International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 7, July - 2014

Automotive Seat Modeling and Simulation for Occupant Safety using Dynamic Sled Testing

Dr. Vikrama Singh
Professor Mech. Engineering Dept.Pad.Dr.D.Y.Patil
Institute of Engineering & Tech.Pimpri Pune

Mr. Sohail Ahmed Shaikh
M.E. Mech. Design Engineering Pad .Dr.D.Y.Patil
Institute of Engineering & Tech .Pimpri Pune

Abstract— The study and analysis of damage caused during rear impact on seat structure and Procedure of sled testing, safety requirement can be decided by deformation and damage caused during sled testing it is kind of dynamic analysis. The projected seat went a complete simulation in L.S.Dyna. This provides us to exhibit our ability to build a seat and give confidence in our component technologies as well as our simulation and analysis methods.FE analysis of automotive seat with Dynamic sled testing is carried out using the Altair Hyper mesh v12 tool for meshing and LS-Dyna Explicit Dynamic Solver for analysis. The behavior of the deformation of seat back frame is studied by changing the stiffness of seat back frame for the safety of the occupant

Keyword- Finite Element Analysis, Sled test, Automotive seat, Dynamic Analysis.

I. INTRODUCTION

The objective is to reveal the methods used, limitations discovered and improvements for future analyses. This study is intended to evaluate and reduce the injuries sustained by an occupant and to evaluate the damage displaced luggage causes during impact on the seat structure.

The purpose of the seat development cycle for designing a seat that may well fit diversity automotive environments at time also giving a prospect to go further than component stage design, full scale dynamic testing of an automotive seat simulation and Testing. By the individual advancements in recliner, track, lock, seat pan and Seatback design the seat was developed, using advanced adjuster link and motor layout. The projected seat went a complete simulation in L.S.Dyna. This provides us to exhibit our ability to build a seat and give confidence in our component technologies as well as our simulation and analysis methods .for applicable to as a standard seat platform the North American and European Motor vehicle standards have been selected as minimum acceptance criteria.
Seating Systems:- FMVSS 207
Specifically, Protection From Displaced Luggage:- ECE 17

We considered rear, the simulations is not clearly defined by an FMVSS or ECE regulation. The rear impacts loading are standard versions of OEM necessities. Throughout the design.

II. PROBLEM AND PURPOSE

Problem Definition- In Automotive seats in case of rear impact loading condition the Luggage displaced and seat bending should have some limiting angle or displacement or deformation otherwise it is unsafe and uncomfortable to occupant paper size.

Aim-To Optimize the deformation or displacement of Back frame to H-point as per FMVSS standard 207E and ECE17E for passenger safety by varying stiffness of seat . Dynamic sled testing is carried out using the Altair Hyper mesh v12 tool for meshing and LS-Dyna Explicit Dynamic Solver for analysis. .

III. METHODOLOGY

A. **Geometric Modeling**- In Design the model of seat is modeled in Catia V5 R18 *software as shown in below*

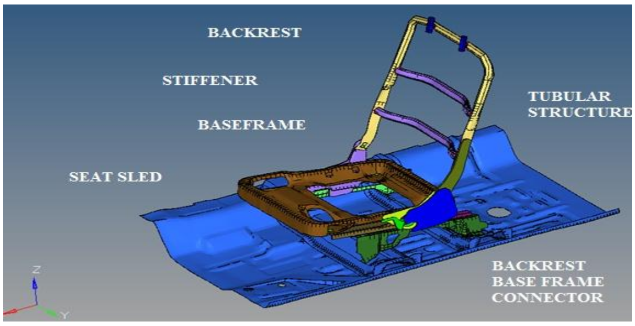


Figure01-Geometric Model of Seat with Sled Modeled in CATIA

B. Finite Elemrnt Modeling and Simulation- The surfaces of the geometric CAD model generated in CATIA V5 are meshed using shell elements to represent the relatively thin sheet metal structures of the seat. Four node quadrilateral elements with size of 8-10 mm have been generated for all the surfaces. Triangular elements are also allowed in the finite element mesh in order to allow good mesh quality

