ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

Study of Rear Impact Analysis using Sled Procedure

Dr. Vikrama Singh*, Mr.Sohail ahmed Shaikh
Professor Mech. Engineering Dept. Pad. Dr.D.Y.Patil, Institute of Engineering & Tech. Pimpri Pune, India
M.E. Mech. Design Engineering Pad .Dr.D.Y.Patil Institute of Engineering & Tech. Pimpri Pune

Abstracts

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.

Introduction

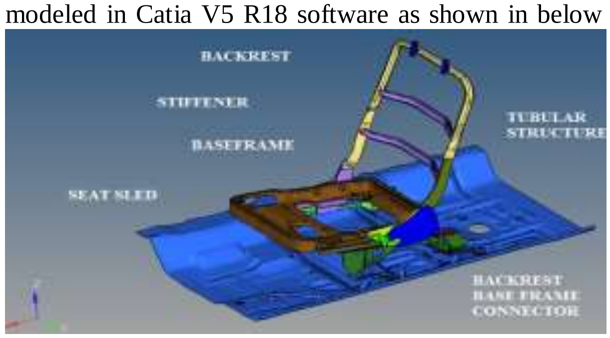
The objective is to reveal the methods used, limitations discovered and improvements for future analyses. This study is intended to evaluate and reduce the injuries sustained by an occupant and to evaluate the damage displaced luggage causes during impact on the seat structure.

The aim of this paper is focused on the calculation of damage caused during rear impact load and validates the FE model using actual tests done in SAE paper. The objective is to expose the approach used, inadequacy and restriction discovered and development for future analyses. This paper is not aimed to estimate and minimize the injuries had by an occupant; rather to estimate 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

An updated information of l.s.dyna, an explicit 3D FE code for analyzing the large deformation dynamic response of inelastic solids and structure. A contact-impact algorithm permits the gaps and sliding along material interfaces with friction.(1)The Comparison of simulations with the test results indicates a reasonable correlate, establishing confidence in FE Methodology used during modeling. Detailed inspection revealed shortcomings in our Modeling and indicates area for future improvements(2) Rear end collision,sled tests were carried out in order to investigate the influence the crash pulse during collision this paper gives the acceleration vs. time diagram**(3)** A vectorized explicit 3D FE code ,This manual help to understand the core concept of L.S.Dyna analysis(4)

Methodology

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



ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

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

FASTENERS AND JOINT – With the parts meshed, the various 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).

MATERIAL MODELING – Based on material cost the design team select the actual steels and thermoplastic used. Once a selection is made a material model to represent the behavior for each component is created. The tabulated Stress Strain data derived from coupon testing served as a starting point for analyses. It is important to note that True Stress Vs. True Plastic Strain is required for analyses beyond the elastic material range, coupon testing generally provide Engineering Stress Strain data. Engineering Stress Strain data does not account for the reduction in cross sectional area during loading or account for drastic cross section change during necking. Refer to Figure 03 for a True and Engineering Stress Strain curve comparison. FE analysis must account for the cross sectional change to predict the deformed shapes.

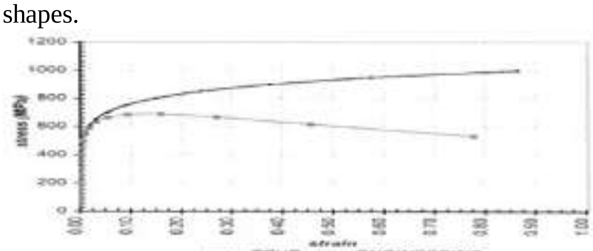


Figure 01 – Stress Strain Curve Engineering & True Data

ELEMENT SELECTION - The selection of element formulation is essential for accurate predictions. Incorrect element selection can result in either under or over predicting the structural stiffness. The classic example would be the exclusive use of triangular elements, double the stiffness of a structure with no change in geometry. A seat structure is primarily sheet metal parts conforming to thin shell theory. For seat applications plate element are assumed for initial modeling. The LSDyna Belystschko-Lin-Tsay type 2 element (default) is very robust and numerically efficient, but suffers from zero energy hourglass deformations. For parts that display high hourglass energies (hourglass energy/ internal energy > 10%) the element is modified to Fully Integrated type 16. Warping stiffness can be added to the type 16 elements converting it to the shell

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

element formulation, to help stiffen the part during out of plane bending (element warping).

