Optimization of the Concept Cab and Cab Components for Heavy Commercial Vehicle

Rahul A. Velankar
D.Y.Patil College of Engineering, Akurdi, pune-44

Vaishali Sawant,
Mahindra Engineering, Chinchwad , pune-19

Prof: Dr (Mrs) R. S. Bindu,
D.Y.Patil College of Engineering, Akurdi, pune-44

Abstract

Many times the importance of cost reduction programs within a company cannot be overstate. Knowing how to implement effective cost reduction strategies can be the determining factor in the survival of a business.
The aim of this paper study is the optimization of the floor reinforcements in account of reduction of cost. The floor panel of cab having the five cross members to reinforce the floor. The challenge was either reduce or respace the floor cross members.
During this optimization technique it is essential to calculate the stress level, determine the critical locations and estimate the life of the components under the vehicle durability requirement and floor stiffness performance.

Keywords: Optimization, Floor Stiffness Condition, Durability Loading Condition.

1. Introduction

The current production design of the floor panel having the five cross members with thickness 1.4mm to reinforce the floor. The performance of the components tested under vehicle durability requirement i.e. under peak inertial loading in all the three directions i.e lateral, longitudinal and vertical and floor stiffness condition i.e. (300lbs load on 5inch area) considering a person standing or sitting on the floor. The schematic of the five cross members (green) are shown in Fig.1. This is bottom view of the floor green colour cross members are under evaluation.

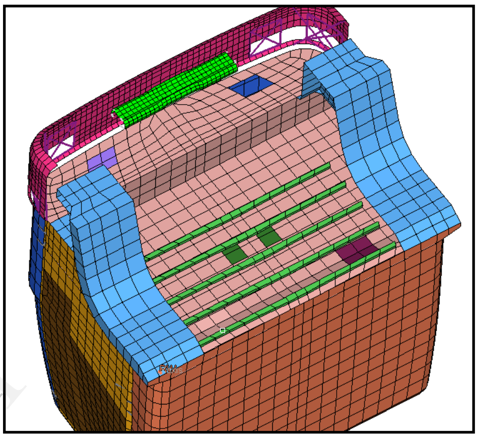

Fig. 1: Schematic of Cross Members in green colour

The five cross members shown in green colour above are having the thickness is 1.4mm and these cross members are spot welded with the floor panel and long cowl at the respective locations. The cross member having the material as Steel Grade 1 with 165 MPa yield strength and 145 MPa fatigue strength. For the non linear analysis the true stress strain curve for Steel Grade 1 was used. Fig. 2 shows the true stress strain curve for Steel Grade1.

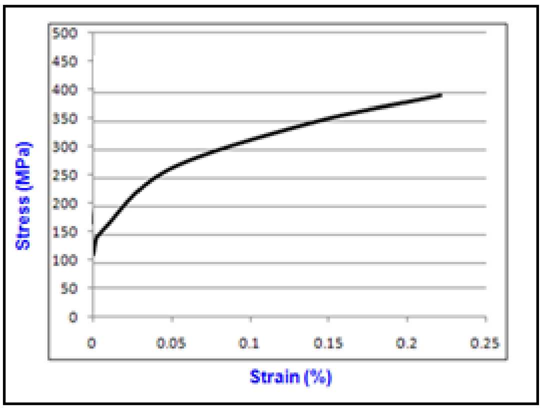

Fig. 2: True stress strain curve for Steel Grade 1

After a brain storming and using engineering judgment here are some design suggestions evaluated during this study work.

Iteration 1: Floor Panel with four cross member. (Second cross member removed and spacing reduced between first and third) refer Fig. 3.
Since there is no load and most of the load is concentrated in the front cross member second cross member was removed.

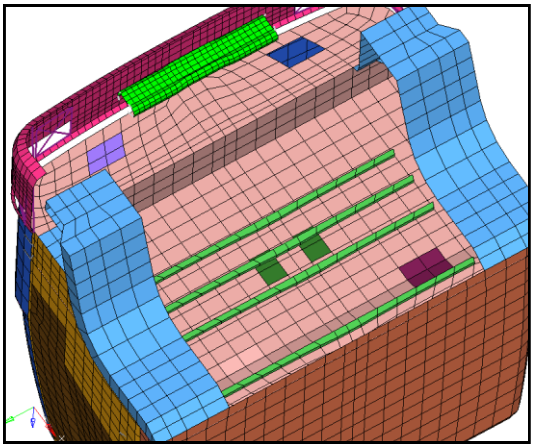

Fig. 3: Floor Panel with four Cross Member

Iterations 2: Floor Panel with three Cross member. (second and forth cross member removed) Refer Fig. 4. Assuming that the load will be distributed over whole floor space another cross member was removed. Iteration 1 and Iteration 2 were two extreme load cases which are taken for evaluation to understand the behaviour of the system.

