Experimental and Numerical Studies on Materials for Electric Vehicle Chassis

Shankar Durgam*, Vikrant Mahesh Deshmukh, Akshay Prakash Jagtap, Mahendra J. Sable and Nikita M. Gawai
Department of Mechanical Engineering, College of Engineering Pune - 411005, India
E-mail: *sod.mech@coep.ac.in

Abstract. This paper presents experimental and numerical studies on materials for electric vehicle chassis design, testing, and analysis. The chassis for a car should have good rigidity and should be light in weight which are very crucial parameters. To create any car frame is the basic and main structure. It connects rear and front suspension rigidly. Along with this it provides support to the various components of the car. This paper describes the simulation of electric vehicle (EV) Chassis frame with square beam for finding stress, strain and deformation properties in structural body by using finite element analysis (FEA) tools and strain gauge. In this case of chassis structure, we are considering engine weight, alternator weight, motor weight as well as battery pack weight. The total weight on chassis structure is approximately around 4 ton. We are working on finding maximum stress, maximum strain and deformation in structural body by using FEA tools as well as strain gauge.

Keywords: Electric Vehicle Chassis, 3D Modelling, Analysis in Ansys, Strain Gauge.

1. Introduction

Chassis of vehicle is nothing but structure of vehicle without including prime mover. Chassis of vehicle includes frame, power train, brakes, steering etc. subsystems attached to it. Function of chassis is to provide attachment for all components of vehicle such as suspension, steering, brakes and power train. Chassis of the vehicle must be strong enough in order to withstand loads acting on it in all possible cases of loading conditions. Chassis provides support to the control systems such as gearbox, steering, suspension and driver. Static loading conditions mostly involve weight of individual components acting over the frame and dynamic loads like braking, acceleration, cornering. Various loads acting on chassis are vertical bending, horizontal bending, horizontal lozenging, and torsion etc. There are various types of frame constructions used for chassis of vehicle.

(i) Space frame construction
(ii) Ladder frame construction
(iii) Monocoque Chassis

There is some loading condition like normal loading, lateral loading, bending loading torsional loading, combined bending and torsional loading has to be consider during designing new chassis structure. In this case of chassis structure, we are considering engine weight, alternator weight, motor weight as well as battery pack weight. The total weight on chassis structure is approximately around 3.5 ton. Riley and George [1] presented experimental and finite
element-based simulation study on formula (Society of Automotive Engineers) SAE car chassis frame design and explained the torsion test method. Luque et al. [2] proposed algorithm for optimal chassis design of the electric vehicle. They found 20% reduction in battery weight with this method. Mat and Ghani [3] presented FEA base simulations for development of light weight chassis to withstand engine and drivers’ weight and that protect the driver at the time of crash. Chinnaraj et al. [4] conducted experiments on dynamic stress strain response of components of heavy truck chassis. They compared experimental results with FEA based simulations and found this study is useful analysis of stresses in frame of truck rail. Kamble et al. [5] numerically studied design technique for Formula 3 sports car monocoque chassis. They optimized for minimum chassis weight, and high torsional rigidity. Pinheiro et al. [6] presented a skate concept to an electric car for low production cost, low weight, and security. They used sheet of different thickness for chassis parts and found reduction in chassis weight. Several authors [7, 8, 9, 10, 11, 12] studied structural and analysis of stress in ladder chassis frame design with various area of cross sections.

Above literature show that the stress strain analysis for the chassis is very limited. Therefore, this paper includes finding maximum stress, maximum strain and deformation in structural body by using FEA tools as well as strain gauge. The design consideration of chassis is weight. It should be as low as possible and having good strength. This paper also describes the design of Chassis with square beam with the material of Mild Steel Indian Standard (MS IS) 1161/2062 and its static and dynamic analysis. Table 1 shows mechanical properties of MS IS 1661/2062.

| Properties of MS IS 1661/2062 |  |
|-------------------------------|--|
| Youngs Modulus               | $2.1 \times 10^5$ |
| Poisons Ratio                | 0.303 |
| Bulk Modulus                 | $1.7766 \times 10^5$ Mpa |
| Shear Modulus                | 80583 Mpa |
| Ultimate tensile strength    | 440 Mpa |
| Yield Strength               | 370 Mpa |
| Density                      | $7.86 \times 10^{-6}$ |

2. Methodology
This section describes the design of an EV Chassis frame and simulation of design using Ansys software in order to obtain stress, strain and deformation values of the chassis frame at different load conditions.

