Design study of electric vehicle roll-cage obtained using modal analysis

Shubham Arun Rao Katait  and Pankaj Dorlikar

1 Department of Mechanical Engineering, Army Institute of Technology, Pune, India
shubhamkatait@gmail.com, pvdorlikar@aitpune.edu.in

Abstract. Understanding the dynamic behaviour of structures has become increasingly essential in the design process of any mechanical system. Modal factors like natural frequencies and mode shapes characterize the dynamic response of the structure and can be used for determination of the possible bending and torsion phenomenon. The Computer-Aided Drafting (CAD) model plays an important role in the design and development phases of an electric vehicle. For doing Finite Element Analysis (FEA) using software CAD model has to be imported so that after conversion Finite Element (FE) model can be solved. In the present case the CAD model is imported in HyperMesh. The FE model is generated and solved by using OptiStruct. Ignoring the first six natural frequencies and mode shapes due to its association with six degrees of freedom, subsequent six natural frequencies and mode shapes are proposed for an electric vehicle roll-cage. We can learn even more from CAD geometry and mode shapes in every direction about its stiffness and weight. Once we know our design baseline, we can use a software simulation to use a design study and improve its performance. We have considered the pipe diameter, thickness, tube width and height as main design parameters which needs to be optimized. In this paper, we optimized all these design parameters to make driver comfortable during driving by increasing natural frequency of the roll-cage structure by 3.34 times the human body threshold.

1. Introduction
Electric vehicles are grabbing everybody's attention as an important solution for enhancing air quality to achieve zero pollution [1]. Today, many of the researchers are actively working on to improve the fields of safety, range, acceleration, economy as well as energy, power density and recharging time of the batteries, which are the main drawbacks of the electric vehicles [2]. Due to availability of FEM analysis capability of the analysis software, we are using it as a model simulator for the manufacturing end products besides a design process tool.

Multi-tubular space frame, frequently suggested to as roll-cage goes about as a structural symbolize for different sorts of automotive vehicles. The roll-cage needs to adjust to the principles indicated by SAE guidelines. The roll-cage is exposed to the different dynamic and static load experienced during front, side, roll-over, torsional impact. The frame of roll-cage ought to be adequately hardened to withstand against all possible loads so that we get with adept strength to weight ratio [3]. To meet these rules it is critical to consider different boundaries engaged with the plan of a roll-cage, directly from the material used for the fabrication and exposing it to the impacts and forces it may experience. Factor of
safety 2 was established, which gave a moderate design. In a modal analysis, the weakest parts of our design expose themselves as low frequency resonant modes [4]. The main objective of the present work is to find the optimum pipe diameter, thickness, tube width and height of roll-cage pipe at the very first step after the CAD geometry creation. This is followed by the further static as well as dynamic analysis of the roll-cage.

2. Dimensions and Material properties

The design of the roll-cage in any condition should consistently guarantee sufficient clearances from roll-cage members for the safety of driver along with his/her free movement. A minimum roll-cage provision that can accommodate two individuals with an arrangement to place lead-acid battery and small cargo (230L) in front to place bags is considered. It can also handle two in-wheel electric motors on the rear suspension frame. Outer dimensions of the roll-cage are presented in Table 1.

| Parameter     | Length   | Width   | Height   | Gross vehicle weight |
|---------------|----------|---------|----------|----------------------|
| Dimension     | 2696mm   | 1558mm  | 1540mm   | 570kg                |

From the availability of huge variety of alloys steel, AISI 4130 chromoly steel was picked to be the reasonable material due to its less weight per meter length when compared with AISI 1018. Steel pipes from AISI-4130 were used as a material for the construction of roll-cages because of high tensile and yield strength. Besides these, the risks of welding capability are also minimal. The Material properties are displayed in Table 2.

| Property               | Values       |
|------------------------|--------------|
| Initial Density        | 7890 kg/m³   |
| Young's Modulus        | 210 GPa      |
| Poisson's Ratio        | 0.3          |
| Yield Stress           | 180MPa       |
| Hardening Parameter    | 230MPa       |
| Hardening Exponent     | 0.2          |
| Maximum Stress         | 300MPa       |
| Ultimate Tensile       | 670MPa       |
| Strength               |              |
| Yield Tensile Strength | 440MPa       |

