Design and Analysis of 40 Tonne Trailer Used in Heavy Commercial Vehicles

Gajanan S. Datar, Rupa S. Bindu & Durvesh V. Dandekar
Dept. of Mechanical engineering, D Y Patil College of Engineering, Pune, India.
SKN SInhgad Institute of Tech and science Lonawala pune
E-mail : gajanan.datar@gmail.com & dvdandekar.sknsits@gmail.com

Abstract - The trailer is main part of a full scale vehicle model. This mainly consists of wheel and axle assembly, landing gears and the most important is frame structure. This trailer frame consists of two main I-sections & end-channel sections, connected by cross members of channel sections. Geometry of various parts has to be modified in consultation with customer to avoid interference.

In this study, the purpose of the vehicle is to carry the heavy load of about 40T. Herein the state of trailer will be static or stable. This study addresses the problem of a different kind of trailer frame. This has to sustain and carry a heavy load of 40T. Calculating the behavior of these loads at static, dynamic conditions on the frame is a challenging task. Therefore, study of static and dynamic calculation of chassis frame is very important from the structural point of view. To achieve this, detailed static and dynamics analysis is carried out on the frame structure of chassis and to understand the characters of the frame at those loads.

Keywords : ISBM, ISMC, structural analysis, dynamic analysis.

I. INTRODUCTION

Generally, truck is any of various heavy motor vehicles designed for carrying or pulling loads. Light duty trucks carry load up to 60 kN, medium duty are for 67kN to 147kN, and heavy duty trucks carry load above 147kN. Most trucks use ladder-type frames with side rails and cross members, and the frames are subjected to three types of loads: vertical, torsional, and side. Side rails support vertical and side loads such as engine, transmission, fuel tanks, battery boxes, suspensions, bodies, work equipment, and cargo. The cross members prevent the side rails from twisting with side loads such as the fuel tank and battery box. A frame section supported at each end loaded in the middle is in compression at the top and in tension at the bottom.

a) Problem Definition for Concept Design

In this project work the truck trailer frame is analyzed for LINEAR STATIC. A heavy load of 40 tonnes (392.4kN) is applied to the frame uniformly and different iterations are carried out to get stresses in safe limit.

Tyre diameter = 1016 mm
Tyre equivalent stiffness = 941.43799 N/mm
Trailer Frame Mass (alone) = 5000 kg

Vehicle speed = 35-45 Km/h
Ditch depth on road = 50mm

Chassis Mass=Density×Volume = 5000Kg
Equivalent Chassis Area = 66265.36531

b) Road patches considered for Concept design.

Plain Road Without Any Ditch

Virtual Proving Road With Continuous Irregular Ditches Upto 50 Mm Depths

Road surface has ditch of 50 mm depth

Table 1.1 Time Pulse

<table>
<thead>
<tr>
<th>Time (sec)</th>
<th>Ditch Depth (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0.05</td>
<td>10</td>
</tr>
<tr>
<td>0.08</td>
<td>22</td>
</tr>
<tr>
<td>0.1</td>
<td>50</td>
</tr>
<tr>
<td>0.3</td>
<td>18</td>
</tr>
<tr>
<td>0.4</td>
<td>12</td>
</tr>
<tr>
<td>0.5</td>
<td>6</td>
</tr>
</tbody>
</table>

2nd tire time taken to travel 1800 mm 0.216 seconds ~ 0.2 seconds
2nd Tire travel in ditch starts from 0.7 seconds to 1.2 seconds
3rd tire time taken to travel 3600 mm 0.41376 seconds ~ 0.4 seconds
3rd Tire travel in ditch starts from 1.6 seconds to 2.1 seconds
4th tire time taken to travel 1403 mm 0.16836 seconds ~ 0.15 seconds
4th Tire travel in ditch starts from 2.25 seconds to 2.75 seconds
Output Time for Results considered is 3-4 times of input excitations.
10 seconds is taken as total time to measure the system response.

