middle vertical tube of the driver’s side;
• Point (5) represents a point on the base horizontal tube of the passenger’s side.

Data collected from points (1), (2), and (3) were used for the inputs to numerical models; data collected from points (4) and (5) were used for the comparison with the simulation results. At point (4) lateral accelerations (Y direction) was measured. The accelerometer at Point (1) was set under velocity mode so the vertical velocity (Z direction) was measured in point (1) while vertical accelerations were measured for other points.

Figure 50. Measured points on the trailer and the speed bump used in the field tests.
The instrument set-up is illustrated in Figure 51. It is comprised of the items listed below:

- Four accelerometers, with a sensitivity of 8 mV/ms\(^{-2}\) and a frequency response of 0.2 Hz to 3,500 Hz;
- An amplifier, with a sensitivity of 0.01 mV/pC to 10 V/pC and frequency range from 0.2 Hz to 100 kHz;
- A data tape cassette recorder.

Microdot cables were used to connect the accelerometers to the signal amplifier and coaxial cables were used to connect the amplifier to the data recorder.

![Diagram of instrumental and data-transferring set-up.](image)

**Figure 51. Instrumental and data-transferring set-up.**

Accelerometers were glued to the measuring points. Microdot cables were used to connect the accelerometers to the signal amplifier, and coaxial cables were used to connect the amplifier to the data recorder. Figure 52 and Figure 53 show the data conditioner and the data tape recorder respectively. The measured data were stored in the tape recorder.
Figure 52. Data conditioner and cables wire-up.

Figure 53. Data tape recorder.
5.2 Test Scenarios

In order to develop a more robust model validation and comparison, three test scenarios were investigated. Data collected from these three tests would be used to compare with the numerical simulation results for the three corresponding simulation conditions. The three test scenarios are described below.

5.2.1 Test Scenario 1

Wheels of both sides of the trailer negotiated the speed bump concurrently with a driving direction perpendicular to the speed bump as shown in Figure 54. In this case, points (1), (2), (4) and (5) were instrumented. The vibration data for points (2) and (3) were assumed identical.

![Figure 54. Test scenario 1.](image)

5.2.2 Test Scenario 2

The vehicle negotiated the speed bump with only the driver’s side wheels passing over the bump with a driving direction perpendicular to the speed bumps as shown in Figure 55. Points (1), (2), (3) and (4) were instrumented.
5.2.3 Test Scenario 3

Both sides negotiated the speed bump with a driving direction 30 degrees measured from the normal direction of the speed bump as illustrated in Figure 56. Points (1), (2), (3) and (4) were instrumented.
5.3 Data Processing

The test data were transferred to a personal computer for further usage from the tape recorder through a multi-channel digital converter (webDAQ/100) as illustrated in Figure 51. The data-sampling rate was set to 3,000 Hz. The data sampling started just before the car-carrier's front wheels hit the speed bump and the sampling duration was 3 seconds. Two repeated tests were conducted for every test scenario.

5.3.1 Calibration Factor Calculation

There are four input channels in the tape recorder, implying that four accelerometers could be connected to the data tape recorder to collect vibration data at four different points concurrently. Before conducting the measurements, each accelerometer connecting to a specific input channel has to been excited to obtain the calibration factor. The excited signals were stored in the tape recorder. For channel 1, the calibration value was 14.4 mm/s/V; for channel 2, channel 3, and channel 4, the calibration value was 14.14 m/s²/V. From the excited signals, the calibration factor for each channel was calculated. Figure 57 shows the calibration signal curves recorded in the tape recorder for the four channels separately. The calculation process of the calibration factors is shown in Appendix B.
Figure 57. Calibration signal curves for the four channels.

5.3.2 Test Data Processing

The test value (in voltage) multiplied by calibration factor (in mm/s·v or m/s²·v) yields the actual velocity or acceleration measurement. After examining the test curves, similar curve profiles were observed for tests under identical same test scenarios. That implies that the conditions for each test scenario were consistent. Figures 58 through 60 illustrate the vibration curves for the data collected from different test scenarios.
Figure 58. Test data curves for test scenario 1 (both-side impact).

Figure 59. Test data curves for test scenario 2 (driver's side impact).
Figure 60. Test data curves for test scenario 3 (diagonal direction impact).

Large amplitude vibrations were observed due to impact with the speed bump. From the acceleration curves of point 2 (as shown in Figures 58 through 60), sinusoidal vibrations were observed for the suspension systems of the test trailer. Detailed examination of these curves revealed that the vibration frequencies immediately after impacts were approximately 1.5 to 2.0 Hz, which lie in the range of the natural vibration frequency of the air-ride suspension system.
6. COMPARISONS BETWEEN SIMULATION RESULTS AND TEST DATA

All numerical simulations were conducted using LS-DYNA version 960 (revision 1488) on a personal computer with 512 megabytes of RAM employing dual AMD Athlon 1.8 GHz processors. The simulation duration was 0.75 seconds and the approximate CPU time was 65 hours for each simulation.

6.1 Simulation Validation

Both hourglass and sliding interface energies were calculated during the simulations as quality control check used during dynamic simulation. Usually, the simulation is acceptable if the hourglass energy is controlled less than 10% of the internal energy, and the sliding energy is less than 5% of the total energy respectively. For all the simulation conditions, the hourglass energy is less than 10% of the internal energy, and sliding energy is less than 5% of the total energy. This implies an acceptable energy balance.

6.2 Comparisons Between Simulation Results and Test Data

The simulation results were compared with the test results from both qualitative and quantitative aspects. Qualitative checks considered the macroscopic motion of the trailer using video footage from the field tests and the animations from the FE simulations. For quantitative comparison, the response data of simulation results and field test measurements from accelerometers were compared. During the simulations, the
data output interval was set to 0.25 milliseconds so that data output frequency was the same as the test data sampling frequency.

6.2.1 Qualitative Comparisons

The field test processes were recorded on videotape for subsequent observation and comparison with computer-generated animations. Comparing the animations of the simulation results with the test video records for the three test scenarios revealed that the dynamic responses of the numerical model and actual trailer were very similar. In the field test, the lateral vibration at the top of the trailer increased from test scenario 1 to test condition 3. Similar results were observed for the lateral vibration data for the different test scenarios in the numerical model. This implies the FE model of the trailer has reasonable responses according to different applied boundary conditions. Engineers from a car-carrier manufacturing company also analyzed the dynamic animations of the simulation results and felt that the simulations predicted the response of the trailer based upon the prescribed inputs of the suspension and the hitch regions.

6.2.2 Quantitative Comparisons

Acceleration data was output during the simulation for points located at positions approximately identical to the test measurement points (4) and (5). The lateral deflection data of some top points in the simulations was also output. For all the three simulation conditions, the maximum lateral deflections at the top region of the trailer model were very close to those estimated from the video records of the field tests. Acceleration data from the simulations was compared to the test data. Figures 61 through 64 show the results of the accelerations for the experimental tests and numerical simulations.
A duration of 0.25 seconds was chosen to compare the simulation data and test data since this time interval represents the period when dynamic effects are most significant. After this period both the field test trailer and the FE model of the trailer experienced natural vibrations. It was observed that the FE simulation results greatly over-predicted the responses of the field test trailer after this period due to the non-existence of damping in the FE model.

The numerical simulation data did not predict the exact accelerations experimentally measured. This was anticipated due to the differences of some aspects of the field tests and simulations. A discussion of the major differences is provided below.

![Graph 1: Vertical accelerations of point (5) for test condition 1.](image1)

![Graph 2: Lateral accelerations of point (4) for test condition 1.](image2)

Figure 61. Vertical accelerations of point (5) for test condition 1.

Figure 62. Lateral accelerations of point (4) for test condition 1.
6.2.2.1 Boundary Condition Differences

The FE model was assigned an initial velocity of 50 km/h, while the velocity of the test trailer may have been slightly different from 50 km/h and the driving velocity of the trailer was unstable as it passed over the speed bump. The simulation inputs did not include lateral and longitudinal vibration data, which actually existed in the field tests and to some extent can affect the structural vibrations. At the simulation start time, the test trailer structures were already excited due to the impact between the front wheels and the speed bump, whereas in the numerical simulation, the components of the FE model were in a pure horizontal motion with an initial velocity. The test trailer has a two-axle
suspension system on each side, and the vibrations at different locations along the suspension frame were different at identical times. In the FE model, identical vibration data was assigned for the same suspension frame.

6.2.2.2 Structural Configuration Differences

The FE model was developed with a number of structural simplifications and omissions. The different structural configurations between the FE model and the test trailer may have also contributed to the differences between the measured and simulated responses.

