Modeling and Analysis and Optimization of Armrest with Implicit and Explicit Method

Vinayak Naik[1], Lokesh[2]
[1, 2] M S Ramaiah Institute of Technology, Bengaluru

***Abstract*-** *Automobile has become one of the important parts of human life. Automobiles with comfort, safety, health, reasonable cost and higher customer satisfaction rate often excel in the market. Comfort is the crucial parameter which is related to seating system of an automobile. The importance of seat design is to satisfy the occupant in comfort, safety, and health aspects. Armrest is one of the upholstery components of a seat, which is seen in the luxurious car which serves the purpose of comfort of the human being The armrest will undergo the different types of loading like abuse loading, lateral loading during crash, due to which armrest material will reach its yield point and permanent deformation will takes place and cause damage to the occupant.*

The project carried out mainly includes the modeling the armrest without affecting the design requirement, mesh the model with optimum element type and element size, creating the suitable contact between the different armrest components, loading the armrest with designed loading conditions. Carryout analysis using LS-Dyna software by explicit and implicit method. And also optimizing the armrest bracket to take specified load.

Keywords- armrest, abuse loading, implicit method, explicit method

I. INTRODUCTION

Automobiles are used to transport people and items from one location to another location. With the popularity of automobiles, the automobile riding comfort has become increasingly important. Comfort is a character that consumers demand for more. The seat plays crucial role in fulfilling these comfort expectations. Seating comfort is a concern for the occupants who are associated to its side effects.[1]

Comfort and the safety of the occupant are mainly dependent on the car seat. Seat incorporates the different component s which will enhance the human safety and comfort. The seating system consists of several component like arm rest, head rest, back rest, tracks etc. [1][2]

This project work is intended to model the armrest without affecting the design requirement, mesh the model with optimum element type and element size, creating the suitable contact between the different armrest components, and load the armrest with designed loading condition. Carryout the analysis using LS-Dyna with different approaches like explicit, and implicit methods. And to find out which is the suitable approach among these. And also, optimize the hinged bracket in order to take the load within the allowable deformation limits.

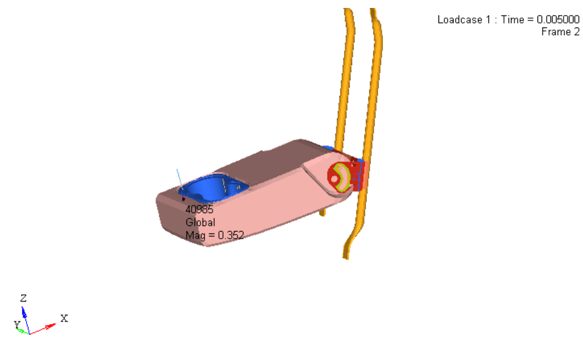


Figure 1:3D model of Armrest

II. PROBLEM DESCRIPTION

The problems observed in the explicit analysis are that Explicit Finite Element Method analysis does the incremental procedure and at the end of each increment updates the stiffness matrix based on geometry changes and material changes. This method is that you do need many small increments for good accuracy and it is time consuming. If the number of increments is not sufficient the solution tends to drift from the correct solution. This method does not enforce equilibrium of the internal structure forces with the externally applied loads.

The analysis is carried out under abuse loading condition with explicit and implicit analysis approaches. Research literature shows comparison of the permanent set, energy absorbed displacement obtained and convergence of result of implicit and explicit analysis methods.

III. FINITE ELEMENT ANALYSIS

3.1 Pre-Processing

The pre-processing of an analysis system involves specifying the material, generating a mesh and defining boundary conditions. The main purpose of an analysis is to evaluate the results. After the boundary conditions are set and loads are applied, we need to specify the desired outcomes of the analysis.

3.1.1 Meshing:

A mesh is the discretization of a component of a component into a number of small elements of defined size. Finite element analysis is dividing the geometry into various small numbers of elements. These elements are connected to each other at point called nodes. Each node may have two or more than two elements connected to it. A collection of these elements is called mesh.

Assumption

- Vehicle floor and body part material are assumed rigid
- All material are assumed to isotropic
- Thickness assumed is nominal thickness.
- Only ultimate stress and corresponding plastic strain (percentage elongation) is considered in the plastic region for all elements.
- Material properties used in the analysis were without failure criteria at high plastic strain values. The advantage of using no failure criteria is that maximum plastic strain in the model at desired location corresponding to particular time step can be found.
- At pivots that are at joints, bolt elements are modeled as ELASTIC BEAM elements with zero rotational stiffness.
- Bolts are modeled with ELASTIC BEAM elements without considering failure criteria.