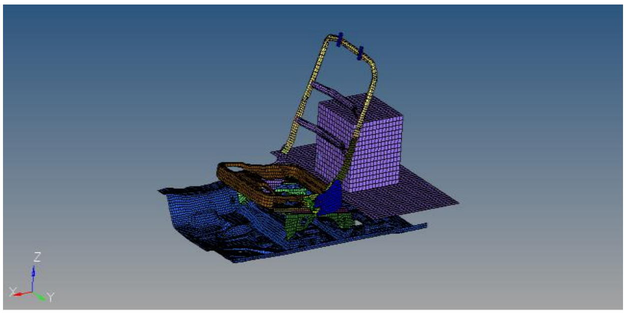


Figure 02: Meshed Model of Seat with Sled

Table 1: Meshing Criteria

1	Average Element Length	10 mm
2	Min. Element Length	5 mm
3	Max.element Length	15 mm
4	Warpage	15^0
5	Aspect Ratio	5:1
6	Skewness	60^0
7	Maximum Quad angle	135^0
8	Minimum Quad angle	45^0
9	Maximum Trias angle	120^0
10	Minimum trias angle	20^0
11	Jacobian	0.6

C. Fasteners and joint – By way of the parts meshed, the a variety of physical connections are modeled. Joints and fasteners must be modeled so that loads and deflections can be determined without adversely affecting the time to obtain a solution. To accurately model the fasteners and joints the actual hardware should be meshed and contacts defined for the part to bolt/rivet/bearing interaction. Unfortunately this requires an extremely fine mesh reducing the time step, which results in an unacceptably long run time. To maintain a suitable time step the fasteners are modeled with beams and rigid multi-point constraints (MPC).

D. Material modeling–Materials were assigned to all the components as shown below. In LS Dyna, there are about more than 150 cards are available for structural materials. These include different materials like metals, rubbers, foams, seat belts, fabric (air bags), and springs. Materials are assigned using *MAT cards available in LS Dyna.In this project work, four different types of materials are used.

1. MATL24 (*MAT_Piecewise_Linear_Plasticity) We assign this type everywhere. All structural and deformable sheet metal parts, BIW, closures and its panel are assigned with *MAT 24 type
2. MATL20 (*MAT_Rigid):- This card is used to define a material which behaves as rigid. In other words, this card is assigned to such components which do not deform
3. MATL100 (*MAT_Spotweld):- This card is used to define materials for spot welds. To use this card, we need to give density of material, poisson's ratio, young's modulus and yield stress as input.
4. MATL1(*MAT_Elastic):- This card is used to define materials which behave as elastic. To use this card, we need to give density of material, poisson's ratio and young's modulus as input.

E.Contact and part interaction - Penalty based contact is used there are two types of penalty based contacts, Automatic and Non automatic type. Automatic type is preferred as it calculates penetration from both the sides where as in non-automatic type it can detect penetration coming from only one side.

- **Automatic Single Surface Contact**:-This is simplest type of contact to define it is used when a surface of one body contacts itself or surface of another body. Its automatically determines which surfaces within a model may come into contact.
- **Automatic Nodes to Surface Contact:-**This contact is used when contacting node penetrates a target surface here flat or concave surface is the target while convex surface is the contact surface. For target surface mesh should be coarser and for contact surface fine mesh is required.
- **Automatic Surface to Surface Contact:-**This contact is assigned when a surface of one body penetrates the surface of another body. It is used for bodies that have arbitrary shapes with relatively large contact areas.

F. **Boundary conditions** -The function of the boundary conditions is to create and define constraints and loads on finite element models.The seat is attached to the floor of an automobile at the slider rails. Bolts at four different locations of the slider rails are used to restrain the seat to floor. The nodes located at the bolt connections are constrained in all 6 degrees-of-freedom at the bolt locations. Figure 6.4 shows the boundary conditions on the finite element reference seat model

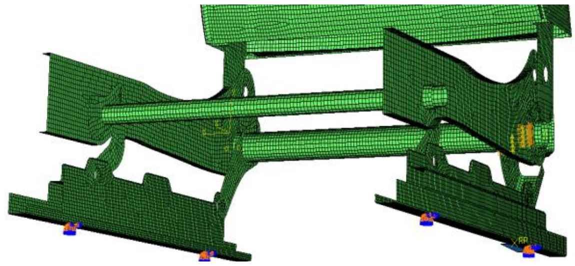


Figure 03: Boundary conditions on reference front seat
Seat constrained at four bolt locations on slider rails

G**. Simulations Method**.