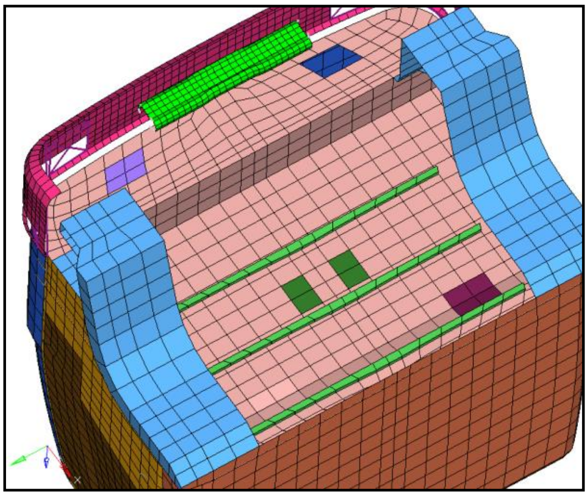

Fig. 4: Floor Panel with three Cross Member

Iteration 3: All the five cross members with 1mm thickness. In addition to Iteration 1 & Iteration 2 we thought of evaluating Iteration 3 with reduced thickness to 1mm overall.

2. Methodology

The modelling of the components was carried out in the HM12. The linear durability analysis was carried out in Nastran and non linear analysis of the floor stiffness was carried out in Abaqus 6.8. The post processing of the results was carried out using Hyperview. The standard unit system used for the model.

2.1 Geometry /Meshing Geometry:

Cab was meshed in Hyper mesh 12. Cab image is not shown due to confidentiality issues. The FE model was prepared for durability loading as well as floor stiffness loading. During FE modeling one should be clear about the area of interest. The area of interest should be model with fine mesh. Mesh transition should be smooth and features should be properly captured. There should not be a triangular elements over fillet.The components of the cab assembly were modelled with shell element (CQUAD4/CTRIA3) and (S3/S4) for linear and nonlinear analysis respectively.
The welding was done between floor panel, cowl and cross members. There should not be triangular elements near weld and the bolt holes.
There is no free edges and duplicate elements in the components. Shell normal's are in one direction. The component should pass all quality checks. Empty collector, property, material should be deleted. The components should be renumbered.

2.2 Boundary/Load Condition:

The boundary conditions are such that, cab assembly was constrained at the mounting locations with all translation and rotational degree of freedom. The cab assembly was subjected to inertial loading in lateral, longitudinal and vertical direction, separately.
Modal analysis was performed to identify vibration modes of the system. This identifies the weakest section of the system. We decided to apply the 300 lbs load on the 5inch area at the centre and RH side of the floor to check floor stiffness by considering the person sitting or standing on the floor. Fig.4 shows the loading details.

ISSN: 2278-0181
Vol. 2 Issue 6, June - 2013

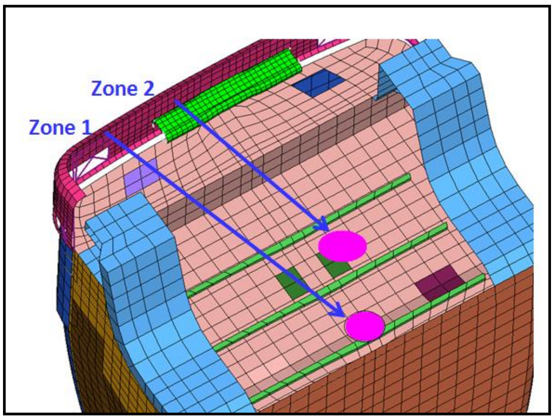

Fig. 4: Loading Details

2.3 Performance Criteria:

Here are some of the criteria planed using Engineering Judgment and the previous experience. The deflection of the floor assembly should not be more than the desired limit. There should not be any permanent deformation in the floor assembly. For durability loading, max stresses are below the yield strength of the material. All the metal components are passes the fatigue life criteria.

3. Results and Discussion:

All the design Iterations were evaluated for Linear as well as non linear materials. Results for current production cross members and optimized design iterations was discussed for linear and nonlinear analysis.

3.1 Linear Analysis

For the linear analysis maximum von Mises stress levels in the current production design of five cross members were below the yield strength of the material. All the components passes the fatigue life criteria.
All suggested three iterations shows the maximum stress level below the yield strength of the martial.

3.2 Non linear Analysis

The maximum displacement in the current production design of five cross member are below the target displacement for centre and RH side loading. The % of elongation in the current production design of five cross member are below the allowable plastic strain of the material. Refer Fig. 5.

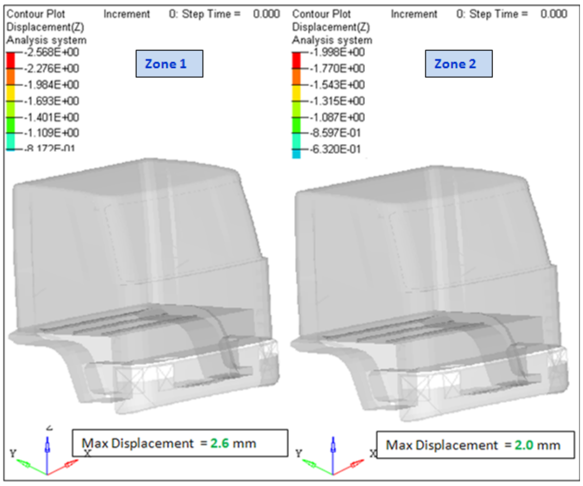

Fig. 5: Deflection Plot for Current production Design

All suggested iterations shows maximum displacement below the target displacement both for centre and RH side loading. The % of elongation in the three iterations are below the allowable plastic strain of the material. Refer Fig. 6.