Case 1

Figure 1.3.1 Tire configuration 2:2 concentrated load applied

Figure 1.3.2 Graph displaying displacement vs. time at 3 points

Case 2

Figure 1.3.3 Tire configuration 1:3 concentrated load applied

Figure 1.3.4 Graph displaying displacement vs. time at 3 points

Case 3

Figure 1.3.5 Tire configuration 2:2 uniformly distributed load applied

Figure 1.3.6 Graph displaying displacement vs. time at 3 points
c) Concept Design Requirements

1. Induced vibration at point A, B and C in 2-2 and 1-3 cases with point mass at middle
2. Induced vibration at point A, B and C with same 20000 kg with UDL
3. System Response Graph for Conclusion on configuration of Tire System

Total mass of chassis = 5 tonnes = 5000 kg

Chassis volume = 11094720000 mm³
Density = Mass/Volume = 7.84999×10⁻⁹ ton/mm³
Young’s modulus = 2×10⁵ MPa
Poisson’s Ratio = 0.29

Single Tyre equivalent stiffness co-efficient = 1.75×10⁶ N/m = 1.75×10³ N/mm
Per axle tyres stiffness = 1.75×10⁶ N/mm × 4 tires (left & right, 2 one each side) = 7000 N/mm

Number of tyres used = 16.

4 Axle wt = 1500kg
1 axle wt = 1500/4 = 375 kg
1 Axle Mass at tire mounting locations = 375 kg = 0.375 tonnes

d) Objectives of the Present Study

The following are the objectives of the study:

- To generate a surface model with standard sections suitable for linear static analysis.
- To generate a finite element model of the same.
- To carry out all the necessary checks on the model.
- To carry out the linear analysis to study the behavior of the frame.
- To determine the natural frequencies and mode shapes
- Validate the model for the limiting load (permissible load)

e) Scope of the PRESENT study

1. This study addresses the problem of a different kind of trailer frame with different sections. This has to sustain and carry a heavy load of 40T.
2. To achieve this, detailed static and dynamic analysis is carried out on the frame structure of trailer to understand the characteristics of the frame at those loads.
3. The normal modes analysis is also carried out to understand the different mode shapes and to get the natural frequency of the model.

II. MATERIALS AND PROPERTIES FOR THE CONFIGURATION

In this work the material used is linear, elastic, homogeneous and isotropic. In addition a reliable assumption is made that the material is continuum (contains no gaps or voids) and that all material properties remain constant (many material properties are functions of temperature; thus, here also inferred that the temperature of the structure is also constant).

The material and properties such as Young’s modulus (E), Poisson’s Ratio (ν) and density (ρ) for the structure are given in the Table-3.5.1
### Table- 2.1: Material Properties Used in the Finite Element Model[9]

<table>
<thead>
<tr>
<th>S N</th>
<th>Section Used</th>
<th>Material</th>
<th>Young’s Modulus ‘E’ N/mm²</th>
<th>Poisson’s Ratio μ</th>
<th>Density (RHO) ρ kg/m³</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ISMB 250</td>
<td>Steel</td>
<td>2×10⁵</td>
<td>0.3</td>
<td>7894</td>
</tr>
<tr>
<td>2</td>
<td>ISMB 350</td>
<td>Steel</td>
<td>2×10⁵</td>
<td>0.3</td>
<td>7894</td>
</tr>
<tr>
<td>3</td>
<td>ISMB 450</td>
<td>Steel</td>
<td>2×10⁵</td>
<td>0.3</td>
<td>7894</td>
</tr>
<tr>
<td>4</td>
<td>ISMC 350</td>
<td>Steel</td>
<td>2×10⁵</td>
<td>0.3</td>
<td>7894</td>
</tr>
</tbody>
</table>

#### III. LINEAR STATIC ANALYSIS

**a) Introduction**

The linear analysis is done for trailer frame structure in its horizontal condition by specifying the load in terms of ‘Newton’ and other dimensions by ‘mm’. The constraints applied are multi point constraints. The load 40T (392.4kN) is applied uniformly on the frame structure. The analysis file is attached to the solver. After solving the model, the .xdb file is attached to the result file. The results can be viewed to the desired condition.

![Figure 3.1.1. Linear stress displayed in software](image1)

**Figure 3.1.1. Linear stress displayed in software**

![Figure 3.1.2. Linear deformation displayed in software](image2)

**Figure 3.1.2. Linear deformation displayed in software**

#### IV. ANALYTICAL VALIDATION

**a) Analytical Linear Static Calculations for 40tonne Load on Trailer Frame.**

Frame Parameters.