3. Research methodology

First we had developed the CAD geometry in SolidWorks. Then the CAD geometry is imported to HyperMesh. The FE model is generated and solutions are obtained using OptiStruct solver. Then, we found out the first feasible mode shapes and then use HyperStudy, SolidWork Simulation is also used to optimize the pipe thickness. As the thickness of our pipe adds more weight than stiffness, we need to find out the critical point in our design. We have to optimize the thickness and maximize the first mode resonant frequency. The detail methodology is shown in Figure 1.
4. CAD Modelling
In CAD modelling, we made one roll-cage and added different parts like the seat, the battery, the mannequin. Then the reach of mannequin is checked to make it comfortable for the human users while driving. We made the final roll-cage in SolidWork with the given specification. Detailed packaging of the roll-cage parts is illustrated in the Figure 2.
5. Modal analysis optimization
Modal analysis is a method used to study the vibrational excitement of a system. The most basic form of analysis is modal analysis and is typically the starting point of further, more detailed analysis of dynamics. Modal analysis is used to classify the vibrational features of a system, including natural frequencies and mode shapes. Modal analysis helps design to prevent resonant vibrations or to vibrate at a certain frequency and allows engineers to see how the design can respond to various complex load types.

Mesh size is a key consideration when doing Finite Element Analysis. The mesh size is defined as the size of the smallest uniform element to be analyzed and thus has a crucial role to play in any analysis procedure. Mixed shell meshing with tria and quad element with 7mm fine mesh element size is used to perform different types of analysis for the consideration of accurate and very low level of assumption. Firstly, we take the circular pipe of a diameter of 25mm and a thickness of 3mm, rectangular pipe of 101.6mm x 50.8mm and a thickness of 3mm, rectangular plate of a thickness of 2.5 mm, is used in roll-cage. The final meshed roll-cage model is having 183652 total elements with 123.5 Kg mass of roll-cage. The final meshed roll-cage is shown in Figure 3.

![Figure 3. FE model](image)

5.1 Free-free modal Analysis FE model
Modal analysis of the roll-cage was performed using solver OptiStruct from finite element software HyperWork. Due to the advantages of reasonably accurate visualization and simulation of the mode shapes, finite element analysis software is used. Also the deformations that occur in the roll-cage could be precisely located. The different mode shapes of the FE model are displayed in Figure 4. The six consecutive mode shape frequencies are summarized in Table 3.

![Figure 4. Mode shapes](image)

(a) F7: 24.971 Hz  (b) F8: 28.510 Hz  (c) F9: 28.671 Hz
5.2 HyperStudy optimization

HyperStudy is a multi-disciplinary software design study that allows engineers to study the performance of their products and optimize their stability. Optimization begins after the execution of a clean basic analysis (e.g. sliding energies and negligible hourglassing). New design variables or further optimization can be restarted conveniently from a new position onto the response surface.

First, we must choose optimizing parameter in HyperStudy. Then we have to include the range in the module as input values of software that is the optimizing parameter's upper and lower bound, which is inferred from the modal analysis. Then we make one of the OptiStruct linear static analysis input files with all set boundary conditions. This file will be imported at the time of HyperStudy optimization and solve in HyperStudy software. After running solver, we get the optimized result of every parameter shown in Table 4.

Table 4. HyperStudy optimization result table

| Optimizing parameter       | Lower Bound | Upper Bound | Optimal   |
|----------------------------|-------------|-------------|-----------|
| Thickness rectangle(mm)    | 4           | 6           | 4.928     |
| Width (mm)                 | 25.4        | 76.2        | 48.7934   |
| Height (mm)                | 50.8        | 152.4       | 102.1334  |
| Thickness pipe (mm)        | 1.5         | 3           | 1.758     |
| pipe diameter (mm)         | 12.7        | 50.8        | 24.9428   |

5.3 SolidWorks Simulation for optimization

A DoE-base optimization approach is used in SolidWorks Simulation. In order to run a problem, the engineer sets the maximum and minimum values for his dimensional design variables, and then chooses the "Standard" or "High Quality" optimization. The Standard approach assumes that the objective response curve between the limit values is linear and only calculates the response to those values. High-quality optimization takes into account the possibility of a second-order response between limits and evaluates both the mid-value and the extremes.
We may also find somewhat the same nature of conclusion through other design analysis (however with less colorful pictures). SolidWork Simulation is providing us with the optimized solution (as not every result must be released), through setting our analysis to increase or decrease the pipe or rectangle tube's thickness by iterating, thus optimizing the first mode of resonance. Here we can even get more accurate results by reducing the increments to just 1 inch. Optimized parameter design study inputs window of SolidWork Simulation shown in Figure 5.