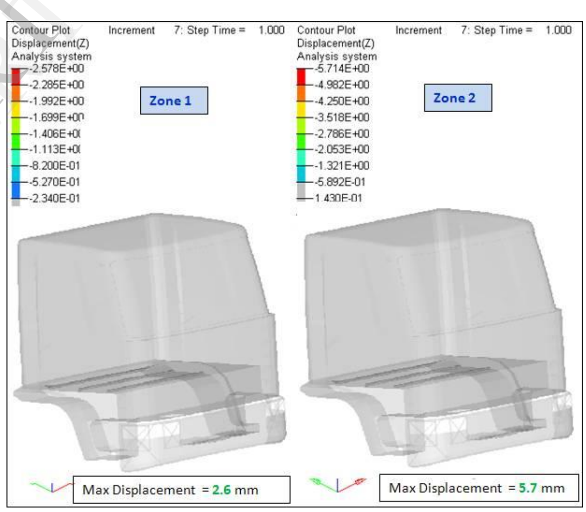

Fig. 6: Deflection Plot for Iteration1

Shaker test is recommended to confirm the final design configuration.

3.3. Costing Summary

Cost reduction is the key to the success of any industry. Weight reduction in the vehicle gives dual benefit to the company. First it reduces cost of the product and second it gives further advantage of additional mileage (fuel efficiency) to the vehicle. In product life cycle, integrated approach of the CAE and conventional design leads to significant reduction in the design cycle time.

With the elimination of the single cross member in iteration 1 ; we can save 1.4 kg of the material. If we assume the cost of the processed steel as Rs 80 and a batch of 10,000 per year we have a saving of Rs 11.2 lakhs. Refer Table 1 for details of all the iterations performed during the study.

Table 1 : Summary of cost saving during analysis study

	Weight of the Reinforcements in the Iteration (kg)	Weight Reduction as compared with Baseline Iteration (kg)	Price of the processed Steel (Rs)	Batch per Year (Qty assumed)	Cost Saved (Rs)
Baseline	7		80	10000	NA
Iteration 1	5.6	1.4			1120000
Iteration 2	4.2	2.8			2240000
Iteration 3	5	2			1600000

4. Conclusion:

Based on the study performed above we come to a conclusion that there is vast scope of Optimization in the concept Cab. Design Optimization is possible with the help of Advance software in Finite Element analysis. FEA analysis can be done effectively to evaluate the strength of the Cabin of the vehicle. Floor deflection is the key parameter in evaluating the strength of the Cab.

Three optimized design iterations are suggested and it shows the acceptable performance considering floor stiffness and durability requirement. This design methodology gave product development a systematic, more precise approach with optimized mass content. The shaker test is recommended to confirm the final design configuration.

High cost in prototypes preparation, specialized set up are some of the drawbacks of the physical test. Finite Element Analysis being a virtual testing facility, can be shared with all types of the testing work and so affordable for various tests for CAE.

.

5. References:

[1] Sethupathi, K. and Somayajula, G., "Conceptual Design Optimization of Cab HVAC System," SAE Technical Paper 941674, 1994, doi:10.4271/941674.

[2] Xin Li, Li Wang.," Based on Topology Optimization Method the Cab Mount Bracket Lightweight Design" Applied Mechanics and Materials, 152-154, 1292.

[3] European Standard EN-1317 Test Vehicle Models
CM/E Group - Politecnico di Milano, Italy
Date posted: 2008 Nov 3 (Unix tar/GNU compressed archive)

[4] Jangyeol Yoon, Wanki Cho, Juyong Kang, et al, Design and evaluation of a unified chassis control system for rollover prevention and vehicle stability improvement on a virtual test track. Control Engineering Practice, Volume 18, Issue 6, June 2010, Pages 585-597

[5] Carol Conroy, David B. Hoyt, A. Brent Eastman, et al, Rollover crashes: Predicting serious injury based on occupant, vehicle, and crash characteristics Accident Analysis & Prevention, Volume 38, Issue 5, September 2006, Pages 835-842

[6] Song, G. and Tan, C., "Shell Elements Based Parametric Modeling Method in Frame Robust Design and Optimization," SAE Int. J. Mater. Manuf. 4(1):716-723, 2011, doi:10.4271/2011-01-0508.

[7] Darko Louit and Rodrigo Pascual., "Optimization Models For Critical Spare Parts Inventories - A Reliability Approach"

[8] Book By Nitin Gokhale "Practical Finite Element Analysis"