2.1. Computer Aided Design (CAD) model
Fig. 1 shows the CAD model with magnitude of loads, moments, loading position, moment nature and fixed supports. The beams A, B, C and D are fixed supports. The 12260 N loads acting at point E, F, G and H. The load of 610 N is acting at point I and J, respectively.
2.2. Meshing

Fig. 2 shows mesh used in the present study with the expanded view of the mesh. Preprocessor generated hexa and quad elements were preferred for structural analysis. Areas with holes prone to stress concentration where mesh density was increased as shown in Fig. 2(a). FEA is helpful to do various calculations at limited number of points only. Further, these results are used for the entire structure. As continuous object has infinite number of degrees of freedom, it is impossible to solve the problem in such format. FEA reduces the degree of freedom of a structure from infinite to finite by using meshing. We have worked on meshing with 311932 nodes and 157330 elements. The convergence time for each case of loading condition is about one and half hour. This mesh is selected because it save time and cost of simulation. Fig. 2(b) shows the expanded view of the mesh used in the present study.
3. Analysis in ANSYS

The analysis of chassis structure for studying normal stress, normal strain and total deformation in body at full load condition of 4000 kg is described in this section.

3.1. Normal Stress

Fig. 3 shows Normal stress distribution in x direction along chassis for static structural model. This analysis done at 4000kg load and we get maximum tensile stress value is 121.87 MPa and maximum compressive stress value is 124.44 MPa. Fig. 3 shows the 3D model of assembly of cylinder block and fins.
3.2. Normal strain
Fig. shows Normal strain distribution along chassis for static structural model. This analysis done at 4000kg load and we get the maximum value of tensile strain value 0.00045324 and maximum value compressive strain value 0.00032846

3.3. Total Deformation
Fig. 5 shows total deformation in static structural model of chassis. This analysis done at 4000kg load and we get maximum Total deformation value is 0.18702 mm.
4. Experimental analysis using Strain Gauge
Strain gauge is working on the principle of measurement of change in electric resistance due to deformation of structure with applied load in order to get the value of rate of deformation. Strain gauge is useful for both tensile and compressive load condition. Strain gauge should be strong and light in weight for industry use. These Strain gauges are used in industries for pressure and weight measurement as a transducer. Strain gauges are widely used in study of strength of material such as vibration, residual lifetime, load condition, analysis of failures, etc. Strain gauge gives the values of stresses at those point only where strain gauge is located. Due to this selection of strain gauge location is important for analysis purpose. In order to validate the Ansys simulation result values strain gauge analysis is used. In this we calculate the values of strain induced at 20 different locations on Chassis structure as shown in Fig. 6.

Figure 5. Total deformation for loading condition of 4000Kg

Figure 6. Total deformation for loading condition of 4000 kg
In order to validate the simulated results with the experimental results of strain gauge we have considered the result values at 10 locations. The experimental results using strain gauges in the form of strain which are further converted into stress values by using Young’s modulus value.

5. Results and discussion
The Results of the above analysis chassis frame is discussed in this section.

(i) Based on Ansys Simulation The simulation values of strain are varying from maximum value of tensile strain 0.00045324 to the maximum value of compressive strain 0.00032846 with a load of 4000 kg. The simulation values of stress are varying from maximum value of tensile stress 121.87 MPa to the maximum value of compressive stress 124.44 MPa with a load of 4000 kg. The maximum total deformation value is 0.18702 mm.

(ii) Based on Strain Gauge (Experimental Values) The result of values of normal stress and normal strain in chassis structure is analyzed at 4000kg load condition by using strain gauge at different locations on chassis frame under study. The strain gauge values are varying from maximum value of tensile strain 0.00059828 to the maximum value of compressive strain 0.000404146 with a load of 4000 kg. The strain gauge values of stress are varying from maximum value of tensile stress 121.87 to the maximum value of compressive stress 124.44 with a load of 4000 kg.

Table 2 shows ANSYS based simulation results.

| Item                  | Value       |
|-----------------------|-------------|
| Maximum tensile stress | 121.87 Mpa  |
| Maximum compressive stress | 124.44 Mpa |
| Maximum tensile strain | 0.00045324  |
| Maximum compressive strain | 0.00032846 |
| Total deformation value | 0.18702    |