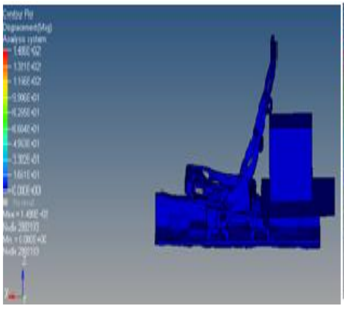
Impact analysis is carried out on tubular section of seat back frame to observe the deformation of the seat and to find the safe design by comparing the results of two simulations.

Simulation 1- Obtained deformation more than the expected one.

Simulation 2-By increasing the stiffness of the seat frame obtained deformation less than the expected limit

VI. SIMULATION AND RESULTS

Simulation 1.



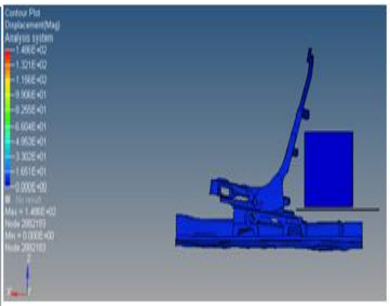
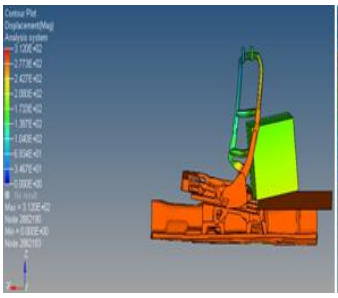


Fig.04: Displacement at Time Step Zero

Fig.05: Displacement at Time Step 0.025 ms

Impact analysis is carried out and we found that the deformation of the seat back is more than the expected limits because of the reduced stiffness of frame members. Hence design is not safe.

we observe that at time step Zero millisecond (ms) the displacement of seat back is zero millimeter because there is no contact between the seat back and block..From figure 05, we observe that at time step 0.025 ms the displacement of seat back is zero millimeter because there is no contact between the seat back and block.



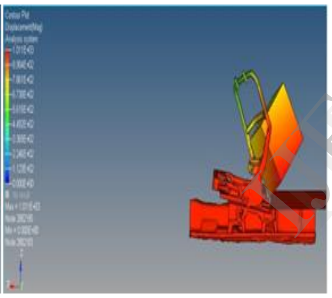
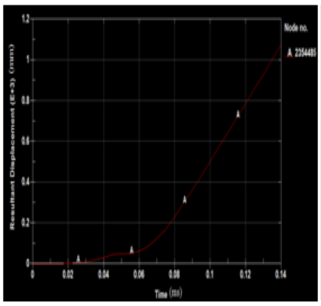


Fig.06: Displacement at Time Step 0.075 ms

Fig.07: Displacement at Time Step 0.125 ms

From figure 06, we observe that at time step 0.075 ms, the displacement of seat back is 277 mm as the block impact on the seat back also From figure 07, we observe that at time step 0.125 ms, the displacement of seat back is 840 mm as the block impacts on the seat back, the seat fails.



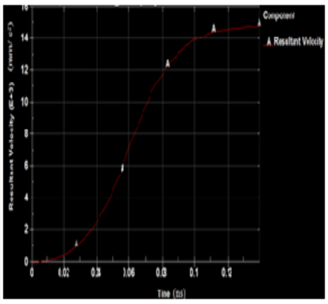


Fig-08 Displacement v/s Time

Fig-09Acceleration with Respect to Time

In the existing design, the cost optimization was focused and two of the back frame stiffening member is eliminated & rest two are adjusted. This will obviously affect & reduce the stiffness of the back frame. So for the design to be safe we carried out two simulations

Table 2: Displacement v/s Time

Sl. No	Time (ms)	Displacement (mm)×10^3
1	0.01	0
2	0.02	0
3	0.03	0.02
4	0.04	0.04
5	0.05	0.05
6	0.06	0.08
7	0.07	0.12
8	0.08	0.24
9	0.09	0.35
10	0.1	0.5
11	0.11	0.68
12	0.12	0.78
13	0.13	0.9
14	0.14	1.08

From the figure 08 we observe that initial displacement of the seat back is nearly equal to zero and as the time step increases then gradually displacement increases and in between time step 0.07 ms to 0.125 ms displacement increase rapidly. At time step 0.125 ms the maximum displacement of the seat back is 840 mm.