- a. Length L = 15.14 m
- b. Width W = 2.4 m
- c. Area = 15.14 × 2.4 = 36.336 sqm

Total load on the truck frame = 40 tonne

Load on per unit area = \( \frac{(40\times10)}{36.336} = 11.01 \) kN/m²

**b) Assumptions**

- The material is homogeneous and isotropic.
- The moduli of elasticity in tension and compression are equal.
- Plane sections remain plane even after bending.
• The secondary member i.e. ISMB 250 is considered as only connecting member.
• So, the load coming is distributed between Primary members.

c) Calculations

**Frame member ISMB 450 design:**
Span of load acting on the member
\[ = \frac{(0.65+1.1)}{2} = 0.875 \, \text{m} \]
Load per meter run \( \text{KN/m} = 9.6 \, \text{kN/m} \)

**Frame member ISMB 350 design:**
Span of load acting on the member
\[ = \frac{(2.4-1.1)}{2} = 0.65 \, \text{m} \]
Load per meter run \( \text{kN/m} = 11.01 \times 0.65 = 7.2 \, \text{kN/m} \)

**Frame member ISMB 250 design**
Max span of member \( = \frac{(1.6+1.2)}{2} = 1.4 \, \text{m} \)
Load per meter run \( = 11.01 \times 1.4 = 15.4 \, \text{kN/m} \)

**Load diagram:**

![Load diagram](image)

**Fixed end moments**

\[ \text{Maa'} = \frac{(WL^2)}{2} = \frac{(9.6 \times 4.4^2)}{2} = 93.24 \, \text{kN/m} \]
\[ \text{Mab} = \text{Mba} = \frac{(WL^2)}{12} = \frac{(9.6 \times 6^2)}{12} = 28.90 \, \text{kN/m} \]
\[ \text{Mcb} = \text{Mcd} = \frac{(WL^2)}{12} = \frac{(0.65 \times 1.37^2)}{12} = 0.10 \, \text{kN/m} \]
\[ \text{Mdc} = \text{Mdd'} = \frac{(WL^2)}{12} = \frac{(9.6 \times 1.37^2)}{12} = 1.51 \, \text{kN/m} \]
\[ \text{Mdd'} = \frac{(WL^2)}{2} = \frac{(9.6 \times 2.085^2)}{2} = 20.94 \, \text{kN/m} \]

**Bending moment diagram**

![Bending moment diagram](image)

**Shear Force Diagram**

![Shear Force diagram](image)

Grade of steel \( \sigma_y = 250 \, \text{MPa} \)
Length of member \( = 8.8 \, \text{m} \)
Max Bending in z direction \( M_z = 93.25 \, \text{kN-m} \)
Max Bending in z direction \( M_z = 12.25 \, \text{kN-m} \)
Permissible Bending stress \( \sigma_{bc} = 0.66 \sigma_y = 0.66 \times 250 = 165 \, \text{MPa} \)

**ISMB450**

Properties of the section
Area \( = 92.27 \times 10000 = 922700 \, \text{mm}^2 \)
Ixx \( = 30390.8 \times 10^4 = 3.04 \times 10^8 \, \text{mm}^4 \)
Iyy \( = 8340000 \, \text{mm}^4 \)
Zxx \( = 1350.7 \times 1000 = 1350700 \, \text{mm}^3 \)
Zyy \( = 111.2 \times 1000 = 111200 \, \text{mm}^3 \)
Slenderness ratio \( \lambda = \frac{(8.8 \times 1000)}{185.1} = 48 \)

**From IS 800**

Table 5.1 σat permissible = 153 MPa
Actual bending stress \( \sigma_{bc} = \frac{93.25 \times 10^6}{1350700} = 69.038 \, \text{MPa} \)

**Secondary Beam Stress ISMB 250**

Grade of steel \( \sigma_y = 250 \, \text{MPa} \)
Length of member \( = 1.1 \, \text{m} \)
Bending in z direction \( M_z = 15.4 \times 1.1^2 / 10 = 1.864817 \, \text{kN-m} \)
Permissible Bending stress \( \sigma_{bc} = 0.66 \sigma_y = 165 \, \text{MPa} \)

**ISMB250**

Properties of the section
Area \( = 47.55 \times 10000 = 475500 \, \text{mm}^2 \)
Ixx \( = 5131.6 \times 10^4 = 51316000 \, \text{mm}^4 \)
th a safety factor

From IS 800 Table 5.1 σat permissible = 201 MPa

Actual bending stress \( \sigma_{bc}^{zz} \) actual

\[ = (1.864817 \times 10^6)/410500 = 4.5 \text{ MPa} \]

