Vehicle Seat Design Strength Assessment using Finite Element Analysis

[1]Rahul Joshi, [2]Dr. Sanjay T Purkar, [3] Dr. Dharmendra Kr Dubey
1Research Scholar, [2]Research Supervisor,
[12]Mechanical Engineering,
[1]Bhagwant University, Ajmer, India.

Abstract : Vehicle seat has great importance as it is the primary component of vehicle that provides comfort to the occupants. The seat should have proper strength and durability in case of accidents of when vehicle passes through bumps. If seat structure breaks, severe injury can be occurred to the passengers. In present scenario fuel consumption of the vehicle should be minimum; hence there is a need to weight optimization of vehicle components without compromising the strength. To check these qualities of any physical components, Finite Element Analysis (FEA) is an imperative simulation tool. In this method CAD model of seat structure is prepared and developed FE model of complete seat assembly. Steel properties were used to model seat members. Passenger mass was distributed to the seat floor and seat back.

IndexTerms - **Seat Design, Safety, FEA, Optimization etc.**

I. Introduction

Now a day's road condition of India is getting better. Because of this average speed of vehicles also increases. Passenger commercial vehicles are commonly used by people for their conveyance. Because of high speed of vehicles the frequency of road accidents has also increased. The most representative accident types involving this type of passenger carriers are frontal and rollover crash. Every year thousands of people die because of these accidents. As per Standard Passenger Commercial Vehicle Guidelines, the passenger seating arrangements can be such that seating ability is maximized. There are a number of potential for the array of seats. Passenger seats can be given in crossways, frontward facing pattern or given in longitudinal rows in front of the centerline of the bus. A limited number of backward facing seats can be used with the articulated sanction of the procurement organization. Also it is promising to have a amalgamation of frontward facing and boundary seating planning. The Procuring organization recognizes that ramp site, foot space, hip-to-knee space, seat building etc ultimately affect seating ability and outline. The most typical accident type involving passenger commercial vehicles are side, rear, frontal and rollover. Although rollover crashes did not occur very frequently, when they did, the number of gravely hurt occupants was far above the ground compared to other collapse types. In case of a rollover, passengers run the danger for being uncovered to expulsion, partial expulsion, protrusion, or imposition and thus uncovered to a high casualty danger. In manufacturing engineering, every part of the vehicle should be manufactured by assuring the safety of the passengers. This study includes the seat design of commercial passenger vehicles with safety features during accidents. The study includes the development the seat design with proper safety features.. As in India, people prefer economical way of transit, it would be better if we can offer safety in seat design itself apart from seat belts and airbags. In this study single seater seat of commercial passenger vehicle is developed using advanced simulations technique called Finite Element Methods.

II Literature Review

Seat belts can play an important role to prevent fatalities in accidents. Albertson and others (2006) studied that 128 injured person in rollover accidents. They found that seat belt helped passengers to keep them on their original position. This will reduce the sudden deceleration during rollover crash and reduce the fatalities. Other studies also done by Matolcsy in 2007 and Rona Kinetics in 2002 on rollover accidents. They found that during a rollover, travelers run the danger for being uncovered to expulsion, protrusion, or invasion and thus opened to a high casualty danger.

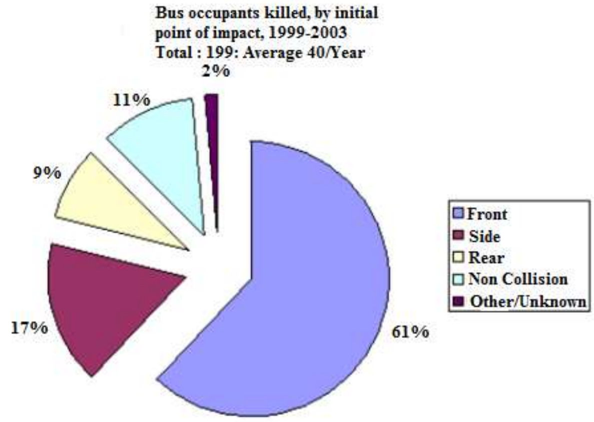

Fig.1 Occupants killed from 1999 to 2003

Mehmet and others performed simulations studies on the effects of seatbelts on these fatalities. Simulation results showed that when dwellers had no seat belt they undergone grave risk of fatalities. Gerardo Olivares performed the study to find the mechanism of fatalities during frontal accidents. They performed s series of tests for frontal impact with crash dummies. The results of this study shown that the most frequent damage mechanisms to bus travelers are head and neck. The main reason behind these injuries is due to the body-body contact between passengers seats.