![Figure 5. Optimized parameter design study input window](image)

We get the final optimized parameter table from SolidWork Simulation as shown in Table 5.

| Optimizing parameter | Current | Optimal | Scenario-1 | Scenario-2 | Scenario-3 |
|----------------------|---------|---------|------------|------------|------------|
| Rectangle Thickness (mm) | 4       | 5       | 3          | 4.5        | 6          |
| Width (mm)            | 38.1    | 50.8    | 25.4       | 63.5       | 76.2       |
| Height (mm)           | 76.2    | 101.6   | 50.8       | 127        | 152.4      |
| Pipe thickness (mm)   | 2.5     | 2       | 1.5        | 1.9        | 3          |
| Pipe diameter (mm)    | 19.05   | 25.4    | 12.7       | 31.75      | 38.1       |
| Frequency (Hz)        | 24.32   | 26.11   | 21.55      | 21.65      | 25.65      |

5.4 Final optimized FE Roll-cage
The result of HyperStudy optimization (Table 4) and SolidWork simulation (Table 5) finalizes the best possible design parameter values as shown Table 6, which are the final optimized FE roll-cage parameters for further design study.

| Optimizing parameter | Optimal |
|----------------------|---------|
| Rectangle Thickness (mm) | 5       |
| Width (mm)            | 50.8    |
| Height (mm)           | 101.6   |
| Pipe thickness (mm)   | 2       |
| pipe diameter (mm)    | 25.4    |

5.5 Free-free modal Analysis optimize FE roll-cage
After final optimization of the parameter check with free-free modal analysis, We get to maximize the first shape of the roll-cage mode as displayed in Figure 6 and are summarized in Table 7.
Table 7. Free-free modal Analysis of optimize FE roll-cage

| Frequency | Free-free(Hz) |
|-----------|--------------|
| F7        | 25.024       |
| F8        | 29.186       |
| F9        | 30.958       |
| F10       | 33.562       |
| F11       | 38.481       |
| F12       | 47.38        |

The natural frequency of a human body in standing posture is about 7.5 Hz and in sitting posture is 4-6 Hz [10]. After referring table 7, we can easily conclude that operator can drive this roll-cage based vehicle comfortably, since the FEA optimised frequency is approximately 3.34 times higher than maximum human body bearable frequency.

6. Design Study of optimize FE roll-cage
The roll-cage strength is such that the working conditions are moderately permanent deflected. To ensure the study of the finite element analysis and to calculate stress levels. The strength of the [roll-cage tubes] frame is set at a value that exceeds the stress value reached in the study. G-force is used for static analyzes, according to the Newton second movement law.

\[ F = mass \times acceleration = 570 \times 9.81 \approx 5592 \]

Where, \( F = \) Force exerted on roll-cage in N; \( m = \) mass of vehicle in kg, \( a = \) acceleration in m/s\(^2\)

To assess its feasibility and structural integrity according to various static and dynamic load tests.

Static Structural Analysis
- Front Impact
- Rear Impact

Explicit Dynamic Analysis
- Front Impact
- Rear Impact
6.1 Static Structural Analysis

Static analysis is performed to check the integrity of the roll-cage and the strength to withstand forces during the actual simulation. Boundary conditions for static structural analysis as shown in Figure 7.

**Figure 7.** Boundary condition for static structural analysis of optimize FE roll-cage

- Front Impact

Front impact analysis is carried out to monitor the integrity of the roll-cage and resist the impact force on the front. 3.5G Nodal strength on the front part nodes shall be equally distributed. The nodes on the rear side of the roll-cage in all directions are fixed to simulate this condition. Front impact result are shown in figure 8.

**Figure 8.** Static structural analysis of roll-cage under frontal impact

- Rear Impact

Rear impact analysis is carried out to monitor the integrity of the roll-cage and resist the impact force on the rear. 3.5G Nodal strength on the front part nodes shall be equally distributed. The nodes on the front side of the roll-cage in all directions are fixed to simulate this condition. Rear impact result are shown in figure 9.

**Figure 9.** Static structural analysis of roll-cage under rear impact
6.2 Explicit Dynamic Analysis

Dynamic analysis is performed to check the performance of the roll-cage under dynamic loading conditions. This simulation is conducted in the software package of HyperWork. Boundary conditions for explicit dynamic analysis as illustrated in Figure 10.

- **Front Impact**
  The Front Impact Analysis was conducted on the HyperWork RADIOSS 2017. For the analyses, the limit conditions used include an infinite wall (RWALL) which is fixed in front of the roll-cage. The initial speed of 56 kmph according to SAEINDIA is the rules for roll-cage. For better graph tracing, ENG-RUN time (Tstop) will be used as 0.1. As the boundary condition is exposed in figure 10.

- **Rear Impact**
  The Rear Impact Analysis was conducted on the HyperWork RADIOSS 2017. Same rigid wall, initial speed and ENG-RUN time like front impact is used in rear impact. As the boundary condition is exposed in figure 10.
7. Result
After studying the outputs from figure 8, 9, 11 and 12, the results of FE optimized roll-cage are summarized as shown in Table 8.

