International Journal of Research in Advent Technology (IJRAT)
(E-ISSN: 2321-9637)
Special Issue National Conference "CONVERGENCE 2018", 09 th April 2017

Design Optimization of Four Wheeler Seat Components & Its Strength Analysis

Ms. Mrunal Deshpande[1], Prof. K. R. Sontakke[2]
Department of Mechanical Engineering[1,2], PLITMSBuldana[1,2]
Email:mrunal22@gmail.com[1], krs_mech@plit.ac.in[2]

Abstract- Automotive Seat is one of the important element tin vehicle, it gives position , safety ,comfort and aesthetic to occupant. It can absorb the impact or vibration during driving state. In addition to those factors, it is needed to have enough strength for passenger safety. From energy efficiency and environmental point of view lighter vehicle seat frame becomes hot issue in the auto industry. In this paper, weight optimization methodology is investigated for commercial seat frame using CAE.

Optimized designs for car seat frame are developed using commercially available finite element code (ANSYS) and design of experiment method. At first, car seat frame is modeled using 3-D computer aided design tool (CATIA) AND simplified for finite element modeling. Finite element analysis is carried out for the case of ECE 17 head rest performance test to check the strength of the original seat frame. After finite element analysis for the new design with similar load condition to the previous test optimization technique is applied for 10% to 50% weight reduction. Results from the analysis provide an accurate prediction of the material yielding and load distribution on the backrest, provide factor of safety estimates on yield and ultimate strength

Index Terms-seat frame,ECE17, ANSYS,

1. INTRODUCTION

In Automotive industries, OEM and Tier 1 are in tremendous pressure to reduce cost to remain competitive. On the other hand there are stringent occupant safety norms (ECE/AIS) which to be adhered to. According to this tendency, the seat structure, about 4% of the weight of automobile, needs to secure the safety and realize the light weight for increasing the competitiveness of automobiles. To secure the safety and the light weight for automotive seat structure must need to analyze the characteristics of load be supplied to seat structure at the initial structure design step. And through the calculation of numerical values, it must be selected suitable materials and designed suitable section condition for load condition.

Automotive seats are generally constructed from metallic frames covered by foam. Cushions, backrests, headrests, armrests, and other foam parts that make up a vehicle seat are designed according to four principal criteria: integration within the vehicle, safety, aesthetics, and comfort. The design of seat backrest frame, however, is primarily based on safety requirements which dictate structural strength and stiffness targets.

As part of this thesis work, a reference seat backrest frame is considered for optimization under the load requirements of ECE R-17 backrest moment and headrest test. Conforming to ECE requirements is mandatory for vehicles manufactured in Europe. The feasibility of the optimization and the subsequent weight reduction can be attributed to the large safety margin, obtained from the results of linear and non-linear finite element analysis, of the reference seat backrest frame.

Finite element analysis is accepted across a wide range of industries as a crucial tool for product design and optimization. When designing car seats, most of the variables to be considered relate to either geometry or materials. A valuable tool for facilitating and shortening this complex design process is numerical simulation using finite element analysis (FEA). Modeling the seats invirtual environment integrates CAD with material databases and allows the input and evaluation of a variety of loads and stresses without the time constraints of reality testing. FEA can predict the response of a particular design under specific circumstances and supply data that can be used to optimize geometry and materials.

2. OBJECTIVE

The objectives of this paper are follows-

- To optimize seat structure with current designed structure, by comparing the stresses and displacement when subjected to loads according to ECE R 17
- Obtain new optimize light weight design for upper cross member.
- esign done by Catia V5,solid modeling and assembly.

- Linear analysis was performed in sequential manner for predicting displacement and corresponding stresses
- Pre/Post processing was carried out using Hyperworks
- Sequential FEA simulation was carried out using LS-dyna

. METHEDOLOGY

Following steps to follow for optimization

- onduct design review on current seat frame and identify possible component which undergoes for new design.
- repare CAD (catia) proposal model for new back frame upper cross member,
- re-processing (Meshing) of current and proposed seat frame assembly
- pplication of material properties and boundary conditions
- pplication of loading in sequential manner for both the seat frame assembly
  - tep 1 - Application of 373 Nm torque at centre of bracket and measure the displacement and stresses induced
  - tep 2 – Application of 373 nm torque at Hear Restraint at seat displaced condition and then measure the displacement and stresses induced.
- omparison of displacement and stress for current and proposed seat frame component