5.1. Validation of Stress Strain Values at 5000kg Load Condition
The validation of Ansys simulation-based strain values with the strain values obtained by using strain gauge at different locations at 4000 kg load condition are shown in Fig. 7. In this, a curve of Ansys simulation-based strain values varying from maximum to minimum values at different positions on chassis frame is shown. These are further comparing with experimental strain values obtained by using strain gauge varying from maximum strain value to minimum strain values. The comparison of Ansys based stress values and strain gauge based experimental stress values are shown in Fig. 8. The experimental and simulated stress strain values are in good agreements. Table 3 show the experimental and simulated strain values.
Table 3. The experimental and simulated strain values

| Location | Simulated strain | Experimental strain |
|----------|------------------|---------------------|
| 1        | 0.00045324       | 0.00059828          |
| 2        | 0.00036638       | 0.000390356         |
| 3        | 0.00027953       | 0.000278232         |
| 4        | 0.00019267       | 0.00019078          |
| 5        | 0.00010582       | 0.000135079         |
| 6        | 0.000018961      | 9.5494×10^−6        |
| 7        | 0.000067895      | 8.8295×10^−6        |
| 8        | -0.00015475      | -0.00015689         |
| 9        | -0.00024161      | -0.000235665        |
| 10       | -0.00032846      | -0.000404146        |

Table 4 shows the experimental and simulated stress values. It is found that the experimental and simulated values are in good match.

Table 4. The experimental and simulated stress values

| Location | Simulated stress | Experimental stress |
|----------|------------------|---------------------|
| 1        | 212.87           | 125.64              |
| 2        | 94.503           | 84.87               |
| 3        | 67.136           | 58.43               |
| 4        | 39.769           | 32.91               |
| 5        | 12.401           | 18.54               |
| 6        | -14.966          | -18.36              |
| 7        | -42.333          | -40.06              |
| 8        | -69.701          | -54.88              |
| 9        | -97.068          | -87.18              |
| 10       | -124.44          | -118                |

Figure 7. Strain validation at 4000 kg load
6. Conclusions

The bending moment that the beam can sustain is good for square beam. The frame having minimum deformation of beam for 4000 kg loading condition. Hence frame is safe to use up to that loading condition. To improve strength of frame, we can use cross members as well as diagonal support members. To optimize design, we can use skateboard type chassis structure and advance automotive material like high speed steel (HSS), dual phase (DP) Steel, ultra high speed steel (UHSS), transformation induced plasticity (TRIP) steel, Twin Steel where we can achieve more strength, crashworthiness.

References

[1] W. B. Riley, A. R. George, Design, analysis and testing of a formula sae car chassis, Tech. rep., SAE Technical Paper (2002).

[2] P. Luque, D. A. Mántaras, Á. Maradona, J. Roces, L. Sánchez, L. Castejón, H. Malón, Multi-objective evolutionary design of an electric vehicle chassis, Sensors 20 (13) (2020) 3633.

[3] M. H. Mat, A. R. A. Ghani, Design and analysis of ‘eco’car chassis, Procedia Engineering 41 (2012) 1756–1760.

[4] K. Chinnaraj, M. Sathya Prasad, C. Lakshmana Rao, Experimental analysis and quasi-static numerical idealization of dynamic stresses on a heavy truck chassis frame assembly, in: Applied Mechanics and Materials, Vol. 13, Trans Tech Publ, 2008, pp. 271–280.

[5] M. Kamble, T. Shakfeh, R. Moheimani, H. Dalir, Optimization of a composite monocoque chassis for structural performance: a comprehensive approach, Journal of Failure Analysis and Prevention 19 (5) (2019) 1252–1263.

[6] L. Pinheiro, F. Passarinho, A. Teixeira, H. Silva, B. Ribeiro, New skate chassis concept for electric vehicles, Tech. rep., SAE Technical Paper (2012).

[7] N. A. Mohammed, N. Nandu, A. Krishnan, A. R. Nair, P. Sreedharan, Design, analysis, fabrication and testing of a formula car chassis, Materials Today: Proceedings 5 (11) (2018) 24944–24953.

[8] H. Radhiwan, Z. Shayfull, S. Nasir, A. Irfan, et al., Design and analysis of solar car chassis, in: IOP Conference Series: Materials Science and Engineering, Vol. 864, IOP Publishing, 2020, p. 012144.

[9] V. V. Patel, R. Patel, Structural analysis of a ladder chassis frame, World Journal of science and Technology 2 (4) (2012) 05–08.

[10] K. Y. Patil, E. R. Deore, Stress analysis of ladder chassis with various cross sections, IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) 12 (4) (2015) 111–116.

[11] W. G. Ferreira, F. Martins, S. Kameoka, A. S. Salloun, J. T. Kaeya, Structural optimization of automotive components applied to durability problems, Tech. rep., SAE Technical Paper (2003).

[12] L. L. Thompson, J. K. Lampert, E. H. Law, Design of a twist fixture to measure the torsional stiffness of a winston cup chassis, Tech. rep., SAE Technical Paper (1998).