6.2.2.3 Material Mechanical Property Discrepancies

The mechanical properties assigned to all components within the numerical model were virgin mechanical properties that did not contain any damage or degradation. The mechanical properties (such as Young's modulus, Poisson's ratio, yield strength) of the structural components for the actual trailer have most likely degraded due to prior use. This assumption of virgin mechanical properties for the numerical model may also give rise to differences in the experimental and numerical observations.

6.2.2.4 The Finite Element Method's Limitations

The finite element method is an approximation to the actual responses of a structure with applied boundary conditions and external loads. Many factors during the model building process, including mesh density, element type, and element formulation, can affect the simulation results. In the numerical model, no global damping was applied. The acceleration data output from LS-DYNA is obtained by applying the central difference scheme to the velocities. The computed LS-DYNA acceleration histories are usually noisy and are incomparable directly with the test measurements.
In an attempt to illustrate some form of acceptability in the numerical model, the average and root mean square (RMS) values of both the experimental and numerical results during 0.25 seconds was calculated and compared as presented in Table 3 for the three different test conditions, in which "g" is the local acceleration due to gravity. Appendix C shows the Average and RMS value calculations using MathCAD for Condition 3.

Table 4 illustrates that the amplitudes of the simulation accelerations are generally higher than the corresponding test values. The RMS values of the simulations (on average) are 22% higher than those of the test data. Since the calculated average values of the accelerations were very small, any slight difference in average values gave rise to significant relative differences. The simulation values are not unrealistic based upon the reasons described above. Combined with the quantitative comparisons, the FE model can be used to predict the actual response of the trailer with satisfaction.

Table 4. Comparison of average and RMS values of acceleration.

<table>
<thead>
<tr>
<th></th>
<th>Condition 1</th>
<th>Condition 2</th>
<th>Condition 3</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Average</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Simulation</td>
<td>-0.08g’s</td>
<td>0.16g’s</td>
<td>0.05g’s</td>
</tr>
<tr>
<td>Test</td>
<td>-0.01g’s</td>
<td>0.013g’s</td>
<td>0.09g’s</td>
</tr>
<tr>
<td>Relative</td>
<td>-700%</td>
<td>130%</td>
<td>-44%</td>
</tr>
<tr>
<td>Difference</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>RMS</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Simulation</td>
<td>1.86g’s</td>
<td>1.69g’s</td>
<td>1.83g’s</td>
</tr>
<tr>
<td>Test</td>
<td>1.15g’s</td>
<td>1.65g’s</td>
<td>1.36g’s</td>
</tr>
<tr>
<td>Relative</td>
<td>-24%</td>
<td>24%</td>
<td>15%</td>
</tr>
<tr>
<td>Difference</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
7. FULLY-LOADED TRAILER MODELLING

In order to simulate loading conditions, the FE model was updated with models of vehicles acting as payloads. The following describes the modelling process of loaded vehicles and tie-down chains, which fasten vehicles to the car-carrier.

7.1 Loaded Vehicle Modeling

This prototype car-carrier was designed to carry cars, pick-up trucks, minivans and sport utility vehicle (SUVs), among which SUV is the heaviest. In this investigation SUVs were chosen as the payload vehicles on the trailer.

For SUVs, the approximate width of tires is from 157 mm to 233 mm. The curb weight (the nominal weight of a vehicle with standard equipment and a full tank of fuel) of a SUV may be as high as 3,175 kg. The overall height range of a typical SUV is from 1,550 mm to 2,032 mm. In the FE model, when the SUV is tied down on the deck, the contact dimensions between the tire and the deck is assumed to be 185 mm by 200 mm. The mass of a SUV was chosen as 3,175 kg to simulate the worst loading condition. The height of the mass centres of the SUVs used in this investigation was set to 0.8 m measured from the tire base.

If detailed FE models for the SUVs were created, the loaded trailer model under investigation would be extremely large and CPU intensive. It is not necessary to create a detailed FE model for the loaded SUVs because only the trailer structures are of concern in this investigation. As long as the SUV model can represent the real impact effect on
the supporting deck, it should suffice in this investigation. The SUV models in this investigation should be as simple as possible in order to reduce calculation cost.

The SUV model was simplified as four deformable blocks and one rigid block as shown in Figure 65, in which the four deformable blocks represent the tires and the rigid block represents the lumped mass of the entire SUV (excluding the mass of the tires). The rigid block with dimensions of 100 mm by 100 mm by 100 mm and the deformable blocks with dimensions of 185 mm by 200 mm by 200 mm were meshed with eight hexahedral elements.

![Diagram of FE model of a loaded vehicle](image)

*Figure 65. The FE model of a loaded vehicle.*

*MAT_ISOTROPIC_ELASTIC_PLASTIC* was used to assign material properties to the four deformable blocks. Young's modulus and Poisson's ratio values were defined using corresponding steel parameters for simplicity.

For the lumped mass, *MAT_RIGID* was used to define the material. The mass density of the rigid block is $3.175 \times 10^6$ kg/m$^3$. Reasonable values were assigned to other parameters in the definition of the rigid material model.
In this investigation, it was assumed that the mass centre of the SUV model was located at a point 0.8 m above the geometry centre of the four deformable blocks. Six SUVs were loaded on the trailer (three on the top decks and three on the lower decks). Possible SUV interferences along the longitudinal direction between the SUVs were not considered in this investigation.

In order to constrain the motion of the rigid block to the four deformable blocks the keyword command *CONSTRAINED_EXTRA_NODES_SET was utilized to constrain the top nodes on the deformable blocks to the rigid block.

7.2 SUV Tie-down Chain Modeling

After the vehicle is driven onto 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 as shown in Figure 66. An initial tensile force of approximately 900 N is generated when the vehicle is tied down by turning the ratchet wheel using a steel round bar. According to the Vehicle Shipping Manual [5], the chain angles must be $35^\circ \pm 10^\circ$, as measured from deck to which the vehicle is secured.

In order to model the tie-down chain, another four nodes were created at the correct positions to represent the slot on the base frames of each vehicle. Those four nodes were constrained to the rigid block to simulate the inter-connection between the slots, the deformable blocks, and the rigid block. The chain was modeled as a beam element connected between the detached node and a selected node on the ratchet tube of the trailer as illustrated in Figure 67.
Material type 71, *MAT_CABLE_DISCRETE_BEAM, was chosen for the beam elements. This model permits the elastic cable to be modelled realistically without the cable providing any load bearing capacity if it is in compression. In the material model,
the cable was given the mechanical properties of steel and an initial tensile force of 889.64 N, which is the average tie-down tensile force within the chain.

Element formulation #6 (discrete beam/cable) was chosen for the beam in order to be consistent with the material model. A tubular cross section type was chosen for the beams. The cross section radius of the beams was assumed to be 10 mm and identical length (275 mm) was given to all beams.

An *AUTOMATIC_SURFACE_TO_SURFACE contact algorithm was chosen to define the contacts between the tires and the supporting decks.

In a trial simulation, the ratchet tubes were assigned material model 3 with ASTM A 519 mechanical properties. Unrealistic plastic deformations were noticed on the tubes on which the beam elements were fastened as illustrated in Figure 68. The plastic deformation was caused by a point load application due to the connectivity between the beam and the tube. In the actual situation, the chain is wound around the tube. In order to solve this problem, the ratchet tubes connecting with beams were assigned a rigid material model.

![Figure 68. Unrealistic deformation due to nodal point stress concentration.](image-url)
7.3 Fully loaded trailer model

The positions of the loading decks in the trailer are changeable. For different loading conditions (number of loaded vehicles, sizes, and masses), the decks’ positions may be different. According to the traffic regulations, the maximum overall height for car-carriers moving on road should not exceed 4.6 m and the total weight of the transport system must be less than the federal gross vehicle weight limit of 36,300 kg. In the complete FE model shown in Figure 69 and Figure 70, the decks were raised to the limit position in order to simulate the worst loading conditions. The total weight of the loaded trailer is approximately 25,000 kg, which included the trailer and six SUVs.

![Figure 69. Isometric view of the FE model of the loaded trailer.](image1)

![Figure 70. Side view of the FE model of the loaded trailer.](image2)
8. ANALYSIS OF SIMULATION RESULTS

Computer-based FE simulations are an invaluable tool for supplementing field tests and investigating design deficiencies. They provide extensive dynamic performance information such as displacement, velocity, stress/strain distribution, and forces, at a fraction of the cost of field tests.