Floor has given accelerations as per industry standards & stopped suddenly (deceleration) due to which seat structure experiences the jerk & the output parameters are measured on seat. The seat frame is subjected to an acceleration of 20×g., in the longitudinal direction opposite to that in which the seat folds. From the Figure 09, we observe that the acceleration initially increases rapidly and at the end becomes stable (15 × 10^3 mm/s^2)

Table 3: Acceleration v/s Time

Sl. No	Time (ms)	Acceleration (mm/s^2) x 10^3
1	0.01	0
2	0.02	0.4
3	0.03	1.1
4	0.04	2.5
5	0.05	4.1
6	0.06	6.5
7	0.07	10
8	0.08	11.7
9	0.09	13
10	0.1	13.9
11	0.11	14.5
12	0.12	14.7
13	0.13	14.9
14	0.14	15

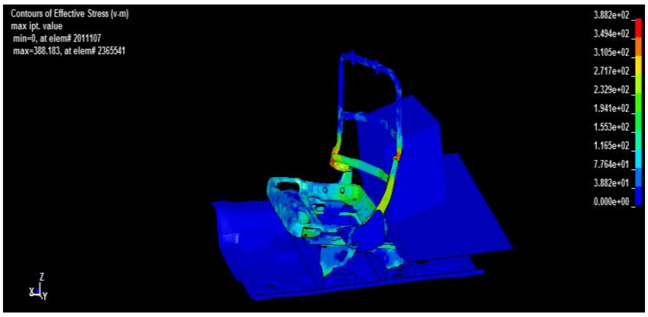


Figure 10: Maximum Von Misses Stress

From the figure 10, we observe the Von Misses stress distribution on the entire seat back and in some places the stress distribution is less and the maximum Von Misses stress occurring on right side of seat back on the element 2365541 is 388.183 MPa at time step 0.125 ms.

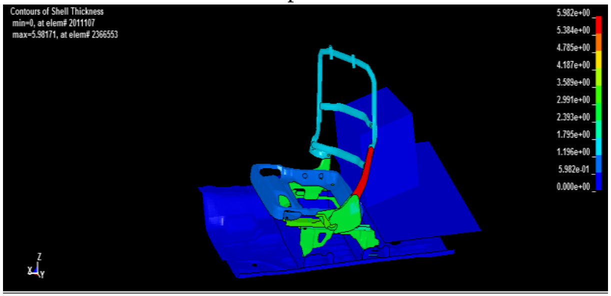


Figure 11: Contours of Shell Thickness

From the figure 11, when the block impacts the seat back, the tubular structure of the seat back thickness reduces and the indentation on the seat back cannot be seen due to reduced thickness as seen in C-shaped structure. we observe that the shell maximum thickness at element 2366553 is 5.98171 and minimum at element 2011107

By observing the above results we found that the deformation of the seat back is quite more than the expected and it violates the AIS regulation as the maximum seat back frame displacement in X-Direction is more than the relative X - coordinate of the seat H-Point.

So conclusion is to change the design stiffness. Hence decided to go for the local stiffness change in back frame tubular & cross members.

Simulation-2-The seat model consists of two stiffening members and the thickness of the tubular structure is 6.25 mm. To compensate the stiffness we have increased the thickness of the local frame and cross members

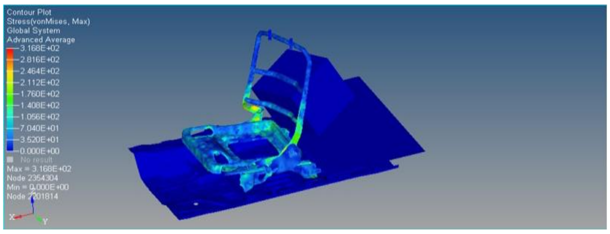


Figure 12: Maximum Von Misses Stress

From the figure 12, we observe the Von Misses stress distribution on the entire seat back and in some places the stress distribution is less and the maximum Von Misses stress occurring on right side of seat back on the element 235404 is 180 MPa at time step 0.125 ms

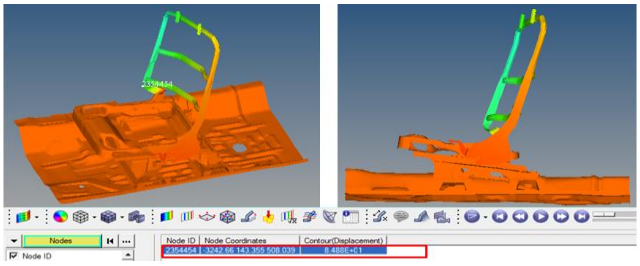


Figure 13: Maximum Displacement at Node =84 mm at Time Step 0.125 ms

From figure 13, we observe that at time step 0.125 ms, the displacement of seat back is 84 mm as the block impact and the deformation of the seat back is within the expected limits and it passes the AIS regulation as the maximum seat back frame displacement in X-direction is less than the relative X - Coordinate of the seat H-Point.