III RESEARCH METHODOLOGY

As physical tests are often expensive and hard to perform for multiple times. In this situation a strong need of an advanced simulation tool is developed. Finite Element Methods is a key player to solve complex and simple experimental methodology by converting them in the form of simulation. In this work we select YSt 210 having tensile steress is 210 MPa. As per the Indian standards we developed a seat (for commercial passenger vehicle) using FEA.

Table I Tensile Property of Hot Formed Sections

Sr. No.	Grade	Tensile Strength, min (in MPa)	Yield Stress min. (in MPa)	Elongation, Min (in %)
1	**YSt 210**	**330**	**210**	**20**
2	YSt 240	410	240	15
3	YSt 310	450	310	10

Fig. 2 Seat Arm development in FEA (Step-I)

Fig.3 Back Rest Development (Step-II)

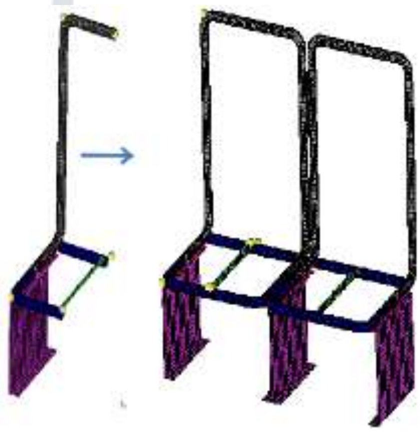

Fig.4 Bottom Rest Development (Step-III)

As we know the following load are considered while design a seat for ant vehicle

(1) Passenger weight

(2) Load act on Corner

(3) Load Act due to Breaking

In this work we only considered passenger weight

Let us take Passenger Weight (W) =100 Kg= 0.1 Ton

$$\text{Force} = \text{Mass} \times \text{Gravitational Acceleration}$$

$$F = 0.1 \times 9810$$

$$F=981\ N$$

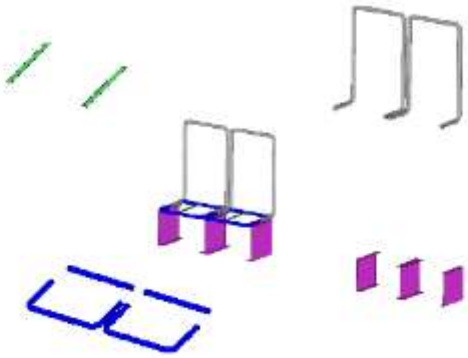

Fig. 5 Different Parts of Seat having steel Tube dia. 1 mm (Step-V)

IV Application of FEA

Now applying FEA on all able four parts of seat step by step and find the stress development on parts. If stress is above the allowable stress i.e 210 MPa than, we modify the dimensions (in tolerable limit) i.e doing iteration than again applying FEA.

A. Step-I First applying FEA on seat Arm i.e. part I, so following stress is plot on FEA. As a result we see that maximum stress developed on knee is **32.152 MPa** which is less than the allowable stress i.e **210 MPa** so design is safe on seat back arm.

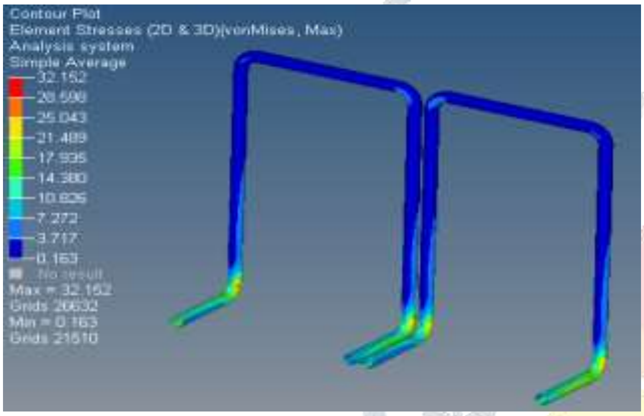

Fig. 6 Application of FEA on Seat Back arm (Part-I)