CONTACT AND PART INTERACTION - For high deformation "crash" applications modeling part interaction is critical. The structures stiffness is determined by the part deformation and corresponding interactions. The seat track sliding, bending resistance and separation resistance is a direct result of contact. During full vehicle crash simulations all the parts are considered in contact with each other. For seat analysis the contact are broken down the into small groups to help avoid penetration issues for tightly nested parts,trouble shooting contact problems (contact energy / internal energy > 10%) and help maintain distributed contact pressure. Avoiding initial penetration is outmost importance; penetrations can lead to pre-stress, negative contact energy, "shooting nodes" and eventually premature simulation termination.

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 location.

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

Simulation

The initial seat components were manufactured using a "cut and weld" process, which involves fabricating parts from flat sheets of steel and welding the pieces together to form a more complex part. These seats where used to perform initial evaluations of the basic functions of the seats: verify seat kinematics (the various seat positions), check for part interferences on a fully assembled seat (all the wires and plastic trim pieces not usually accounted in the CAD models). To perform dynamic structural testing the initial "cut and weld" prototype models are insufficient to determine the structural response of the seat during dynamic loading. To gain confidence in the testing results soft tooled seat components were manufactured for durability and dynamic impact testing. Soft-tooled parts are production intent parts that can be manufactured much more quickly than hard tooled parts for a lower cost but for a limited quantity.

SLED TESTING - Dynamic sled testing of the seat was performed using a contract testing facility's CVC Horizontal Accelerator. The soft-tooled seat parts were assembled and fastened to the accelerators sled, refer to

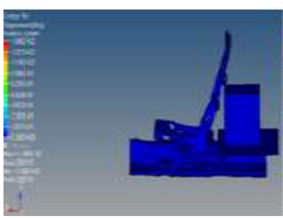
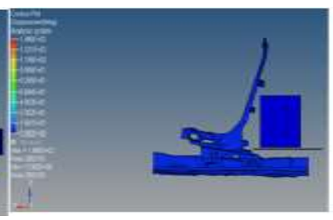



Fig.04: Displacement at Time
Fig.05: Displacement at Step Zero
Time Step 0.025 ms

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.

Figure 01: Sled Testing Set-up (Courtesy:- SAE Paper)

FEA MODEL VS. TEST - The first step in correlation is to review the assumptions made during FE modeling and comparing them with the actual test setup. The analyst must verify that, the parts modeled match the parts tested and the positioning modeled corresponds with physical seat adjustment and placement during testing. The part modeling and boundary conditions assumed during simulations must be equivalent to test setup. After the simulation cycle is complete minor design change can occur. Changes to accommodate anchor point relocation or Occupant H-Point modifications maybe required by the interior or frame manufacturer. Initial steel selection is altered to reduce part costs, ease manufacturing, or prevent specific steel grade or alloy shortages. During prototype parts manufacturing material substitution and geometry changes can occur. The prototype manufacturer may not have the specified steels or steel cost may be prohibitive for low volume soft-tooled manufacturing. The soft tooling used may not be capable of forming the part to the tolerances or geometry specified, forcing simplification of the tools and parts geometry. During the physical testing, the actual seat position, adjustment and fixture positioning may differ from that assumed during simulation. The differences between the test setup and the modeling assumptions must be taken into account to produce a valid correlation. These changes are not intended to force the FEA simulation to match the test result; rather they are made to ensure that the same geometry, materials and boundary conditions are being compared.

AIS regulation standard. Hence, the optimized design with reduced reinforced members but with increased thickness is safe & validated for safety

Conclusion

The purpose of this correlation is to evaluate the idealizations and assumptions made during FE Modeling by comparing the FEA predictions with the results of physical testing. Comparisons of simulations with the test results indicate a reasonable correlation,

By validating the FEA Model we can do further iteration using different boundary conditions varying the stiffness safety of occupant can be obtained.

Acknowledgment

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

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

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

[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. Hallquest on June 1991 Oct.1992 Rev.1 ,July 1993 Rev.2 , Livermore Software Technology Corporation, Livemore, CA, USA.:- This manual explains a vectorized explicit 3D FE code ,This manual help to understand the core concept of L.S.Dyna analysis
[5] Federal Motor Vehicle Safety Standards (FMVSS) United Nations / Economic Commission for Europe (UN/ECE)