Table 8. Design Study result of optimize FE roll-cage

| Analysis     | Loading | Maximum Deformation (mm) | Max. Deformation (mm) (Matlab) | Max. Stress (MPa) | Max. Stress (MPa) (Matlab) | FOS |
|--------------|---------|--------------------------|---------------------------------|-------------------|---------------------------|-----|
| Front Impact | 3.5G    | 1.046                    | 1.24                            | 195               | 187                       | 3.43|
| Front Impact (Dynamic) | - | -                        | -                               | 299.97            | -                         | 2.23|
| Rear Impact  | 3.5G    | 17.94                    | 18.10                           | 253.63            | 254.10                    | 2.64|
| Rear Impact (Dynamic) | - | -                        | -                               | 353               | -                         | 1.86|

We have obtained the optimum design parameter (Thickness, width, Height, diameter) of the pipe as well as tubular section by modal analysis. We have also written a Matlab code for validation. As seen from the Table 8, the roll-cage is found secure when loaded statically and dynamically with a factor of safety for the roll-cage reaches around 2.

Free-free modal Analysis of optimize FE roll-cage after Ignoring the first six natural frequencies and mode shapes due to its association with six degrees of freedom, subsequent six natural frequencies and mode shapes are proposed for an electric vehicle roll-cage range between 25.024hz to 47.38hz.

8. Conclusion
This paper provide the optimum design parameter (Thickness, width, Height, diameter) of a roll-cage considering the modal analysis. We come up with different static structural and explicit dynamic analysis from that we can figure out the Table 7 that the roll-cage is secure when loaded statically and dynamically. The factor of safety for the roll-cage reaches 2, which is safe from design point of view. The naturally produced first bending mode will meet the dynamic frame performance requirements under different conditions. The frequency of the first mode is higher than the frequency of road excitation in the most common operating conditions and is also higher than the frequency of the electric motor, thus trying to avoid these common resonance problems. For improving the comfort level of driver it is necessary to either reduce or increase the natural frequency of roll-cage. Since the FEA optimised frequency is approximately 3.34 times higher than maximum human body bearable frequency.

References
[1] Bonisoli E, Delprete C, Esposito M and Mottershead J 2011 Structural dynamics with coincident eigenvalues: modelling and testing. Modal Analysis Topics, Volume 3, Proceedings of the 29th IMAC, 2011.
[2] Klerk D, Rixen D and Voormeeren S 2008 General Framework for Dynamic Substructuring : History, Review and Classification of Techniques AIAA Journal, Vol. 46, No. 5, (2008), pp. 1169–1181.
[3] Heylen W and Sas P 2006 Modal Analysis Theory and Testing Katholieke Universiteit Leuven, Departement Werktuigkunde.
[4] Richardson M H 1978 Measurement and Analysis of the Dynamics of Mechanical Structures, Hewlett-Packard Conference for Automotive and Related Industries Detroit MI pp 1–46
[5] Lee D, Choi H and Han S 2006 Design of Automotive Body Structure Using Multicriteria Optimization Journal of Structural and Multidisciplinary Optimization, Vol. 32, 2006, pp. 161-167.
[6] Lee N and Nikravesh E 1994 Steady State Analysis of Multibody Systems with Reference to Vehicle Dynamics *Journal of Nonlinear Dynamics, Vol. 5, 1994, pp. 181-192.*

[7] Riley W and George A 2002 Design, Analysis and Testing of a Formula SAE Car Chassis *SAE TECHNICAL PAPER SERIES 2002-01-3300.*

[8] Balmes E 2018 Shapes & DOF: on the use of modal concepts in the context of parametric nonlinear studies *ISMA, Sep 2018, Leuven, Belgium, hal-01912722*

[9] Balaguru, Natarajan E, Ramesh and Muthuvijayan 2019 Structural and modal analysis of scooter frame for design improvement *International Conference on Advances in Materials, Manufacturing and Applied Sciences, 2214-7853*

[10] Anas M, Nandu N, Anjai K, Nairc A and Sreedharand P 2017 Design, Analysis, Fabrication and Testing of a Formula Car Chassis *International Conference on Advances in Materials and Manufacturing Applications [IConAMMA 2017], 24944–24953.*

[11] Ren W, Peng B, Shen J, Li Y, and Yu Y 2018 Study on Vibration Characteristics and Human Riding Comfort of a Special Equipment Cab *Journal of Sensors Volume 2018, Article ID 7140610, 8 pages*

[12] Lombard M 2018 *Mastering SolidWorks* (Inc., Indianapolis, Indiana, Canada)

[13] Gokhale N, Deshpande S, Bedekar S and Thite A 2008 *Practical Finite Element Analysis* (India)