8.1 Analysis of Effective Stresses

The loaded trailer model was simulated in the identical conditions with the three test scenarios. Examination of the effective stress (vonmisses stress) or effective plastic strain (calculated using the same approach as the vonmisses stress) distribution over the duration of the simulation showed several regions of high stress/strain concentration as illustrated in Figure 71. These locations are:

1. The front tow bar region in the front middle area;
2. The front base corner regions on both side of the trailer model
3. The middle base corner regions on both side of the trailer model;
4. The rear base corner regions on both side of the trailer model;
5. The suspension frame regions on both side of the trailer model;
6. The middle adjustable tubes on both sides of the trailer model.

Most fractures and cracks were also found at those locations in a similar trailer in actual service. Figures 72 through Figure 74 illustrate the effective stress contours of some stress concentration regions of the FE model for simulation condition 3.
Figure 71. Illustration of stress/strain concentration regions in the FE model.

Figure 72. Stress concentration at the front tow bar region.
Numerical examination of the maximum effective stress values shows that some maximum stresses are greater than the yield stress of the material. Plastic strains would be generated at these regions. The use of numerical spotweld connections could potentially cause the high stresses as illustrated in Figure 72. The high stresses in Figure 73 were caused by the hole in the structure. The high stresses in Figure 74 were caused due to the large weight of the loaded vehicles on the two top decks.

Table 5 summarizes the maximum effective plastic strain or maximum effective stress at stress/strain concentration regions for different simulation conditions. In the
table, if the maximum effective stress is greater than yield stress, plastic effective strains
are listed; otherwise, the maximum effective stresses are listed.

Table 5. The maximum effective plastic strain or maximum effective stress at
stress/strain concentration regions for different simulation conditions.

<table>
<thead>
<tr>
<th>Condition</th>
<th>Front tow bar region (strain)</th>
<th>Front base corners (strain)</th>
<th>Middle base corners (stress)</th>
<th>Rear base corners (strain)</th>
<th>Suspension frames (strain)</th>
<th>Adjustable tubes (strain)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Condition 1</td>
<td>$3.4 \times 10^{-2}$</td>
<td>$3.35 \times 10^{-3}$</td>
<td>195 MPa</td>
<td>$7.8 \times 10^{-3}$</td>
<td>$6.0 \times 10^{-2}$</td>
<td>$2.5 \times 10^{-2}$</td>
</tr>
<tr>
<td>Condition 2</td>
<td>$3.5 \times 10^{-2}$</td>
<td>$3.25 \times 10^{-3}$</td>
<td>260 MPa</td>
<td>$4.0 \times 10^{-3}$</td>
<td>$2.8 \times 10^{-2}$</td>
<td>$1.2 \times 10^{-2}$</td>
</tr>
<tr>
<td>Condition 3</td>
<td>$4.5 \times 10^{-2}$</td>
<td>$4.0 \times 10^{-3}$</td>
<td>211 MPa</td>
<td>$3.7 \times 10^{-3}$</td>
<td>$2.6 \times 10^{-2}$</td>
<td>$6.7 \times 10^{-3}$</td>
</tr>
</tbody>
</table>

**8.2 Lateral Deflection/Deformation Analysis**

Figure 75 shows the maximum lateral deflection curves for the three simulation
conditions. The maximum lateral deflections for simulation condition 1, condition 2, and
condition 3 is 62 mm, 73 mm, and 75 mm respectively. The simulation condition 3
(diagonal impact) caused most serious lateral deflection in the simulation.

![Maximum lateral deflection curves for original model](image)

Figure 75. The maximum lateral deflection curves for the three simulation conditions.
Figure 76 shows that a large deformation was observed on the slender C-plates on each side of the FE model, which may cause potential buckling on the C-plates. This large deformation was caused by the significant lateral deflections at the top decks, giving rise to high stress on the slender structure of the C-plates.

Figure 76. Back view of the deformation of slender C-plate.
9. LOCAL STRUCTURAL MODIFICATIONS

9.1 Structural Modifications

Local geometry modifications as shown in Figure 77, were investigated to decrease stress/strain concentrations previously identified. For the front tow bar region, a stiffening plate was extended and two gusset plates were added with the anticipation of strengthening that region. In the front, middle, and rear base stress concentration regions, the original plates were enlarged. In addition, support tubes were added in the front and the middle to reduce the stress concentrations on the base tubes and to reduce lateral deflections at the top. In order to reduce the deformation on the slender C-plates, the thickness of the plates was doubled from 6.35 mm to 12.7 mm. Figure 78 illustrates the details of the modifications at the front tow bar region and the front base corners.

After local modifications, the total mass of the loaded trailer model is 25,275 kg, which is a 284 kg (1%) increase relative to the original loaded trailer model. This small mass increase will not have significant effects to the loading conditions.

Figure 77. Local geometric modification locations in the trailer.
Figure 78. Bottom view of the modifications at the front tow bar region and the front base corner regions.

9.2 Reduction of the Stress/Strain Concentrations

By comparing the maximum effective plastic strain or effective stress at each region of concern, before and after the modifications, the influences of the structural changes can be observed.

Table 6 through Table 8 summarize the maximum effective plastic strains or effective stress observed in each region of concern for the three simulation conditions, before and after local structural modifications.

Table 6. Maximum effective plastic strain and/or maximum effective stress comparison for simulation condition 1, before and after local modifications.

<table>
<thead>
<tr>
<th></th>
<th>Front tow bar region (strain)</th>
<th>Front base corners (strain)</th>
<th>Middle base corners (stress)</th>
<th>Rear base corners (strain)</th>
<th>Suspension frames (strain)</th>
<th>Adjustable tubes (strain)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Before Modifications</td>
<td>$3.4 \times 10^{-2}$</td>
<td>$3.4 \times 10^{-3}$</td>
<td>195 MPa</td>
<td>$7.8 \times 10^{-3}$</td>
<td>$6.0 \times 10^{-2}$</td>
<td>$2.5 \times 10^{-2}$</td>
</tr>
<tr>
<td>Local Modifications</td>
<td>$9.0 \times 10^{-3}$</td>
<td>$6.5 \times 10^{-3}$</td>
<td>142 MPa</td>
<td>$6.8 \times 10^{-3}$</td>
<td>$5.0 \times 10^{-2}$</td>
<td>$2.2 \times 10^{-2}$</td>
</tr>
<tr>
<td>Reduction</td>
<td>44%</td>
<td>-94%</td>
<td>27%</td>
<td>13%</td>
<td>17%</td>
<td>12%</td>
</tr>
</tbody>
</table>
Table 7. Maximum effective plastic strain and/or maximum effective stress comparison for simulation condition 2, before and after local modifications.

<table>
<thead>
<tr>
<th></th>
<th>Front tow bar region (strain)</th>
<th>Front base corners (strain)</th>
<th>Middle base corners (stress)</th>
<th>Rear base corners (strain)</th>
<th>Suspension frames (strain)</th>
<th>Adjustable tubes (strain)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Before Modifications</td>
<td>$3.5 \times 10^{-2}$</td>
<td>$3.25 \times 10^{-3}$</td>
<td>260 MPa</td>
<td>$4.0 \times 10^{-3}$</td>
<td>$2.8 \times 10^{-2}$</td>
<td>$1.2 \times 10^{-2}$</td>
</tr>
<tr>
<td>Local Modifications</td>
<td>$1.1 \times 10^{-2}$</td>
<td>$6.0 \times 10^{-3}$</td>
<td>197 MPa</td>
<td>$5.5 \times 10^{-3}$</td>
<td>$2.2 \times 10^{-2}$</td>
<td>$7.0 \times 10^{-3}$</td>
</tr>
<tr>
<td>Reduction</td>
<td>69%</td>
<td>-84%</td>
<td>24%</td>
<td>-37%</td>
<td>21%</td>
<td>42%</td>
</tr>
</tbody>
</table>

Table 8. Maximum effective plastic strain and/or maximum effective stress comparison for simulation condition 3, before and after local modifications.

<table>
<thead>
<tr>
<th></th>
<th>Front tow bar region (strain)</th>
<th>Front base corners (strain)</th>
<th>Middle base corners (stress)</th>
<th>Rear base corners (strain)</th>
<th>Suspension frames (strain)</th>
<th>Adjustable tubes (strain)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Before Modifications</td>
<td>$4.5 \times 10^{-2}$</td>
<td>$4.0 \times 10^{-3}$</td>
<td>211 MPa</td>
<td>$3.7 \times 10^{-3}$</td>
<td>$2.6 \times 10^{-2}$</td>
<td>$6.7 \times 10^{-3}$</td>
</tr>
<tr>
<td>Local Modifications</td>
<td>$1.1 \times 10^{-2}$</td>
<td>$7.0 \times 10^{-3}$</td>
<td>197 MPa</td>
<td>$2.6 \times 10^{-3}$</td>
<td>$2.1 \times 10^{-2}$</td>
<td>$3.0 \times 10^{-3}$</td>
</tr>
<tr>
<td>Reduction</td>
<td>76%</td>
<td>-75%</td>
<td>7%</td>
<td>30%</td>
<td>19%</td>
<td>55%</td>
</tr>
</tbody>
</table>

The above three tables show that the local modifications reduced stress and/or strain concentrations effectively for the majority of the regions concerned. The possible reason for the stress increase in the front base corner is most likely due to that the stiffened tow bar region transferred more force to the front base corners.

