International Journal of Research Studies in Science, Engineering and Technology
Volume 2, Issue 7, July 2014, PP 69-73
ISSN 2349-4751 (Print) & ISSN 2349-476X (Online)

Design and Analysis of Tiltable Truck Cabin Floor

D. Murali Krishna[1]

[1]Department of Mechanical Engineering,
MLR Institute of Engineering,
Hyderabad, India, Student.

Dr. S. Madhu[2]

[2] Department of Mechanical Engineering,
MLR Institute of Engineering, Hyderabad,
India, Head of the department.

Abstract: *The cabin floor of a truck plays an important role in transferring loads to drivers and co-driver. It should withstand to loads obtaining from the roads, assemblies and the loads carrying. The floor should be designed to ensure fatigue life and should not fail in service to the instantaneous over-load. It should not pass the vibrations from the bulkheads and engine assembly. To meet the requirements, the floor is designed and analysis is carried out to investigate the behaviour of the floor to the applied loads. By using CATIA and ANSYS softwares, the floor is designed and analysis is performed respectively.*

Keywords: *CATIA, HYPERMESH, ANSYS, MODAL, HARMONIC, TRANSIENT ANALYSIS*

1. Introduction

An Automobile is a self propelled vehicle which is used for transportation of goods and passengers. The motor vehicles, both passenger car and trucks are generally considered to be made up of two major assemblies: Body and Chassis. Chassis is a frame or main structure of a vehicle. The chassis contains all the major units necessary to propel the vehicle. Body is the super-structure of the vehicle. Body is bolted to the chassis. The chassis and the body make the complete vehicle.

The truck consists of different assemblies performing their functions smoothly. Although there are many important parts, the cabin is a place where the driver and co-driver are seated. Their weight will be mainly on the floor where it should withstand many loads coming from different ways in different directions. This makes the driver seated without any vibrations and distractions.

A flat sheet of thin material such as the floor panel is very flexible for out-of-plane loads. The aim of the floor is to carry the local applied loads from their point of application to the major structural components of the vehicle, such as the side frames. Floors are subject to loads normal to their plane. Under such circumstances they do not act as simple structural surfaces. The floor is stiffened against out-of-plane loads by added beam members arranged into a planar framework.

The advantages of tilting cabin over rigid cabin is easy for servicing, less weight, easy for design modifications and provide less vibrations.

2. Load Distribution in Floor Members

Point loads, such as passenger seat reactions, are usually fed straight into local, probably minor, beam members. Distributed payloads frequently rest on the floor panel itself, which will need local reinforcement like swages. The local load is carried along the swages to adjacent beam members. The local members will transfer it to larger members (such as the transmission/services tunnel) and so forth until it reaches the side frame. The progression of the loads through the structure follows the pattern shown below.

Load gathering in floor
Swages
Minor Beams
Major Beams
Side Frames

3. Forces Acting on Truck Cab

The vehicle designer needs to know the most damaging loads to which the structure is likely subjected. If the structure can resist the worst possible loading which can be encountered then it is likely to have sufficient fatigue strength.

1. Vertical symmetrical ('bending case') cause bending about Y – Y axis.
2. Vertical asymmetric ('torsion case') cause torsion about the X – X axis and bending about Y – Y axis.
3. Fore and aft loads (braking, acceleration, obstacles, towing).
4. Lateral (cornering, nudging kerb).
5. Local load case, e.g. door slams etc.
6. Crash cases.

In real life many of the loads will occur in combination with each other. For example the weight is always present, so that the longitudinal and lateral cases are always accompanied. On the road, almost any combination of the pure cases can be encountered. For example, a cornering vehicle might encounter a bump (or pot hole) with one wheel. This could involve the extreme vertical asymmetric load case plus the extreme factored longitudinal bump case plus the cornering case and possibility also the braking case.

4. Problem Statement

The objective is to analyse the dynamics behavior of floor of a truck using Finite Element modeling under different load conditions. The floor under different loads such as driver seat loads and co-driver seat loads along with the driver and co-driver acts on the floor directly, where as the road loads are taken by the vehicle front suspension and the cab rear and front suspension system. Analysis is done to determine the dynamic behavior of the vehicle in Static Harmonic and Transient excitations. Modal analysis is also carried out to read the natural frequency of the floor along with Static analysis to determine the deflections and stresses.

5. Methodology

The model is developed by using the CATIA and meshing of the model is performed in HYPERMESH. The analysis is performed in ANSYS softwares and the results were also noted.