3. ECE R-17 load for this paper

The headrest test prescribed in section 6.4.1 of UNECE R-17 describes the strength and stiffness requirements of the headrest and backrest. For carrying out this test, the headrest must be placed in the most unfavorable position allowed by its adjustment system (generally it is the topmost position). This test is performed in two steps. In the first step, a load equivalent to a moment of 373 N-m about the R (or H) point is applied rearwards on the back frame of the seat through a component replicating the back plate of a manikin. The first step displaces the seat to a new reference line r1. Fig.1. shows the displaced position of the seat backrest at the end of the first step. In the second step a load equivalent to a moment of 373 N-m about the R (or H) point is applied on the headrest. This force is applied through a spherical ball of 165 mm in diameter, at a distance of 65 mm below the top of the head rest and at right angles to the displaced reference line r1. Under the combined moments of backrest and the headrest loads, the deflection of the headrest must be less than 102 mm from the displaced reference line r1. In addition to the previously applied loads, the load on the headrest block is increased to 890 N as an extension of the second loading step. This increment is a part of the strength requirement and the headrest must not collapse or fail under this increased load.

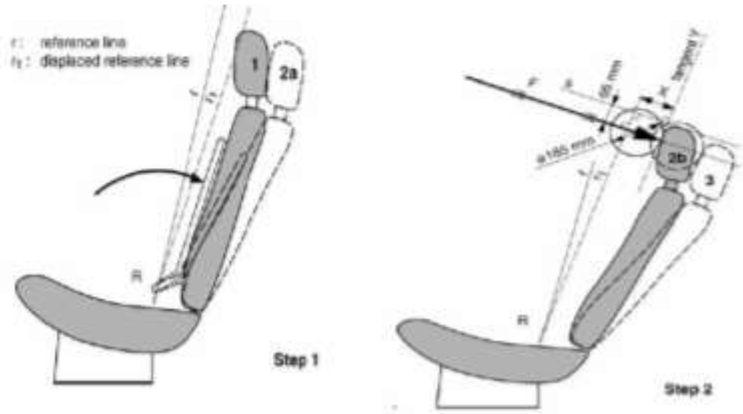


Fig.1. ECE R-17 load case

Step 1: Backrest moment of 373 N-m is applied rearwards about H-point
Step 2: A force equivalent to a moment of 373 N-m is applied on the headrest about H-point. The force on the headrest is increased to a magnitude of 890 N.

This paper does not include 890N load ,as intention is to compare maximum stresses and displacement with current and new design. Load application is point load at center of back frame, and displacement comparisons with old and new design

4. SEAT FRAME

A typical automotive seat back frame is shown in fig.1.it has a sheet metal seatback frame of rectangular shape (from front view), comprising of pair of left and right side frame sections, an upper cross member interconnecting upper portion of side frame sections and lower cross member interconnecting lower end portion of the side frame sections, this seat back frame is pivoted to lower side members, they are fastened to BIW of vehicle. A seat frame houses lumber support bracket, recliner mechanism, back foam support wires and headrest mechanisms.

The upper cross member of seat back frame is considering for optimization in this paper. cushion frame considered carryover frame old design and constant hence not consider during analysis.

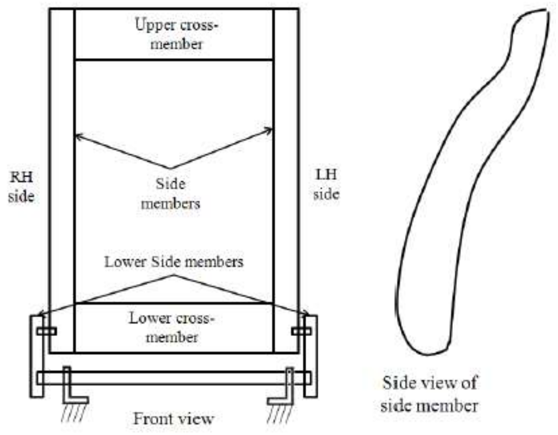
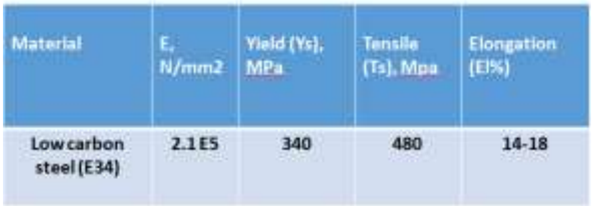


Fig.2. Typical seat frame

5. SHEET METAL PART AND MATERIAL

Material used for this case is low carbon steel. Having material property as shown in Table. 1



Material	E, N/mm2	Yield (Ys), MPa	Tensile (Ts), Mpa	Elongation (El%)
Low carbon steel (E34)	2.1 E5	340	480	14-18

Table 1.. low cabon steel mechanical property

Current design part have thickness of 2.5mm. is a stamp member whereas new part is a standard size C-channel with 1.6mm thickness and 20X20mm size

6. MODEL PREPARATION

Model preparation consist of first cad model which define the design concept and assembly boundary ,second model is mesh model which will undergoes for analysis.