### 9.3 Reduction of Lateral Deflections

The maximum lateral deflections are listed in Table 9 for the three simulation conditions, before and after local modifications.
Table 9. Maximum lateral deflection comparisons between original model and locally modified model.

<table>
<thead>
<tr>
<th></th>
<th>Condition 1</th>
<th>Condition 2</th>
<th>Condition 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original Model</td>
<td>62 mm</td>
<td>73 mm</td>
<td>75 mm</td>
</tr>
<tr>
<td>Locally Modified</td>
<td>59 mm</td>
<td>62 mm</td>
<td>67 mm</td>
</tr>
<tr>
<td>Reduction</td>
<td>5%</td>
<td>15%</td>
<td>11%</td>
</tr>
</tbody>
</table>

The locally modified model decreased the maximum lateral deflections for all simulation conditions. Checking the animation motions of the model, it was found there were still large deformations on the slender C-plates for the three simulation conditions, although its thickness has been doubled from the original design. This large deformation implies that the potential to decrease the deformation in the C-plate by increasing the thickness is very limited. The global configuration of the trailer is questionable.
10. GLOBAL STRUCTURAL MODIFICATION

Local modifications to the trailer model resulted in better overall dynamic responses with terms of reducing stress/strain concentrations and lateral deflections. The developments of the plastic strains in the locally modified model indicate that high values of stress are still present. The lateral deflections of the locally modified trailer model are still substantial, especially at the rear top regions, due to lack of strong vertical tubes to support the rear top decks. In the original design, these slender, vibration prone tubes in the rear region could not effectively prevent lateral deflection of the rear top decks. The locally modified model still exhibited large deformations on the C-plates. These analyses show that the potential to improve the model further is very limited if only the local modification approach is used. Therefore, global modifications were considered in order to further improve the overall performance of the trailer.

10.1 Global Structural Reconfigurations

According to the numerical simulation analyses and service practice, global modifications were made to the trailer. The main modifications are summarized in Figures 79 and 80, in which the modification number makes reference to the description provided after these figures.

After global modifications, the total mass of the loaded trailer model is 25,260 kg, which is a 15 kg (1%) reduction relative to the local modified trailer model. This small mass variance will not have significant effects to the loading conditions.
1. In the globally modified model, the two rear top decks are separate. The middle top deck is supported by two fixed vertical tubes, and the rear top deck is supported by a fixed rectangular vertical tube with hollow cross section dimensions of 6.35 mm × 101.6 mm × 76.2 mm and two adjustable tubes on each
side. In order to make this arrangement, the side vertical tubes are reconfigured. This modification will give the two rear top decks stronger supports.

2. All six vertical tubes on each side of the trailer are connected with a steel tube with cross section dimensions of \( 3.175 \text{ mm} \times 38.1 \text{ mm} \times 152.4 \text{ mm} \), which will provide for more support on the upper decks.

3. In the middle of the suspension frame, a main vertical tube with cross section dimensions of \( 4.76 \text{ mm} \times 177.8 \text{ mm} \times 76.2 \text{ mm} \) and two other small vertical tubes were added to strengthen this region as shown in Figure 81, where significant bending stresses were observed in the original design. In order to decrease the possible stress concentrations at the corners of the suspension frames, gusset plates with thickness of 6.35 mm were added at the front and rear corners on both sides.

![Diagram showing structural improvements for the suspension frame.](image)

Figure 81. Structural improvements for the suspension frame.

4. A plate is extended and two gusset plates were added in the front tow bar region. The thickness of the plates at the front base inner corners was increased from 4.76
mm to 9.5 mm. The thickness of the plates at the front base external corner was increased from 9.5 mm to 12.7 mm. Great torque and impact force will be generated in the corner regions when the trailer is in transit.

5. In the original design, the top decks were supported on the slide tubes by slender plates, which made the mass centres of the top decks higher relative to the support slide tubes, thus increasing the lateral vibration of the top decks. In the globally modified model, the side frames of the top decks are connected directly to the vertical slide tubes. The side C-section guide frames of the top decks are connected directly to the vertical slide tubes as shown in Figure 82. The top deck can slide along the C-section guide frame and the position pin can prevent the relative movement between the slide tube and the C-section tube.

![Figure 82. Support configuration of the front top deck.](image)

6. The thickness of the connecting plates between the two middle vertical tubes of the trailer was increased from 6.35 mm to 12.7 mm. The support plates for the front bottom deck were enlarged and reconfigured.
Figure 83 shows the details of the front support mechanism of the front top deck. Slide tubes 1 and 2 can slide inside along the C-section tube 1 and 2 respectively. The two C-section tubes are welded together. The solid block and slide tube 2 are welded together as well as the pin, the slide tube 3 and the side plate. The pin can rotate around its own axis in the hole (not shown in the figure) on the block. Slide tube 3 can move along the support tube. In this way, the top deck can be raised and lowered along the vertical support tube and slide along the two C-section tubes to adjust its position horizontally.

Figure 84 shows the rear support mechanism of the top front deck. C-section tube 1 and the support plate are welded together as well as the slide tube, the pin, and the side plate. The pin supports the deck through the support plate.

Figure 83. The front support mechanism of the top front deck.

The top middle deck uses the same support mechanism as the top front deck. The rear support mechanism of the top rear deck did not change from the original design. The
front support mechanism of the top rear deck is illustrated in Figure 85, in which the support plate and the C-section tube are welded.

![Diagram of the rear support scheme of the top front deck.](image)

**Figure 84.** The rear support scheme of the top front deck.

![Diagram of the front support mechanism of the rear top deck.](image)

**Figure 85.** The front support mechanism of the rear top deck.
10.2 Reduction of Stress/Strain Concentrations

The three simulation conditions were used to simulate the globally modified model. The maximum plastic strain and/or effective stress of the stress/strain concentration regions are listed in Tables 10 to Table 12. In the table, Equations (1) and (2) are used to calculate the “Reduction to Original” and “Reduction to Local modifications” respectively in the table.

\[
\text{Value}_{\text{Reduction from Original}} = \left( \frac{\text{Value}_{\text{Global Modifications}} - \text{Value}_{\text{Before Modification}}}{\text{Value}_{\text{Before Modification}}} \right) \times 100\% \tag{1}
\]

\[
\text{Value}_{\text{Reduction from Local Modifications}} = \left( \frac{\text{Value}_{\text{Global Modifications}} - \text{Value}_{\text{Local Modifications}}}{\text{Value}_{\text{Local Modifications}}} \right) \times 100\% \tag{2}
\]

Table 10. Maximum effective plastic strain or maximum effective stress comparisons among the original model, the locally modified model, and the globally modified model, for simulation condition 1.

<table>
<thead>
<tr>
<th></th>
<th>Front tow bar region (strain)</th>
<th>Front base corners (strain)</th>
<th>Middle base corners (stress)</th>
<th>Rear base corners (strain)</th>
<th>Suspension frames (strain)</th>
<th>Adjustable tubes (strain)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Before Modifications</td>
<td>3.4 \times 10^{-2}</td>
<td>3.4 \times 10^{-3}</td>
<td>195 MPa</td>
<td>7.8 \times 10^{-3}</td>
<td>6.0 \times 10^{-2}</td>
<td>2.5 \times 10^{-2}</td>
</tr>
<tr>
<td>Local Modifications</td>
<td>9 \times 10^{-3}</td>
<td>6.5 \times 10^{-3}</td>
<td>142 MPa</td>
<td>6.8 \times 10^{-3}</td>
<td>5.0 \times 10^{-2}</td>
<td>2.2 \times 10^{-2}</td>
</tr>
<tr>
<td>Global Modifications</td>
<td>5.8 \times 10^{-3}</td>
<td>212 MPa</td>
<td>150 MPa</td>
<td>4.5 \times 10^{-3}</td>
<td>1.8 \times 10^{-2}</td>
<td>Not Present</td>
</tr>
<tr>
<td>Reduction from Original</td>
<td>83%</td>
<td></td>
<td>23%</td>
<td>42%</td>
<td>70%</td>
<td></td>
</tr>
<tr>
<td>Reduction from Local Modification</td>
<td>36%</td>
<td></td>
<td>-6%</td>
<td>34%</td>
<td>64%</td>
<td></td>
</tr>
</tbody>
</table>

* The maximum effective stress of the global modifications is under the yield stress due to global structural modifications; it is not comparable with the locally modified and original models.
Table 11. Maximum effective plastic strain or maximum effective stress comparisons among the original model, the locally modified model, and the globally modified model, for simulation condition 2.