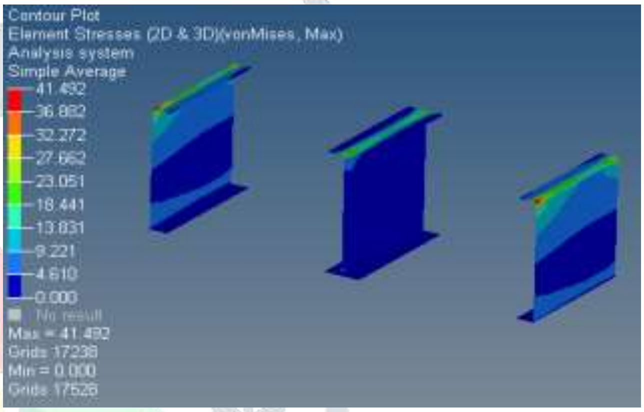

Fig. 7 Application of FEA on Seat Bracket (Part-II)

Step-II Now applying FEA on seat side bracket i.e. part II, so following stress is plot. As a result we see that maximum stress developed on bracket is **41.492 MPa** which is less than the allowable stress i.e **210 MPa** so design is safe on seat bracket.

Step-III Now applying FEA on bottom of seat i.e. part III, so following stress is plot. As a result we see that maximum stress developed on bracket is **274.351 MPa** which is more than the allowable stress i.e **210 MPa** so our design is not safe on bottom of seat. Now we take different iterations while design in not safe or stress is come below 210 MPa.

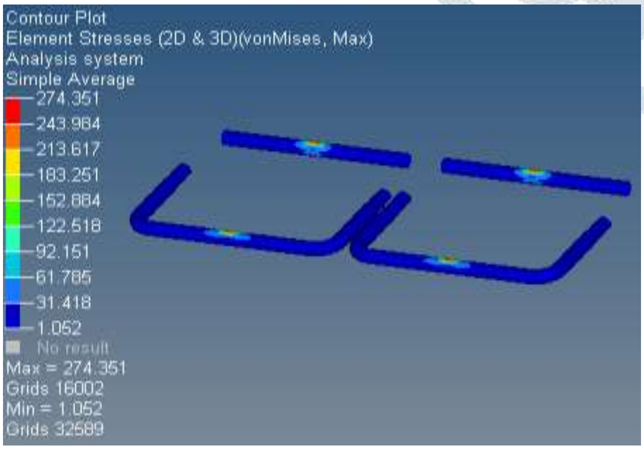

Fig**. 8 Application of FEA on Bottom of seat (Part-III)**

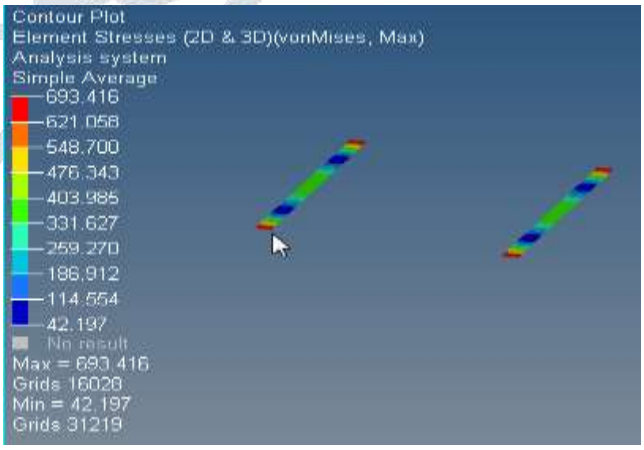

Fig. 9 Application of FEA on Bottom Strip of seat (Part-IV)

Step-IV Now applying FEA on bottom strip of seat i.e. part IV, so following stress is plot. As a result we see that maximum stress developed on bottom strip is **693.416 MPa** which is less than the allowable stress i.e **210 MPa** so our design is not safe on bottom strip of seat. Now we take different iterations on strip while design stress is become below 210 MPa.

B. Iteration: First Iteration As a result we see that the stress in two parts (i.e. in part-I & Part-II) is less than allowable stress i.e. 210 MPa but in remaining two parts (i.e. in part-III & Part-IV) the stress level is below the allowable stress i.e 210 MPa. First iteration is apply on bottom strip having 2 mm thickness (i.e. on Part IV).

So following counter is plot on FEA. In this plot we can see that the maximum stress is **187.579 MPa** which is less than the previous stress **693.416 MPa** and also less than design stress **i.e. 210 MPa as shown in fig. 12**. So for this particular part design is safe.