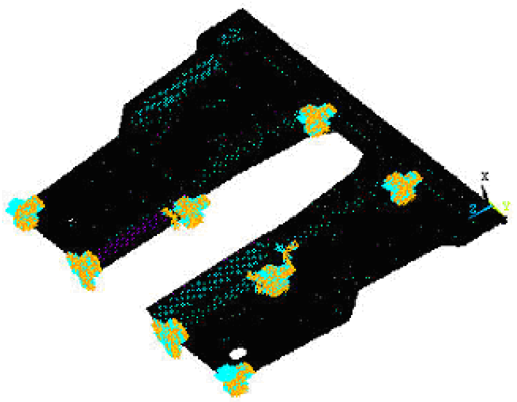

Figure1. *Meshed model of floor along with boundary conditions*

For solving any problem analytically, we should define (1) solution domain (2) physical model (3) boundary conditions and (4) physical properties. The model along with boundary conditions applied is shown in the above figure 1. The material assigned are young's modulus = 1.95 E 5 N/mm^2, poisons ratio = 0.29, density = 7.8E –9 N/mm^3. Shell 63 element type is preferred where it has both bending and membrane capabilities. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. Stress stiffening and large deflection capabilities are included.

5.1. Loads and Constraints

For any commercial software there are three steps for the solving of problem. Pre-processing, Processing, Post-processing. Pre-processing deals with CAD data, meshing the model, Boundary conditions. Processing deals with matrix formulations, inversion, multiplication and solution for unknown variables internally in software. Post-processing is viewing results, verifications of variables and drawing conclusions.

In this model, the front end of the floor is hinged through the hinges to the front of the chassis whereas at the rear side the floor is mounted on the suspension system and is locked through the locks which are fixed under the floor reinforcement. The floor is supported on the mounts in middle which are fixed to the chassis (please refer to figure 1). All the degrees of freedom are arrested at all the mounting locations and the same boundary conditions are used for all the analysis.

6. Results and Discussions

6.1. Modal Analysis

Modal analysis is used to determine the vibration characteristics of a structure or a machine component while it is being designed. The Table 1 shows the frequencies obtained by using modal analysis:

Table1. *Frequencies obtained from modal analysis*

Set	Frequency, Hz
1	6.288
2	6.406
3	19.105
4	19.326
5	23.312
6	23.773
7	31.298
8	33.273
9	39.035
10	39.154

From the frequencies obtained from the Modal analysis the fundamental frequency is observed as 6.288Hz. The frequencies obtained from modal analysis lies between the working frequencies range, hence resonance conditions exists. Resonance can be avoided by reinforcing the cross members by another high stiffness members.

6.2. Static Analysis

A force of 300N is applied on the floor at the driver and co-driver seats. The model represents the variation or resultant deformation with respect to geometry. In the Figure 2 shown below, the maximum resultant deformation of 0.988784mm is observed at the edges of the floor.

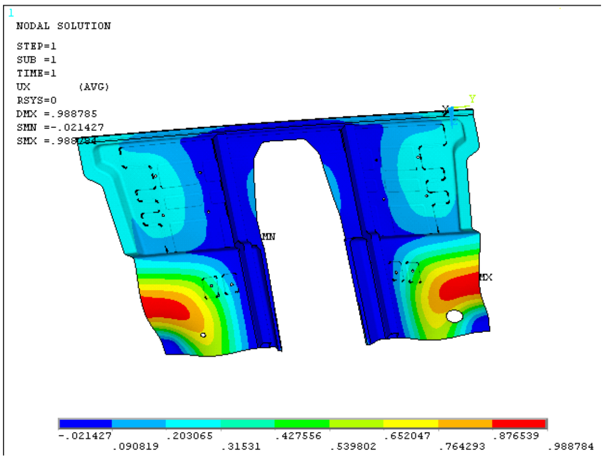

Figure2. *Contour plot showing Max. Deflection*

The variation of von-mises stresses in the truck cabin floor in the Figure 3 is observed to be 55.652 Mpa which is less than allowable stress of 340 Mpa. The entire section has a relatively low stress value except for the area under impact. This is because of the loads and constraints acting on the floor.