<table>
<thead>
<tr>
<th></th>
<th>Front tow bar region (strain)</th>
<th>Front base corners (strain)</th>
<th>Middle base corners (stress)</th>
<th>Rear base corners (strain)</th>
<th>Suspension frames (strain)</th>
<th>Adjustable tubes (strain)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Before Modifications</td>
<td>$3.5 \times 10^{-2}$</td>
<td>$3.25 \times 10^{-3}$</td>
<td>260 MPa</td>
<td>$4.0 \times 10^{-3}$</td>
<td>$2.8 \times 10^{-2}$</td>
<td>$1.2 \times 10^{-2}$</td>
</tr>
<tr>
<td>Local Modifications</td>
<td>$1.1 \times 10^{-2}$</td>
<td>$6.0 \times 10^{-3}$</td>
<td>197 MPa</td>
<td>$5.5 \times 10^{-3}$</td>
<td>$2.2 \times 10^{-2}$</td>
<td>$7.0 \times 10^{-3}$</td>
</tr>
<tr>
<td>Global Modifications</td>
<td>$6 \times 10^{-3}$</td>
<td>$0.3 \times 10^{-3}$</td>
<td>200 MPa</td>
<td>$1.2 \times 10^{-3}$</td>
<td>$9 \times 10^{-3}$</td>
<td>Not Present</td>
</tr>
<tr>
<td>Reduction from Original</td>
<td>83%</td>
<td>91%</td>
<td>30%</td>
<td>70%</td>
<td>68%</td>
<td>–</td>
</tr>
<tr>
<td>Reduction from Local</td>
<td>45%</td>
<td>95%</td>
<td>-1.5%</td>
<td>78%</td>
<td>59%</td>
<td>–</td>
</tr>
</tbody>
</table>

Table 12. Maximum effective plastic strain and/or maximum effective stress comparisons among the original model, the locally modified model, and the globally modified model, for simulation condition 3.

<table>
<thead>
<tr>
<th></th>
<th>Front tow bar region (strain)</th>
<th>Front base corners (strain)</th>
<th>Middle base corners (stress)</th>
<th>Rear base corners (strain)</th>
<th>Suspension frames (strain)</th>
<th>Adjustable tubes (strain)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Before Modifications</td>
<td>$4.5 \times 10^{-2}$</td>
<td>$4.0 \times 10^{-3}$</td>
<td>260 MPa</td>
<td>$3.7 \times 10^{-3}$</td>
<td>$2.3 \times 10^{-2}$</td>
<td>$6.7 \times 10^{-3}$</td>
</tr>
<tr>
<td>Local Modifications</td>
<td>$1.1 \times 10^{-2}$</td>
<td>$7.0 \times 10^{-3}$</td>
<td>197 MPa</td>
<td>$2.6 \times 10^{-3}$</td>
<td>$2.1 \times 10^{-2}$</td>
<td>$3.0 \times 10^{-3}$</td>
</tr>
<tr>
<td>Global Modifications</td>
<td>$9 \times 10^{-3}$</td>
<td>$1.1 \times 10^{-3}$</td>
<td>185 MPa</td>
<td>$7.7 \times 10^{-4}$</td>
<td>$5.8 \times 10^{-3}$</td>
<td>Not Present</td>
</tr>
<tr>
<td>Reduction from Original</td>
<td>80%</td>
<td>73%</td>
<td>29%</td>
<td>80%</td>
<td>75%</td>
<td>–</td>
</tr>
<tr>
<td>Reduction from Local</td>
<td>18%</td>
<td>84%</td>
<td>6%</td>
<td>70%</td>
<td>72%</td>
<td>–</td>
</tr>
</tbody>
</table>

Large stress or strain reductions were observed for most stress/strain concentration regions with the global modifications. In the middle base corner regions, the reduction of the maximum effective stresses were not eminent, but the effective stress at these regions of the globally modified model were well below the yield stress. In the global
modifications, the slender C-plates were excluded, hence, the stress concentrations in these structures are not of a concern.

10.3 Reduction of Lateral Deflections

Figure 86 through Figure 88 illustrate lateral deflection curves representing the maximum lateral deflections for the three simulation conditions. Note that the positions where the lateral deflection data is measured may be different in the figures.

![Maximum Lateral Deflections for Simulation Condition 1](image)

Figure 86. The maximum lateral deflection curves for simulation condition 1.
Figure 87. The maximum lateral deflection curves for simulation condition 2.

Figure 88. The maximum lateral deflection curves for simulation condition 3.

The maximum lateral deflection comparisons are listed in Table 13 for the simulations of the original model, the locally modified model, and the globally modified model, for the three simulation conditions.
Table 13. Maximum lateral deflection comparisons among the original model, the locally modified model, and the globally modified model.

<table>
<thead>
<tr>
<th></th>
<th>Condition 1</th>
<th>Condition 2</th>
<th>Condition 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original Model</td>
<td>62 mm</td>
<td>73 mm</td>
<td>75 mm</td>
</tr>
<tr>
<td>Locally Modified</td>
<td>59 mm</td>
<td>62 mm</td>
<td>67 mm</td>
</tr>
<tr>
<td>Model</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Globally Modified</td>
<td>25 mm</td>
<td>40 mm</td>
<td>46 mm</td>
</tr>
<tr>
<td>Model</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reduction from</td>
<td>60%</td>
<td>45%</td>
<td>39%</td>
</tr>
<tr>
<td>Original</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reduction from</td>
<td>58%</td>
<td>36%</td>
<td>31%</td>
</tr>
<tr>
<td>Local Model</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 13 shows that for all simulation conditions tremendous deflection reductions were observed with the globally modified model. In total, the comprehensive effects of global modifications are more efficient than those of local modifications, with respect to reducing stress/strain concentrations and lateral deflections.
11. CONCLUSIONS AND RECOMMENDATIONS

This thesis dealt with the experimental and numerical structural analysis of a prototype car-carrier (specifically focusing on the trailer portion of the carrier) when subjected to harsh driving conditions. In addition, geometrical modifications were considered to investigate the potential for increasing the structural performance of the trailer.

11.1 Summary and Conclusions

Based upon the methods used within this research the following conclusions can be made:

11.1.1 Conclusions Regarding Field Testing

1. Field-testing has provided data for the response of a number of structural locations on the carrier trailer when negotiating a speed bump at approximately 50 km/h. The observations illustrate that significant shock and vibrations are induced within the trailer structure, which may give rise to potential damage of loaded vehicles during transit and decrease the life of the trailer.

2. The maximum measured vertical accelerations at the suspension frames were between 5 g’s and 10 g’s for the three test scenarios. After the rear wheels of the trailer impacted the speed bump, the free vibration frequencies of the measured data at the suspension frames were between 1.5 Hz and 2 Hz. The actual natural frequency of an air-ride suspension system is between 1Hz to 2 Hz. These
observations suggest that the test instruments, test methods, experimental set-up, and the data processing techniques used in the tests were reliable.

11.1.2 Summaries and Conclusions on FE Model Development and Validation

1. A highly detailed numerical (FE) model of the trailer has been developed to investigate the stress/strain profile as well as the lateral deflections of the trailer. This approach represents the first attempt in the vehicle carrier design industry to use numerical models to aid in the design and development of vehicle carriers.

2. Comparison of the results between the field tests and numerical simulations suggest that the predictive capabilities of the FE models are acceptable.

3. The numerical simulations of the loaded trailer model revealed several stress/strain concentration regions, which were consistent with the crack regions often found in practice for a trailer with a similar configuration. Also, great lateral deflections were observed in the FE simulation for the original design.