As we know from precious counter plot the part- III is also unsafe so counter plot of part-III after first iteration as follows. We can see that the maximum stress is **248.625 MPa** which is less than the previous stress **274.351 MPa** more than design stress **i.e. 210 MPa**. So for this particular part design is safe again unsafe. Here again need for more iteration because seat bottom is remain s unsafe although seat strip is safe.

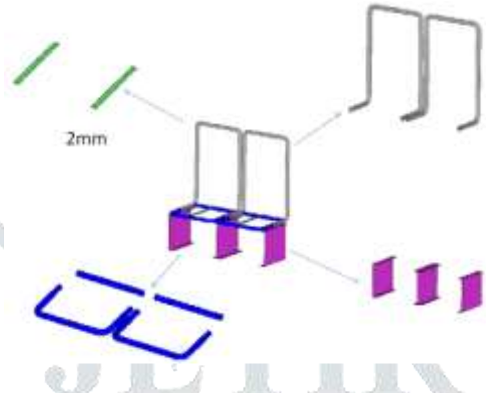

Fig. 10 First Iteration on Seat Bottom Strip (with 2 mm strip)

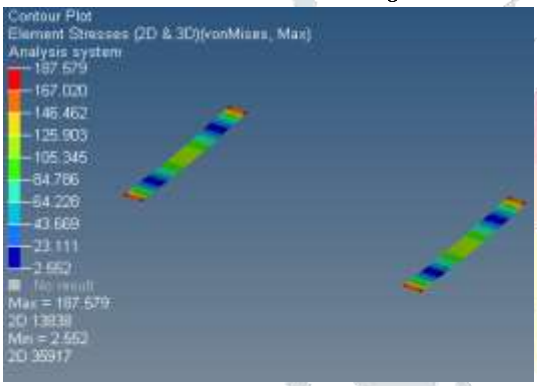

Fig. 11 Result of First Iteration on Seat Bottom Strip on FEA (with 2 mm strip

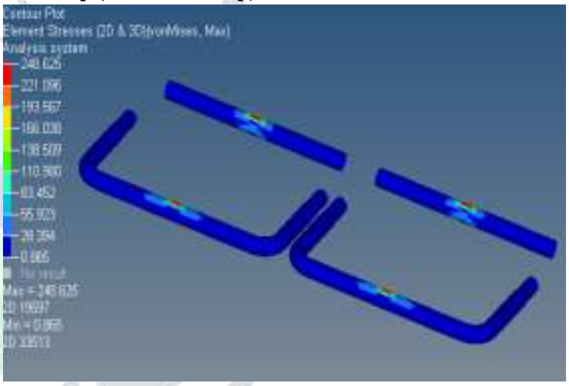

Fig.12 First Iteration on Seat Bottom (with 2 mm strip)

Second Iteration In previous iteration the maximum stress on bottom part is 248.625 MPa, it means that more load is exerted on bottom portion of seat, so for given more strength on bottom portion we join 2 mm more strip with bottom strip as shown in Fig. 13.

It is clear to understand from fig 15 counter plot that part-III & part-IV is safe because the maximum stress develop in pert-III & part-IV is below allowable stress i.e. 210 MPa.

Third Iteration Alougth in First iteration our design is safe but now I am trying to optimize the material weight. In bottom strip we take a 2 mm strip, which is weld. So weight of bottom strip is increase and due to this small increase in weight of bottom strip of one seat, overall weight is increase of vehicle, because a commercial vehicles bus having more than 30 seats.

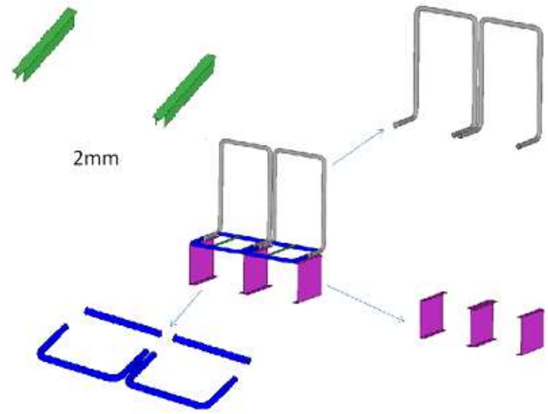

Fig. 13 Second Iteration on Seat Bottom Strip (with 2 mm strip)