**Secondary beam design:**

- **Grade of steel** \( \sigma_y = 250 \text{ MPa} \)
- **Length of member** = 1.1 \text{ m}
- **Bending in z direction** \( M_z = 2.33 \text{ MPa} \)
- **Permissible Bending stress** \( \sigma_{bc} = 0.66 \sigma_y \)
  \[ = 0.66 \times 250 = 165 \text{ MPa} \]

**ISMB250**

**Properties of the section**

- **Area** = 47.55 \times 10^3 \text{ mm}^2
- **Ixx** = 5131.6 \times 10^4 \text{ mm}^4
- **Iyy** = 334.5 \times 10^4 \text{ mm}^4
- **Zxx** = 410.5 \times 10^4 \text{ mm}^3
- **Zyy** = 53.5 \times 10^4 \text{ mm}^3

**Slenderness ratio** \( \lambda = (1.1 \times 1000)/10.39 = 105.871 \)

**σa permissible** = 201 MPa

Actual bending stress \( \sigma_{bc}^{zz} \) actual

\[ = (2.33 \times 10^6)/410500 = 5.67 \text{ MPa} \]

**V. DURABILITY ANALYSIS PROCEDURE**

Durability analysis here is performed in 3 steps. First, acting forces at mounting points to the body are acquired from a quasi-static load analysis of a vehicle suspension system or from direct measuring of force. Second, static strength analysis of body is accomplished using the forces as input loads, stresses on the shell body are calculated and the durability of the body is evaluated with a safety factor against the yield stress briefly. Last, the fatigue life is predicted on the stress magnitude with load history.

\[ a) \ 3g \text{ Gravity Loading for Durability Analysis.} \]

Now a day’s vehicle is under some conflicting demands: for example, it must weigh light and also have best reliability. In automotive companies reduction in cost, developing time, and vehicle weight is an indispensable theme to promote the competitiveness and then to survive in the global market. In spite of the internal need outer situations are in reverse direction, i.e. regulations are to be more strict in the car industry and new regulations are being invented for the sake of the right of customers. A vehicle is being developed not only for one district but for the globe, so it runs on an asphalt pavement and also must go crawling into a hole and hurdling on a bump with passengers full in or some in a hot weather or cold. In the side of vehicle durability the above-mentioned circumstances will cause minus effects to the vehicle. It is surely impossible of a car to be tested on a lot of pattern road in the field. One test takes away a lot of time and money and provides only visible results. But it doesn’t often give us a clue for finding accurate causes of a trouble and it is difficult of effective answers to be put out from the test alone. So other methods were introduced on the aim of replacing or cutting-down tests in the proving ground, i.e. the laboratory test and the finite element analysis. Of laboratory durability tests the road simulator test and the multi-axial test are best similar to the proving ground test, but those are different in the input load and constraint terms. Current durability analysis is performed in 3 steps. First, acting forces at mounting points to the body are acquired from a quasi-static load analysis of a vehicle suspension system or from direct measuring of force. Second, static strength analysis of body is accomplished using the forces as input loads, stresses on the shell body are calculated and the durability of the body is evaluated with a safety factor against the yield stress briefly. Last, the fatigue life is predicted on the stress magnitude with load history [6, 7].

The method brings fast and accurate results only in few laboratory tests where the body is under the equal condition in the load and constraint to the durability analysis, but has some limitations in being applied to the vehicle durability test in the proving ground. So assumptions are made in calculating the loads; 3G on the bump/hole, 1G at braking, 1G on cornering, where G is the force acting to the tire from the road and transferring to the trailer frame when the vehicle is in gravity. New method in approaching the vehicle durability evaluation in the proving ground is studied in this paper. In stead of specific load cases, the 3-dimensional road of object was represented with shell elements to its profile. The chassis system which consists of the ladder type frame is shaped of shell and solid elements to its dimension. Chassis runs on the digitized road (virtual proving ground).

In this study the finite element chassis frame was modeled mostly close to the actual shape that was tested on the proving ground, and the test road with one cycle was represented with shell elements. For this new

---

durability analysis, the road with the 50mm ditch has been considered where the tire, axle and the frame gets into the ditch and come back with the representative time due to which the maximum 3g gravity loads will be acted on the frame.