6.1. *CAD model*

Cad model is prepared by using Catia V5,Considering scope here two model for one is current design, and new proposal design, also in this case only back frame is consider keeping cushion frame carry over and constant.Fig.2 shows frame with current design of upper cross and new proposed design upper for upper cross member,

As per CAD current design weight of upper cross member is 1.2kg thickness 2.5mm.propsed design have weight 0.8kg and 1.6mm thick C-channel.

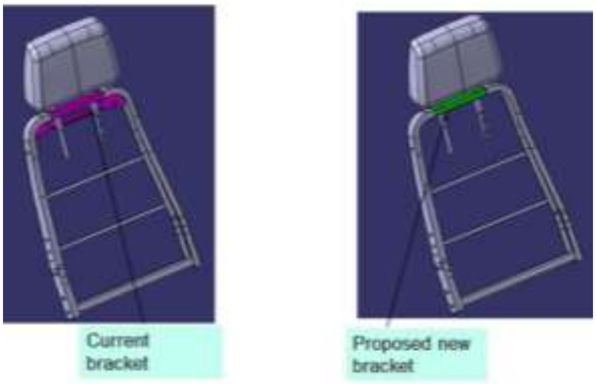


Fig.3. Catia V5 cad model for current and proposed upper cross member

6.2. *Mesh Model*

CAE mesh model is important step in analysis ,mesh model further apply with material ,assembly constrain and boundary conditions is the input for analysis,

Fig.3 shows hypermesh 2D mesh model for current and proposed design, with number of nodes and element

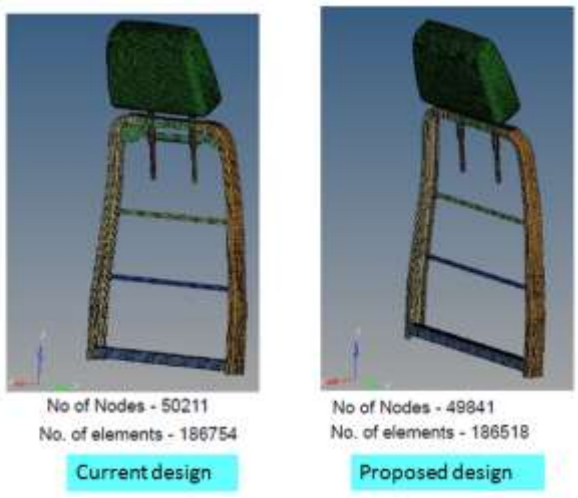


Fig.4. hypermesh 2D mesh model for current and proposed design

7. LOADING AND BOUNDARY CONDITION

Apply material, joints to mesh model, defined rigid and flexible condition. also apply constrain gives preprocessing model for analysis.
Application of loading in sequential manner for both the seat back frame assembly (refer fig 5)

- tep 1 - Application of 373 Nm torque at centre of bracket and measure the displacement
- tep 2 – Application of 373 nm torque at Hear Restraint at seat displaced condition and then measure the displacement

8. ANALYSIS

By applying boundry condition and load as mentioned above compare the result of displacement in step 1 as well as step 2.Fig 5.and 6. Shows displacement and stress for step 1.And fig 7.and 8..shows displacement and stress for step 2

8.1. *Step 1*

When 373 Nm torque is applied on frame bracket (Step 1) following displacements are observed:
•Current Design - 4.490 mm
•Proposed design - 5.463 mm \
And stresses are
Current Design – 381 MPa
•Proposed design – 260 MPa

