Finite Element Analysis of A Ladder Chassis Frame

Avinash V. Gaikwad¹, Pravin S. Ghawade²
¹, ²Assistant Professor in Jawaharlal Darda Institute of Engg. & Tech., Yavatmal, INDIA
Email: agaikwad@rediffmail.com¹ pghawade@rediffmail.com²

Abstract — Automotive chassis is an important part of an automobile. The chassis serves as a frame work for supporting the body and different parts of the automobile. Also, it should be rigid enough to withstand the shock, twist, vibration and other stresses. Along with strength, an important consideration in chassis design is to have adequate bending stiffness for better handling characteristics. So, strength and stiffness are two important criteria for the design of the chassis. This report is the work performed towards the static structural analysis of the truck chassis. Structural systems like the chassis can be easily analyzed using the finite element techniques. So a proper finite element model of the chassis is to be developed. The chassis is modeled in PRO-E. FEA is done on the modeled chassis using the ANSYS Workbench.

Index Terms — FEA, Ladder chassis frame

I. INTRODUCTION

Automobile chassis usually refers to the lower body of the vehicle including the tires, engine, frame, driveline and suspension. Out of these, the frame provides necessary support to the vehicle components placed on it. Also the frame should be strong enough to withstand shock, twist, vibrations and other stresses. The chassis frame consists of side members attached with a series of cross members Stress analysis using Finite Element Method (FEM) can be used to locate the critical point which has the highest stress. This critical point is one of the factors that may cause the fatigue failure. The magnitude of the stress can be used to predict the life span of the truck chassis. The accuracy of prediction life of truck chassis is depending on the result of its stress analysis.

II. BASIC CALCULATION FOR CHASSIS FRAME

Model No. = 11.10 (Eicher E2)
Side bar of the chassis are made from “C” Channels with 210mm x 76 mm x 6 mm
Front Overhang (a) = 935 mm
Rear Overhang (c) = 1620 mm
Wheel Base (b) = 3800 mm
Material of the chassis is St 52
E = 2.10 x 10⁵ N /mm²
Poisson Ratio = 0.31
Radius of Gyration R = 210/2 = 105 mm
Capacity of Truck = 8 ton = 8000 kg = 78480 N
Capacity of Truck with 1.25% = 78480 N
= 98100 + 19620 = 117720 N
Chassis has two beams. So load acting on each beam is half of the Total load acting on the chassis.
Load acting on the single frame = 117720/2
= 58860 N / Beam

A. Calculation for Reaction

Chassis is simply clamp with Sock Absorber and Leaf Spring. So Chassis is a Simply Supported Beam with uniformly distributed load. Load acting on Entire span of the beam is 58860 N. Length of the Beam is 6355 mm

Uniformly Distributed Load is 58860 / 6355 = 9.262 N/mm
Now taking the reaction around the Support A.

\[ R_a = \frac{w}{2}L (1-2c)/2b \]
\[ = 9.262 \times 6355 \times (6355-2 \times 1620)/2 \times 3800 \]
\[ = 24124.85 N \]

\[ R_c = \frac{w}{2}L (1-2a)/2b \]
\[ = 9.262 \times 6355 \times (6355-2 \times 935)/2 \times 3800 \]
\[ = 34735.15 N \]

Calculation for Shear Force and Bending Moment

Shear Force

\[ V_1 = wa \]
\[ = 935 \times 9.262 \]
\[ = 8660 N \]

\[ V_2 = R_c - V_1 \]
\[ = 24124.85 - 8660 \]
\[ = 15464.88 N \]

\[ V_3 = Rd - V_4 \]
\[ = 34735.15 - 15004.44 \]
\[ = 19730.71 N \]

\[ V_4 = wc \]
\[ = 9.262 \times 1620 \]

B. Bending Moment

\[ M_1 = -\frac{wc^2}{2} \]
\[ = -9.262 \times 9352/2 \]
\[ = -4048536 \text{ N-mm} \]

\[ M_2 = -\frac{wc^2}{2} \]
\[ = -9.262 \times 16202 /2 \]
\[ = -12153596.4 \text{ N-mm} \]
\[ M_3 = R_c \left( \frac{R_c}{2w} - a \right) \]  
\[ = 8862418.107 \text{ N-mm} \] (9)

### C. Calculation for Stress Generated

Mmax = 12153596.4 N-mm

**Moment Of Inertia Around The X – X Axis**

\[ I_{xx} = \frac{bh^3 - b_1 h_1^3}{12} \]  
\[ = \frac{(76 \times 2103) - (70 \times 1983)}{12} \]  
\[ = 13372380 \text{ mm}^4 \] (10)

**Section of Modules Around The X – X Axis**

\[ Z_{xx} = \frac{bh^3 - b_1 h_1^3}{6h} \]  
\[ = \frac{(76 \times 2103) - (70 \times 1983)}{6 \times 210} \]  
\[ = 127356 \text{ mm}^3 \] (11)

Stress produced on the beam is as under

\[ M = \frac{M_{\text{max}}}{z} \]  
\[ = 95.43 \text{ N/mm}^2 \]

### D. Check The Deflection of The Beam With All Assembly of Chassis

**Moment of inertia of side bars**

\[ I_{b1} = 13372380 \text{ mm}^4 \]

\[ I_{b2} = 13372380 \text{ mm}^4 \]

**Moment of inertia of cross bar**

\[ I_{b3} = 10023948 \text{ mm}^4 \]

Total mass moment of inertia

\[ = [(13372380 \times 2) + (10023948 \times 6)] \]  
\[ = 86888448 \text{ mm}^4 \]

### E. Deflection of chassis

\[ Y = \frac{wx(b-x)(b-x + b^2 - 2(c^2 + a^2) - 2/b(c^2 x + a^2(b-x))}{24 EI} \]  
\[ = 2.84 \text{ mm} \]

That is within safe limit according deflection span ratio.

### F. FE analysis of existing chassis frame

For carrying out the FE Analysis of chassis as per standard procedure first it requires to create merge part for assembly to achieve the connectivity and loading and constraining is required to be applied also idealization of parts is done on structure this will lead to faster analysis since the connected structure will not be physical but it will be a sketch with mechanical properties of mechanical structure. Procedure is followed in this section.

Cross Section of Main Frame

\[ h = 210 \text{ mm}, b = 76 \text{ mm}, t = 6 \text{ mm} \]
I. Element and Nodes

The meshed truck chassis model has 24840 elements and 48762 nodes. The element is tetrahedral. In order to get a better result, locally finer meshing applied in the region which is suspected to have the highest stress.

III. RESULTS

The location of maximum Von Misses stress and maximum shear stress are at corner of side bar which in Figure. The Von Misses stress magnitude of critical point is 190.38 MPa and the maximum shear stress magnitude is 106.08 MPa.

IV. CONCLUSION

The highest stress occurred is 106.08 MPa by FE analysis.

The calculated maximum shear stress is 95.43 Mpa. The result of FE analysis is bigger 10% than the result of analytical calculation.

The maximum displacement of numerical simulation result is 3.0294 mm.

The result of numerical simulation is bigger 5.92% than the result of analytical calculation which is 2.85 mm.

The difference is caused by simplification of model and uncertainties of numerical calculation.

REFERENCES