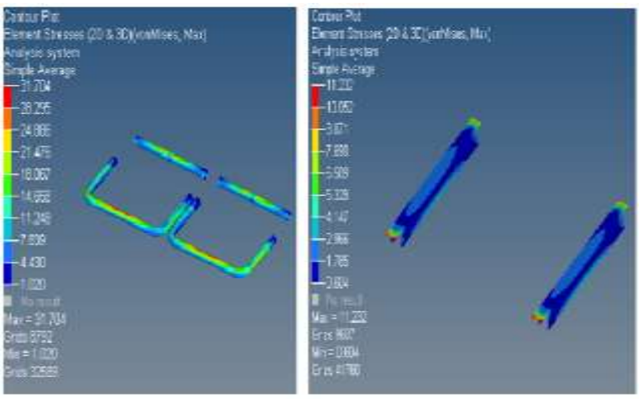

Fig. 14 Second Iteration after FEA (on Part-III & Part-IV)

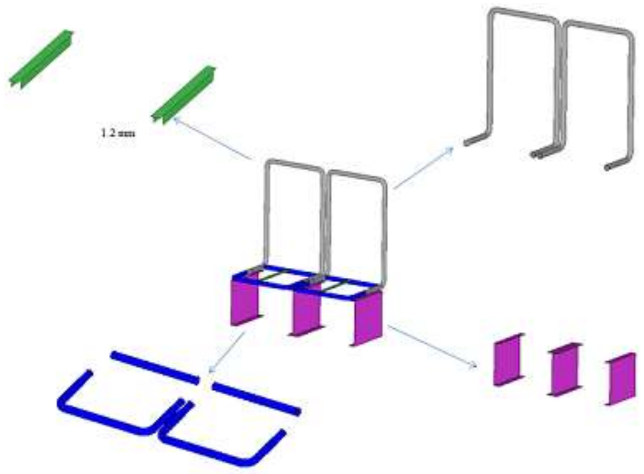

Fig. 15 Third Iteration (weight optimization of Part-IV)

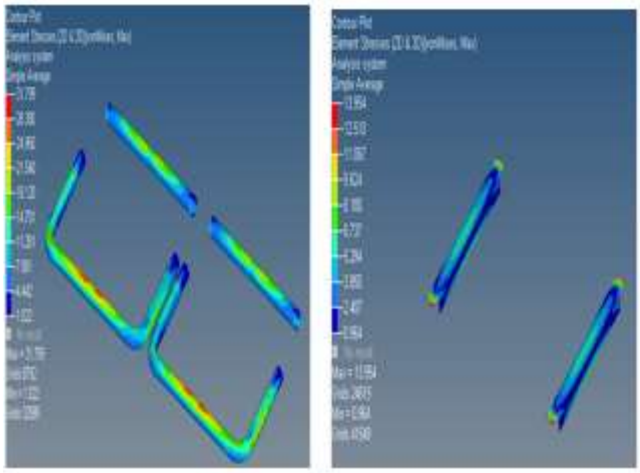

Fig. 16 Result of Third Iteration on Part-III & part-IV of FEA (with 1.2 mm strip)

From the Above figure it is clear that, stress developed in part-III and Part-IV is less than the 210 MPa after optimized the material. The stress in part-III is 31.799 MPa and part-IV is 13.954 MPa.

RESULTS AND CONCLUSION

The main objective of this research is to assess the strength of seat and safety of passenger, so under the view if this objective the work is done on FEA and using FEA we can say that from the figure 17 the strength as well as material assessment is done under the design rule. So due to this result passenger is safe while traveling.

REFERENES

[1] Kirk, R. Grant, (2001),ECBOS Workpackage 1 Task 1.1 "Statistical collection" Final Report, European Union.

[2] Albertsson, P., Falkmer, T., Kirk, A., Mayrhofer, E. and Bjornstig, U. (2006) 'Case study: 128 injured in rollover coach crashes in Sweden–Injury outcome, mechanisms and possible effects of seat belts', Safety Science, Vol. 44, pp.87–109.

[3] Matolcsy, M., (2007), "The severity of bus rollover accidents", ESV Paper 989, 20th ESV Conference, Lyon, France.

[4] Rona Kinetics and Associates Ltd. (2002). Evaluation of occupant protection in busses. Transport Canada, Report RK02-06, British Columbia, Canada.

[5] Anon., Traffic Safety Facts 1999: A Compilation of Motor Vehicle Crash Data from the Fatality Analysis Reporting System and the General Estimates System, National Highway Traffic Safety Administration, National Center for Statistics and Analysis, U.S. Department of Transportation, Washington DC 20590. December 2000.

[6] ARAI Standards, Pune, India.