11.1.3 Conclusions on Structural Improvements

1. Based upon the numerical simulations, the suggested local modifications resulted in a significant reduction in stress/strain concentrations as well as lateral deflections of the trailer. The local modification decreased the maximum effective stress or plastic strain by 76% in the front tow bar region, 24% in the middle base corners, 30% in the rear base corners, and 9% in the suspension frames. The local modifications also decreased the maximum lateral deflection by 11%, however it caused a 50% increase of the plastic strain in the front base corners.
3. The global modifications considered in this research, when simulated under identical testing conditions to the previous models, illustrated a significant decrease in localized stresses/strains as well as lateral deflections on the top portion of the trailer. The global modifications decreased the maximum effective stress or plastic strain by 80% in the front tow bar region, 73% in the front base corners, 29% in the middle base corners, 80% in the rear base corners, and 75% in the suspension frames, and decreased maximum lateral deflection by 39%, relative to the original model. The suggested global modifications could significantly increase the trailer life and decrease potential damage to vehicles in transit.

11.2 Recommendations for Future Work

Since this research represents a first attempt in using numerical tools to investigate the structural performance of a prototype car-carrier, a significant amount of future work is warranted. Recommendations for future work are stated below.

11.2.1 Recommendations on the Field Test

1. Tri-axial accelerometers should be used in any further field tests. This will aid in providing better prescribed motions for FE modeling and providing more comprehensive field test data.

2. In order to gather more realistic vibration data for the suspension frames, at least four measuring points (one at each corner of the suspension frames) should be instrumented with accelerometers. Thus, five tri-axial accelerometers are needed to measure the vibration data for input to the FE model and another
three to five tri-axial accelerometers should be used to measure vibration data for other points of the trailer.

11.2.2 Recommendations on the FE Model Development and Validation

1. Additional research investigating the numerical use of spotwelds in the vehicle carrier as methods of joining entities should be conducted. The use of spotwelds in numerical models can potentially give rise to “stress risers” due to the application of point loads through single point load paths which develop. Different methods of joining finite elements should be investigated and an evaluation of the effectiveness of these numerical methods in the analysis of vehicle carriers should be considered.

2. In the FE model, a finer mesh with more spotweld connections is desirable around the stress concentration regions in order to reduce unrealistic stress/strain increases.

3. Longer simulation time with the inclusion of damping within the FE model is desirable (extend to approximately one second after the impact between the rear wheels and the bump) for better comparison between the field test results and the numerical simulations. After the impact the trailer structures vibrate independent of the boundary conditions, for both the field test and numerical condition. Thus, the simulation after the impact is free of the effects of the input discrepancies with the test.

4. Possible use of datum smoothing or filtering techniques should be considered to better analyze the vibration data for both experimental and numerical testing methods.
11.2.3 Recommendations on Structural improvements

This research project represents a first attempt in applying current engineering tools to an automotive device which traditionally has been designed and developed using past experience and a "trial and error" approach. Some structural improvement suggestions are stated below.

1. To maximize the potential of the FE method for the trailer structural improvement, an elaborate combination of local and global modifications and iterative investigations should be considered until the most acceptable results are reached.

2. As computing power increases and software development results in more robust numerical analysis codes, the use of large numerical models with greater complexities and non-linearities can be explored. For example, in this research, loaded vehicles were replaced with simplified "point masses" and blocks. This simplification may potentially give rise to incorrect observations which do not completely describe the structural response of the trailer. Further model development is needed and should continue to aid in performing a more complete, thorough, and accurate structural analysis.
REFERENCES:


APPENDIX A
PARAMETER CALCULATIONS USING MATHCAD FOR
ASTM A 36 AND ALUMINIUM 5052 H32

ASTM A36

Define

\[ \text{ksi} := 1000 \text{psi} \quad \text{KN} := 1000 \text{N} \]

Mass density

\[ \rho := \frac{7800 \text{kg}}{\text{m}^3} \quad \rho := \frac{7.8 \times 10^3 \text{kg}}{\text{m}^3} \]

Young's modulus

\[ E := 210 \times 10^9 \text{Pa} \quad E := \frac{210 \text{KN}}{\text{mm}^2} \]

Poisson's ratio

\[ \nu := 0.28 \]

Yield strength

\[ Y := 36 \text{ksi} \quad Y := \frac{0.248 \text{KN}}{\text{mm}^2} \]

Strain at yield point

\[ \varepsilon_Y := \frac{Y}{E} \]

Engineering tensile strength

\[ T_E := 64 \text{ksi} \]

Engineering strain (elongation) at tensile point

\[ \varepsilon_E := 0.21 \]

True tensile strength

\[ T := T_E \left( 1 + \varepsilon_E \right) \]

True strain at tensile point

\[ \varepsilon_T := \ln \left( 1 + \varepsilon_E \right) \]

Tangent modulus

\[ \text{ETAN} := \frac{T - Y}{\varepsilon_T - \varepsilon_Y} \quad \text{ETAN} := \frac{1.508 \text{KN}}{\text{mm}^2} \]
Aluminium 5052 H32

Define \( \text{ksi} := 1000 \text{psi} \quad \text{KN} := 1000 \text{N} \)

Mass density \( \rho := 2711 \frac{\text{kg}}{\text{m}^3} \)

Young's modulus \( E := 71 \cdot 10^9 \text{Pa} \)

Poisson's ration \( \nu := 0.333 \)

Yield strength \( Y := 195 \cdot 10^6 \text{Pa} \)

Strain at yield point \( \varepsilon_Y := \frac{Y}{E} \)

Engineering tensile strength \( T_E := 230 \cdot 10^6 \text{Pa} \)

Engineering strain (elongation) at tensile point \( \varepsilon_E := 0.25 \)

True tensile strength \( T := T_E \left(1 + \varepsilon_E\right) \)

True strain at tensile point \( \varepsilon_T := \ln \left(1 + \varepsilon_E\right) \)

Tangent modulus \( \frac{T - Y}{\varepsilon_T - \varepsilon_Y} \)
APPENDIX B
CALIBRATION FACTOR CALCULATION PROCESS

Read in data starting at time 0:00:10D.  ORIGIN = 1

Data_1 := READPRN("1.txt")

\[ \text{Time} = \frac{\text{Data}_1^{(i)}}{\text{rows(Data}_1) - 1} \cdot 3 \text{ s} \]
\[ \text{msec} := \frac{1}{1000} \text{ s} \]

Read in data starting at time 1:47:10D

Data_2 := READPRN("2.txt")
Read in data starting at time 4:25, ID9

\[
\text{Data}_3 := \text{READPRN}(*3.txt*)
\]

Read in data starting at time 5:28, ID4

\[
\text{Data}_4 := \text{READPRN}(*4.txt*)
\]
Max_Channel_1 := \max(Data_1^{(2)})
Max_Channel_2 := \max(Data_2^{(3)})
Max_Channel_3 := \max(Data_3^{(4)})
Max_Channel_4 := \max(Data_4^{(5)})

Max_Channel_1 = 0.16
Max_Channel_2 = 0.175
Max_Channel_3 = 0.182
Max_Channel_4 = 0.702

v := \frac{14.14 \text{mm}}{s}
a := \frac{14.14 \text{m}}{s^2}

Max_Channel_1 = 0.16
Max_Channel_2 = 0.175
Max_Channel_3 = 0.182
Max_Channel_4 = 0.702

C_1 := \frac{v}{\text{Max_Channel}_1}
C_2 := \frac{a}{\text{Max_Channel}_2}
C_3 := \frac{a}{\text{Max_Channel}_3}
C_4 := \frac{a}{\text{Max_Channel}_4}

C_1 = 0.088 \frac{\text{m}}{s}
C_2 = 80.985 \frac{\text{m}}{s^2}
C_3 = 77.735 \frac{\text{m}}{s^2}
C_4 = 20.145 \frac{\text{m}}{s^2}
APPENDIX C

AVERAGE AND RMS VALUE CALCULATIONS USING MATHCAD FOR CONDITION 3.