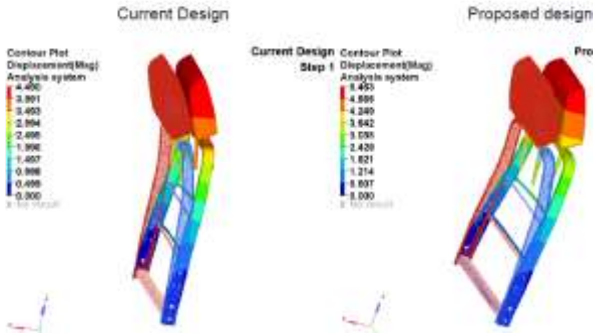


Fig.5. displacement of current and propose design at step 1

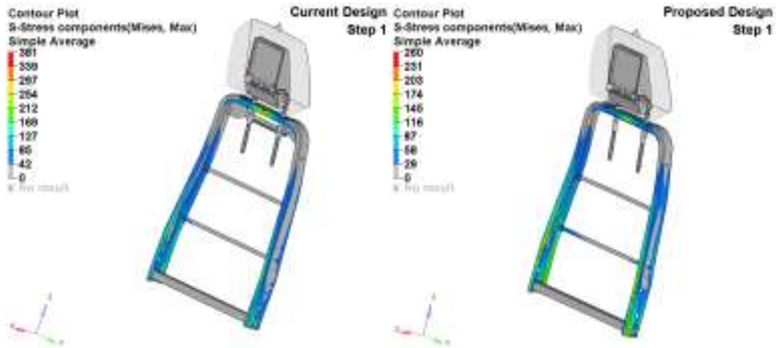


Fig.6. stressed of current and propose design at step 1

8.2. *Step 2*

When 373 Nm torque is applied on Head restraint (Step 2) following displacements are observed:
•Current Design – 36.391 mm
•Proposed design – 31.237 mm
And stresses are
Current Design – 996 MPa
•Proposed design – 664 MPa

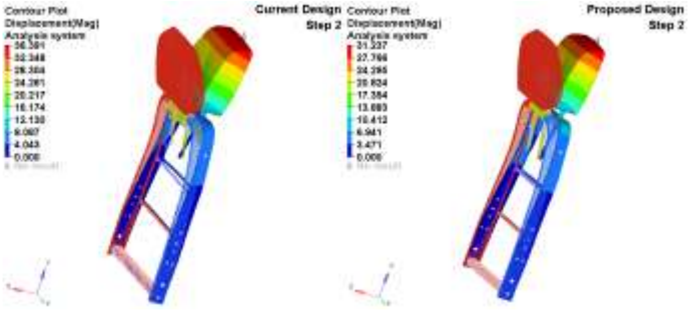


Fig.7. displacement of current and propose design at step 2

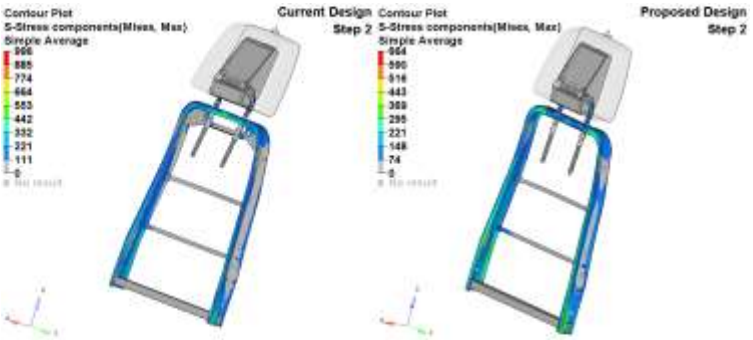
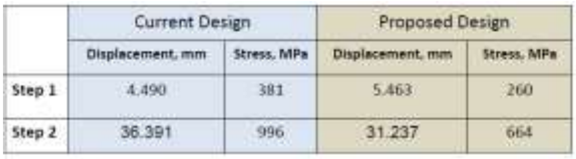


Fig.8. stressed of current and propose design at step 2

9. RESULT AND CONCLUSION

The result summary of stresses and displacement is shown in fig 9.



	Current Design		Proposed Design	
	Displacement, mm	Stress, MPa	Displacement, mm	Stress, MPa
Step 1	4.490	381	5.463	260
Step 2	36.391	996	31.237	664

Fig.9. Result Summary

As the induced stresses in proposed designed are less than current design, also displacement also similar current hence new design can be implemented. Proposed bracket is simple standard size C-channel and light weight too. As its std size less tooling cost is involve than stamped bracket.

Hence this study give optimized design by using Cad and CAE

(A.1)

REFERENCES

[1] United nations agreement Reg.Name:UN/ECE strength of seat and their anchorages Reg. Number ECE-17.07
[2] Altair Hypermesh V9.0 on line manual
[3] “Design Optimization of Vehicle Structures for Crashworthiness Improvement”by Hesham Kamel Ibrahim
[4] http://papers.sae.org/2010-01-0390
[5] Andrzejak D V, Hering W E and Viano D C (1995), “Seatback Load Applying Device”, US Patent No. 5,379,646,
[6] Hallquist J.O.LS-DYNA ,theoretical manual (Livermore software technology corporation,Livermore,CA