Table 5.1 Gravity responses at respective ditch depth

<table>
<thead>
<tr>
<th>Ditch Depth (mm)</th>
<th>Gravity responses (g)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>10</td>
<td>0.6</td>
</tr>
<tr>
<td>22</td>
<td>1.32</td>
</tr>
<tr>
<td>50</td>
<td>3</td>
</tr>
<tr>
<td>18</td>
<td>1.08</td>
</tr>
<tr>
<td>12</td>
<td>0.72</td>
</tr>
<tr>
<td>6</td>
<td>0.32</td>
</tr>
</tbody>
</table>

Table 5.1 Gravity responses at respective ditch depth

![Figure 5.1.1. Maximum Stress on frame when 3g gravity loading applied.](image)

VI. RESULTS AND DISCUSSIONS

The results for different cases for 40 Tonnes of load are as follows.

a) Static Results

Table-6.1 Table showing static stress.

<table>
<thead>
<tr>
<th>Sr. No</th>
<th>Position</th>
<th>Stress Mpa</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>01</td>
<td>Front end</td>
<td>72.43</td>
<td>Minimum</td>
</tr>
<tr>
<td>02</td>
<td>Middle part</td>
<td>32.19</td>
<td>Minimum</td>
</tr>
<tr>
<td>03</td>
<td>Rear end</td>
<td>8.05</td>
<td>Minimum</td>
</tr>
</tbody>
</table>

Table-6.2 Table showing deformation

<table>
<thead>
<tr>
<th>Sr. No</th>
<th>Position</th>
<th>Deflection ‘mm’</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>01</td>
<td>Front end</td>
<td>3.68</td>
<td>Minimum</td>
</tr>
<tr>
<td>02</td>
<td>Middle part</td>
<td>1.64</td>
<td>Minimum</td>
</tr>
<tr>
<td>03</td>
<td>Rear end</td>
<td>1.19</td>
<td>Minimum</td>
</tr>
</tbody>
</table>

After observation of the deflections at different parts of frame, it can be inferred that; after modifying, the structure is safe with almost minimum deflection at major parts. At front end it is showing maximum, i.e. 3.68mm. This is very less for this large structure. This is the result obtained through the modifications in the trailer frame i.e. moving tyre locations from 4th span to 3rd span, thus subsequently changing the wheel base from 6000mm to 7200mm.

The results we obtained by this design modification are satisfactory and safe. However, the load of 40T is large enough to induce any physical disturbance to structure in road travel conditions. Considering dynamic vibrations may come from non smooth road conditions, we are processing further for optimizing stress and deflections.
b) Dynamic Stress Results

Table-6.3. Table showing Dynamic stress.

<table>
<thead>
<tr>
<th>Sr. No</th>
<th>Position</th>
<th>Stress Mpa</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>01</td>
<td>Front end</td>
<td>81.737</td>
<td>Minimum</td>
</tr>
<tr>
<td>02</td>
<td>Middle part</td>
<td>36.328</td>
<td>Minimum</td>
</tr>
<tr>
<td>03</td>
<td>Rear end</td>
<td>18.164</td>
<td>Minimum</td>
</tr>
</tbody>
</table>

As we have mentioned the structural response / deflection / stress of the system is minimum. So, we conclude the place of tyre locations at 3rd segment is still recommended looking at large length of structure.

Maximum limiting tensile stress of steel structures (AISI) is 600 N/mm². The maximum stress here is 81.737 N/mm². So this is well below the tensile strength of steel, and the structure is strong enough to take this load safely.

If this structure is taken for production purpose and here the structure shown is in whole one piece structure. As this type of one piece trailer frame is not possible or production cost will be much more, hence the only alternate procedure is to manufacture long members and cross members separately and weld it together to form a trailer frame structure. Here the concept level design and durability is satisfactory for the load of 40 ton, hence this trailer frame can now be taken to the actual analysis considering the welding and joints. The stresses are assumed to be doubled in that condition.

c) Comparison of the Static Stress Results Between Analytical and Software Used.

Static stress results shown by the software is 72.43 MPa and the actual bending stress by the analytical procedure is 69.038 MPa. Hence the software error is 4.63%.

VII. REFERENCES

1. “STRESS ANALYSIS OF HEAVY DUTY TRUCK CHASSIS USING FINITE ELEMENT METHOD”, by O Kurdi, R Abd- Rahman, M N Tamin, Faculty of Mechanical Engineering, University Technology Malaysia
15. INCLUDEPICTURE http://www.sciencedirect.com/cache/MiamiImageURL/B6V36-45CNCSX-3-1/0?wchp=dGLbVlbzSkWb"MERGEFORMATINET