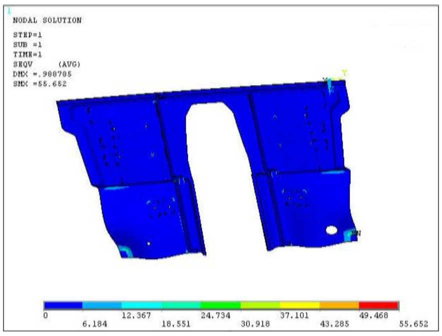

Figure3. *Vonmisses stress plot*

6.3. Harmonic Analysis

Harmonic response analysis gives the ability to predict the sustained dynamic behavior of structures, thus it enabling to verify whether or not designs will successfully overcome resonance, fatigue and other harmful effects of forced vibrations.

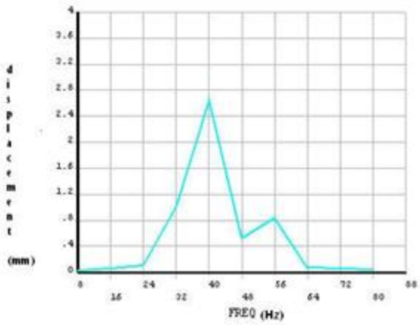

Figure4. *Displacement Vs Frequency*

From this analysis the displacements of various nodes and stress for different elements over the entire frequency range 0 – 80 Hz. From the above Figure 4 it states that the maximum frequency occurs at 40Hz. with a displacement of 2.5mm.

6.4. Transient Analysis

Transient dynamic analysis (sometimes called time-history analysis) is a technique used to determine the dynamic response of a structure under the action of any general time-dependent loads. The force is variable in nature and the time step taken as 0.001sec.

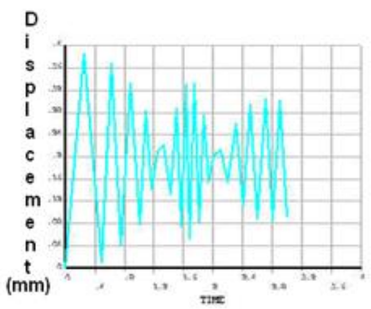

Figure5. *Displacement Vs Time*

From the Transient analysis shown in Figure 5 and Figure 6, the maximum Displacement is observed to be 0.3844mm .The velocity is observed to be 2.2839mm/sec. The Acceleration is varying in the range of 1186.93e2 mm2/sec to –1169.98e2 mm2/sec.

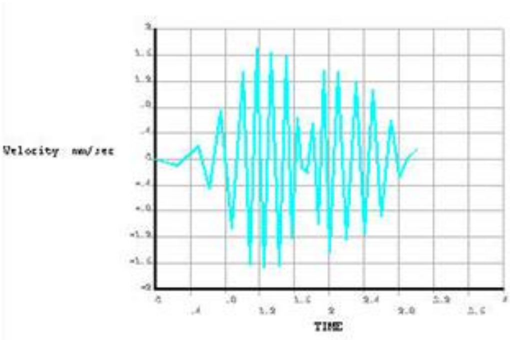

Figure6. *Velocity Vs Time*

7. Conclusion

The parameters considered for the floor are allowed to be used for the successful operation of the floor. Hence, the floor will operate safely and with ensured life. Hence the design is safe based on strength. From the Modal Analysis the fundamental frequency is obtained as 6.288Hz and Resonance condition exists. Resonance can be decreased by adding stiffness members like cross members etc. From Transient analysis, the maximum displacement is 0.3884mm and the velocity is 2.2839mm/sec. Transient displacements are quite high in the front portion.

References

[1] Ali.R, Hedges.J.L, Mills.B, “Dynamic Analysis of an automobile chassis frame”, Institution of Mechanical Engineers, Automobile Division, Vol. 185.

[2] V. Balamurugan, “Dynamic analysis of a military-tracked vehicle”, Defence Science Journal, Vol. 50, No. 2.

[3] Curtis.F.vail,” Illustrations of automotive finite element models-dynamics”, SAE Technical Paper 740005.

[4] Karuppaiah.N, Deshpande.P.s, C.Sujatha and Ramamurthi.V, “Vibration analysis in a light passenger vehicle by rigid body/finite element modeling”, Vol 2(2), pp 106-120.

[5] Kiyoshi Miki, Mamoru Nagai, and Eiji Adachi, “Elastic vibration analysis of passenger car bodies (bending and torsion)”, SAE Technical Paper 690272.

[6] Vehicle Body Construction & Design Engineering By Dr. Pawlowski.

[7] Hand Book of Vehicle Design & Analysis by John Fenton.