Calculate test lateral acceleration at top point for Test Scenario 3

1. List Calibration factors for the four channels:

\[
\begin{align*}
C1 &= 0.088 \frac{m}{s} \\
C2 &= 80.985 \frac{m}{s^2} \\
C3 &= 77.73 \frac{m}{s^2} \\
C4 &= 20.145 \frac{m}{s^2}
\end{align*}
\]

ORIGIN = 1

2. Input test data for a duration of 0.25 seconds:

\[
msec := 1 \times 10^{-3} \text{s}
\]

\[
\text{\textit{t}} := \text{l:\\Diagonal id18.bdt}
\]

\[
\text{\textit{t}}_{13} := \frac{\text{test3}_{1} - 4090}{9048} \cdot 3 \text{s}
\]

3. Define test data:

Hitch pin region vertical velocity:

\[
\text{Pin3} := \text{test3}_{2} \cdot 10 \times C1
\]

Left suspension vertical acceleration:

\[
\text{Left3} := \text{test3}_{3} \cdot 0.5 \times C2
\]

Right suspension vertical acceleration:

\[
\text{Right3} := \text{test3}_{4} \cdot 0.5 \times C4
\]

Top point lateral acceleration:

\[
\text{Top3} := \text{test3}_{4} \cdot C3
\]

4. Calculate the average and root mean square values of the lateral acceleration:

\[
\begin{align*}
\text{ExpT3}_{\text{avg}} &= \frac{1}{\text{max}(\text{Top3})} \sum_{m=2}^{\text{rows}(\text{Top3})-1} \left( \frac{\text{Top3}_{m} + \text{Top3}_{m-1}}{2} \right) \left( \text{T13}_{m} - \text{T13}_{m-1} \right) \\
\text{ExpT3}_{\text{rms}} &= \sqrt{\frac{1}{\text{max}(\text{Top3})} \sum_{m=2}^{\text{rows}(\text{Top3})-1} \left( \frac{\text{Top3}_{m} + \text{Top3}_{m-1}}{2} \right)^2 \left( \text{T13}_{m} - \text{T13}_{m-1} \right)}
\end{align*}
\]

\[
\text{ExpT3}_{\text{avg}} = 0.099
\]

\[
\text{ExpT3}_{\text{rms}} = 2.176
\]
Calculate simulation lateral acceleration at top point for Test Scenario 3

1. Input simulation data at the top point for a duration of 0.25 seconds:

\[ \text{SimulationT3} := \text{READPRN} \left( \text{"Dia_36353.txt"} \right) \quad \text{SimT3} := \text{SimulationT3} \cdot \frac{\text{mm}}{\text{msec}} \quad \text{Ts3} := \text{SimulationT3} \cdot \text{msec} \]

2. Calculate the average and root mean square values of unfiltered data:

\[ \text{SimT3avg} := \frac{1}{\max(Ts3)} \sum_{m=2}^{\text{rows(SimT3)}-1} \left( \frac{\text{SimT3}_m + \text{SimT3}_{m-1}}{2} \right) \left( \text{Ts3}_m - \text{Ts3}_{m-1} \right) \]

\[ \text{SimT3rms} := \sqrt{\frac{1}{\max(Ts3)} \sum_{m=2}^{\text{rows(SimT3)}-1} \left( \frac{\text{SimT3}_m + \text{SimT3}_{m-1}}{2} \right)^2 \left( \text{Ts3}_m - \text{Ts3}_{m-1} \right)} \]

Comparison of Simulation Results with the Test Data at the Top Point

![Curves of Simulation and Test Data](image)
APPENDIX D
PORTION OF THE LS-DYNA INPUT FILE

$ LS-DYNA (950) DECK WRITTEN BY : eta/FEMB-PC version 27.0
$ ENGINEER :
$ PROJECT :
$ UNITS : MM, KG, MSEC, KN
$ TIME : 07:15:59 PM
$ DATE : Monday, February 10, 2003

*KEYWORD

*TITLE
LS-DYNA USER INPUT

*CONTROL CARD

*CONTROL TERMINATION

*CONTROL_TIMESTEP

*CONTROL ENERGY

*DATABASE CONTROL FOR ASCII

*DATABASE_OGLSTAT

*DATABASE MATSUM

*DATABASE_SLEOUT

*DATABASE_NODEOUT

*DATABASE_BINARY_D3PLOT

*DATABASE HISTORY CARDS
```plaintext
*DATABASE_HISTORY_NODE_SET

$OUTPUT

$ NSID1  NSID2  NSID3  NSID4  NSID5  NSID6  NSID7  NSID8

4

$ PART CARDS

$ PART

$ HEADING TUBE134

$ PID  SECID  MID  EOSID  HGID  GRAV  ADPOPT  TMID

 1  4  3  0  0

$ PART

$ HEADING TUBE3/16

$ PID  SECID  MID  EOSID  HGID  GRAV  ADPOPT  TMID

 2  1  2

$ PART

$ HEADING CCP2R

$ PID  SECID  MID  EOSID  HGID  GRAV  ADPOPT  TMID

 251  8  1  0  0

$ SECTION CARDS

$ SECTION SHELL

$ P3/16

$ SECID  EFORM  SHRF  NIP  PROPT  QR/IRID  ICOMP  SETYP

 1  1  1.0  2  0.0  0.0  0  1

$ T1  T2  T3  T4  NLOC

  4.7625  4.7625  4.7625  4.7625  0

$ SECTION SOLID

$ SOLID

$ SECTION SHELL

$ P1/8

$ SECID  EFORM  AET

 2  1  0

$ SECTION SHELL

$ P134

$ SECID  EFORM  SHRF  NIP  PROPT  QR/IRID  ICOMP  SETYP

 3  1  1.0  2  0.0  0.0  0  1

$ T1  T2  T3  T4  NLOC

  3.175  3.175  3.175  3.175  0

$ SECTION SHELL

$ P180

$ SECID  EFORM  SHRF  NIP  PROPT  QR/IRID  ICOMP  SETYP

 5  1  1.0  2  0.0  0.0  0  1

$ T1  T2  T3  T4  NLOC

  4.572  4.572  4.572  4.572  0

$ SECTION SHELL

$ P120

$ SECID  EFORM  SHRF  NIP  PROPT  QR/IRID  ICOMP  SETYP

 6  1  1.0  2  0.0  0.0  0  1

```

116
<table>
<thead>
<tr>
<th>SECID</th>
<th>EFORM</th>
<th>SHRIF</th>
<th>NIP</th>
<th>PROPT</th>
<th>QR/IRID</th>
<th>ICOMP</th>
<th>SETYP</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>1</td>
<td>0.0</td>
<td>6.35</td>
<td>6.35</td>
<td>0.0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>0.0</td>
<td>7.9375</td>
<td>7.9375</td>
<td>0.0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>9</td>
<td>1</td>
<td>0.0</td>
<td>9.525</td>
<td>9.525</td>
<td>0.0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>0.0</td>
<td>12.7</td>
<td>12.7</td>
<td>0.0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>11</td>
<td>1</td>
<td>0.0</td>
<td>25.4</td>
<td>25.4</td>
<td>0.0</td>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>

*SECTION_BEAM*

<table>
<thead>
<tr>
<th>SECID</th>
<th>EFORM</th>
<th>SHRIF</th>
<th>QR/IRID</th>
<th>CST</th>
<th>SCOOR</th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
<td>6</td>
<td>2.0</td>
<td>0.0</td>
<td>1.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

**VOL**

- **MATERIALS**

**MAT_RIGID**

<table>
<thead>
<tr>
<th>MID</th>
<th>RO</th>
<th>E</th>
<th>PR</th>
<th>N</th>
<th>COUPLE</th>
<th>M</th>
<th>ALIAS</th>
</tr>
</thead>
<tbody>
<tr>
<td>10.000000783</td>
<td>207.0</td>
<td>0.28</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

**SLCO_OR_A1**

<table>
<thead>
<tr>
<th>MID</th>
<th>RO</th>
<th>E</th>
<th>PR</th>
<th>SIGY</th>
<th>ETAN</th>
<th>BETA</th>
</tr>
</thead>
<tbody>
<tr>
<td>20.000000783</td>
<td>210</td>
<td>0.28</td>
<td>0.317</td>
<td>0.85</td>
<td>1.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

**MAT_PLASTIC_KINEMATIC**

<table>
<thead>
<tr>
<th>MID</th>
<th>RO</th>
<th>E</th>
<th>PR</th>
<th>SIGY</th>
<th>ETAN</th>
<th>BETA</th>
</tr>
</thead>
<tbody>
<tr>
<td>30.000000783</td>
<td>210</td>
<td>0.28</td>
<td>0.345</td>
<td>0.913</td>
<td>1.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>
### *MAT_Plastic_Kinematic*

**A36**

<table>
<thead>
<tr>
<th>MID</th>
<th>RO</th>
<th>E</th>
<th>PR</th>
<th>SIGY</th>
<th>ETAN</th>
<th>BETA</th>
</tr>
</thead>
<tbody>
<tr>
<td>40.00000783</td>
<td>210</td>
<td>0.28</td>
<td>0.248</td>
<td>1.508</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>

**SRC  | SRP  | FS  | VP  |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### *MAT_Plastic_Kinematic*

**A53F**

<table>
<thead>
<tr>
<th>MID</th>
<th>RO</th>
<th>E</th>
<th>PR</th>
<th>SIGY</th>
<th>ETAN</th>
<th>BETA</th>
</tr>
</thead>
<tbody>
<tr>
<td>50.00000783</td>
<td>210</td>
<td>0.28</td>
<td>0.207</td>
<td>0.931</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>