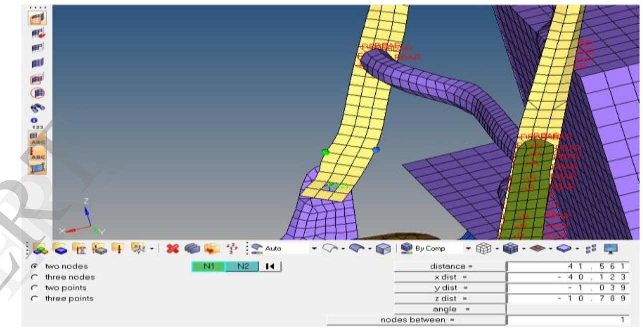


Figure 14: Actual Displacement at Node = 84 – 40 = 44 mm

From figure 14, when the seat back deforms the distance of the seat from the initial position to final position is calculated at time step 0.125 ms, the actual displacement at node is 44 mm which is less than the X-coordinate of H-Point

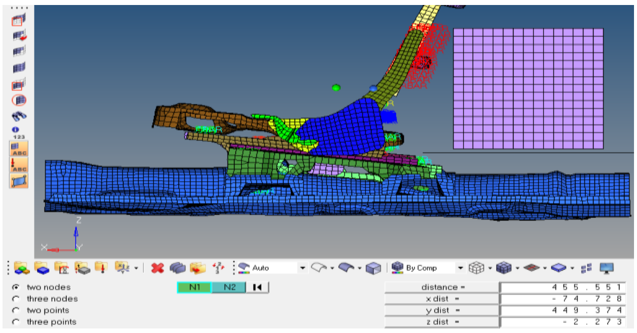


Figure 15: Distance of Seat back from H-point

From figure 15, we observe that the distance between the seat back and the H-Point is 74 mm as per standards for the design to be safe. The deformation of the seat should be within the 74 mm for it to consider as a safe

From the figure 14 we observe the maximum displacement is 44 mm which is less than the 74 mm distance between the seat back and the X-coordinate of H-point as per AIS regulation standard. Hence, the optimized design with reduced reinforced members but with increased thickness is safe & validated for safety

V.CONCLUSION

After completing analysis the total energy absorption, absorption, acceleration and displacement of the seat model are compared for simulation 1 and simulation 2. Following are the conclusions drawn from the work,

By considering the simulation 2 results, we got the deformation of the seat back which is within the expected limits and it passes the AIS regulation as the maximum seat back frame displacement in X-direction is less than the relative X - Coordinate of the seat H-Point position for which seats are designed and it assumes that this nature of deformation will not hit & damage the back portion passengers body. So the seat can be considered as safe in design.

VI. ACKNOWLEDGMENT

We would like to thanks to Department of mechanical engineering Pad.Dr.D.Y.P.I.E.T Pimpri Pune for valuable guidance.

VII. REFERENCES

1. LS-DYNA User’s Manual, Version 950 Livermore Software Technology Corporation, Livermore, CA, USA, (2001)
2. SAE paper Correlation of an Automotive Seat Finite Element Simulation with Dynamic Sled Testing by Timothy N. MacNaughtan and Shafiq R. Khan Dura Automotive Systems (Canada), Ltd
3. Rear end collisions- The Effect of seat belt and crash pulse on occupant motion by Gert Nilson, Mats Y. Svensson Per Lovsund , Ynge Haland and Kristina Wiklund paper no. 94-S10-O07 presented at the ESV Conference in Miinchen ,Germany.
4. Theoretical user manual for L.S.Dyna 3D by John O. Hall quest on June 1991 Oct.1992 Rev.1, July 1993 Rev.2, Livermore Software Technology Corporation, Livemore, CA, USA.
5. Federal Motor Vehicle Safety Standards (FMVSS)
6. United Nations / Economic Commission for Europe (UN/ECE)
7. Mohan D Karambe., “Stress Analysis of Seat Backrest of Car”., 2013