**SRC  | SRP  | FS  | VP  |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### *MAT_Plastic_Kinematic*

**Aluminum**

<table>
<thead>
<tr>
<th>MID</th>
<th>RO</th>
<th>E</th>
<th>PR</th>
<th>SIGY</th>
<th>ETAN</th>
<th>BETA</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>2.71E-06</td>
<td>71</td>
<td>0.334</td>
<td>0.195</td>
<td>0.42</td>
<td>1</td>
</tr>
</tbody>
</table>

**SRC  | SRP  | FS  | VP  |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### *MAT_Plastic_Kinematic*

**A519 (1026)**

<table>
<thead>
<tr>
<th>MID</th>
<th>RO</th>
<th>E</th>
<th>PR</th>
<th>SIGY</th>
<th>ETAN</th>
<th>BETA</th>
</tr>
</thead>
<tbody>
<tr>
<td>70.00000783</td>
<td>210</td>
<td>0.28</td>
<td>0.241</td>
<td>1.429</td>
<td>1</td>
<td></td>
</tr>
</tbody>
</table>

**SRC  | SRP  | FS  | VP  |
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### *MAT_Rigid*

**Rigid_Mass**

<table>
<thead>
<tr>
<th>MID</th>
<th>RO</th>
<th>E</th>
<th>PR</th>
<th>N</th>
<th>COUPLE</th>
<th>M</th>
<th>ALIAS</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>0.0031752</td>
<td>207.0</td>
<td>0.28</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>CMO</th>
<th>CON1</th>
<th>CON2</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

### *MAT_Isotropic_Elastic_Plastic*

**Tire**

<table>
<thead>
<tr>
<th>MID</th>
<th>RO</th>
<th>G</th>
<th>SIGY</th>
<th>ETAN</th>
<th>BULK</th>
</tr>
</thead>
<tbody>
<tr>
<td>9</td>
<td>0.0000010</td>
<td>82.0</td>
<td>2.5</td>
<td>1.5</td>
<td>160.0</td>
</tr>
</tbody>
</table>

### *MAT_Cable_Discrete_Beam*

**Cable**

<table>
<thead>
<tr>
<th>MID</th>
<th>RO</th>
<th>E</th>
<th>LCID</th>
<th>F0</th>
</tr>
</thead>
<tbody>
<tr>
<td>100.00000783</td>
<td>207.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.88964</td>
</tr>
</tbody>
</table>

### SEGMENT_SET CARDS

**N_Deck**

<table>
<thead>
<tr>
<th>SID</th>
<th>DA1</th>
<th>DA2</th>
<th>DA3</th>
<th>DA4</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>N1</th>
<th>N2</th>
<th>N3</th>
<th>N4</th>
<th>A1</th>
<th>A2</th>
<th>A3</th>
<th>A4</th>
</tr>
</thead>
<tbody>
<tr>
<td>55178</td>
<td>55179</td>
<td>55180</td>
<td>55173</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>55174</td>
<td>55181</td>
<td>55182</td>
<td>55175</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>55176</td>
<td>55175</td>
<td>55182</td>
<td>55183</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>55177</td>
<td>55176</td>
<td>55183</td>
<td>55184</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>55180</td>
<td>55181</td>
<td>55174</td>
<td>55173</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>

### NODE_SET CARDS
```
*SET_NODE_LIST
$*SUS_LEFT
$ S  SID  DA1  DA2  DA3  DA4
  2  0.0  0.0  0.0  0.0  0.0
$  NID1  NID2  NID3  NID4  NID5  NID6  NID7  NID8
  6571  30868  6596  6589  30876  6611  6582  30877
  6614  30881  6591  30884  6627  30887  30865  30866
  30869  30870  30871  30873  30874  30875  30878  30879
  30880  30882  30883  30885  30886  30888  6531  30916
  30922  6548  30924  6098  6545  30925  6101  6552
  30929  6557  30932  6105  30935  30913  30914  30917
  30918  30919  30921  30922  30923  30926  30927  30928
  30930  30931  30933  30934  30936

$*SUS_RIGHT
$ S  SID  DA1  DA2  DA3  DA4
  3  0.0  0.0  0.0  0.0  0.0
$  NID1  NID2  NID3  NID4  NID5  NID6  NID7  NID8

*SET_NODE_LIST
$*PART_SET_LIST

*SET_NODE_LIST
$*D_F
$ S  SID  DA1  DA2  DA3  DA4
  1  0.0  0.0  0.0  0.0  0.0
$  PID1  PID2  PID3  PID4  PID5  PID6  PID7  PID8
    109    110

*SET_NODE_LIST
$*DF
$ S  SID  DA1  DA2  DA3  DA4
  2  0.0  0.0  0.0  0.0  0.0
$  PID1  PID2  PID3  PID4  PID5  PID6  PID7  PID8
    159    160

*BOUNDARY_PRESCRIBED_CARDS

*BOUNDARY_PRESCRIBED_MOTION_SET
$*PRESCRIBED
$  NSID  DOF  VAD  LCID  SF  VID  DEATH  BIRTH
  2   3   1   1   1.0  1.000E+28  0.0

*BOUNDARY_PRESCRIBED_MOTION_SET
$*PRESCRIBED
$  NSID  DOF  VAD  LCID  SF  VID  DEATH  BIRTH
  3   3   1   2   1.0  1.000E+28  0.0

*BOUNDARY_PRESCRIBED_MOTION_SET
$*PRESCRIBED
$  NSID  DOF  VAD  LCID  SF  VID  DEATH  BIRTH
  186   3   0   3   1.0  1.000E+31  0.0

*CONstrained_EXTRA_NODES_CARDS

*CONstrained_EXTRA_NODES_SET
$*F_B1/4
$  PID  NSID

```
*CONSTRAINED_SPOTWELD
$ M1 M2 SN SS N M TF EP
128764 26787 0.0 0.0 0.0 0.0 0.0 0.0
$-------1-+-2-+-3-+-4-+-5-+-6-+-7-+-8
$
$
 INITIAL VELOCITY CARDS
$
$
$-------1-+-2-+-3-+-4-+-5-+-6-+-7-+-8
*INITIAL VELOCITY
$*VELOCITY
$ NSID NSIDX BOXID
$
$ VX VY VZ VXR VYR VZR
-13.889 0.0 0.0 0.0 0.0 0.0
$-------1-+-2-+-3-+-4-+-5-+-6-+-7-+-8
$
$
 CONTACT CARDS
$
$
$-------1-+-2-+-3-+-4-+-5-+-6-+-7-+-8
*CONTACT_AUTOMATIC NODES TO SURFACE TITLE
$
$ CID
1
$
$ SSID MSID SSTYPE MSTYP SBOXID MBOXID SPR MPR
1 15 2 3 0 0
$
$ FS FD DC VC VDC PENCHK BT DT
0.30 0.20 0.0 0.0 0.0 20 0 0 0.0 1.00E+23
$
$ SFS SFM SST MST SPST SPMT FSF VSF
10.0 10.0 0.0 0.0 1.0 1.0 1.0 1.0 1.0
$
$ SOFT SOFSCL LCIDAB MAXPAR PENTOL DEPTH BSORT PRCFRQ
0 0.10 1.025 0.0 2.0 0 1
$ PENMAX THKOPT SHLTHK SNLOG ISYM I2D3D
0.0 0 0 0 0 0
$
$
 LOAD BODY CARDS
$
$
$-------1-+-2-+-3-+-4-+-5-+-6-+-7-+-8
*LOAD_BODY_Z
$*BODY
$
$ LCID SF LCIDD RT XC YC ZC
4 1.0 0.0 0.0 0.0
$-------1-+-2-+-3-+-4-+-5-+-6-+-7-+-8
$
$
 LOAD CURVE CARDS
$
$
$-------1-+-2-+-3-+-4-+-5-+-6-+-7-+-8
*DEFINE_CURVE
$*LEFT_WHEEL
$
$ LCID SIDR SFA SFO OFFA OFFO DATTPY
1 0 1000.0 0.0010 0.0 0.0 0.0
$ A1 01
0 0
0.00066713 1.4085
0.00099469 1.8661
$
$
*DEFINE_CURVE
$*RIGHT_WHEEL
$
$ LCID SIDR SFA SFO OFFA OFFO DATTPY
VITA AUCTORIS

Name: Zhanbiao Li

Place of Birth: Honghu City, Hubei Province, China

Date of Birth: March 10th, 1966

Education: Huazhong University of Science and Technology, China

Degree: Mechanical Engineering. Graduated, B.A.Sc. In July 1987