Quality criteria that have to be maintained are

a) Skewness: Skew in trias is calculated by finding the minimum angle between the vector from each node to the opposing mid-side and the vector between the two adjacent mid-sides at each node of the element.

b) Aspect ratio: Aspect ratio in two-dimensional elements is calculated by dividing the maximum length side of an element by the minimum length side of the element. The aspect ratio check is performed in the same fashion on all faces of three-dimensional elements.

c) Warpage: Warpage in two-dimensional elements is calculated by splitting a quad into two trias and finding the angle between the two planes which the trias form. The quad is then split again, this time using the opposite corners and forming the second set of trias. The angle between the two planes which the trias form is then found. The maximum angle found between the planes is the warpage of the element.

- **Minimum and maximum length**: the length of the elements mainly affects the accuracy, computation time of the analysis

Minimum angle for TRIA
Maximum angle for TRIA
Minimum angle for QUAD
Maximum angle for QUAD

Table 1: Quality criteria

Criteria	Values
Aspect ratio	4
Warpage	15
Skewness	45
Minimum length	2.5
Maximum length	6
Min/ max angle quad	45/135
Min / max angle tria	30/120

3.1.2 Defining the connection:

Figure below shows the different connection between the components. For any component proper connection should be given otherwise there may be chance of improper functioning of the part, relative motion may or may not be defined, rigid component may fail etc.

Nodal rigid bodies (NRB) are used for defining the connection between the components where there is no relative motion is required. The main function is to introduce rigid links between the nodes.

In this model connection is defined between the foam, supporting rod, wire, plastic part a

Revolute joint are used when there is relative rotational motion between the components.

Weld connection is defined using the FE elements between the brackets, supporting rod1 and supporting rod 2

Types of meshing elements

1. **1.**1D elements: one of the dimensions is very large compared to other two Element shape: line
2. 2D elements: two dimensions are large compared to other two dimensions
3. 3D elements: all dimensions are comparable
4. Others: like mass, damper and rigid bodies

3.2 Solution:

In an analysis, after pre-processing is done, the next step is to solve the analysis. In LS-DYNA, solver runs in the background of software and acquires results of an analysis, based on the specified boundary conditions and loading

3.2.1 Structural Analysis

The structural analysis is one the most basic types of analysis. This system analyses the structural components for displacements (deformation), stresses, strains, and forces under different loading conditions. The loads under this analysis system are assumed not to have characteristics (time dependent).

3.3 Post-Processing

After the analysis is complete, we will generate the report in the post processing software. .Structural analysis provides us know-how on structural behavior of a component under a specific boundary and static loading conditions. We can visualize the stress distribution along the component. The stress evaluated through LS-DYNA is compared with the yield stress of the material to decide whether the design is safe or not.

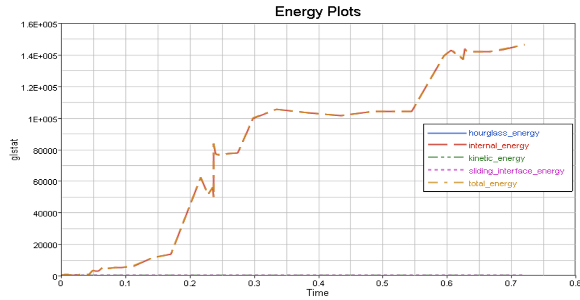
Here the analysis is carried out by applying the load of 360N is applied on the armrest using the 1D beam on the foam and plastic part here the hinged bracket of armrest is optimized to get desired value of the permanent deformation. The passing criteria specified by the OEM is permanent deformation of 12 mm measured from 25mm from free end. And also there should not be any structural deformation.

IV. RESULTS AND DISCUSSIONS

Here the analysis is carried out by applying the load of 360N is applied on the armrest using the 1D beam on the foam and plastic part here the hinged bracket of armrest is optimized to get desired value of the permanent deformation. The passing criteria specified by the OEM is permanent deformation of 12 mm measured from 25mm from free end. And also there should not be any structural deformation.

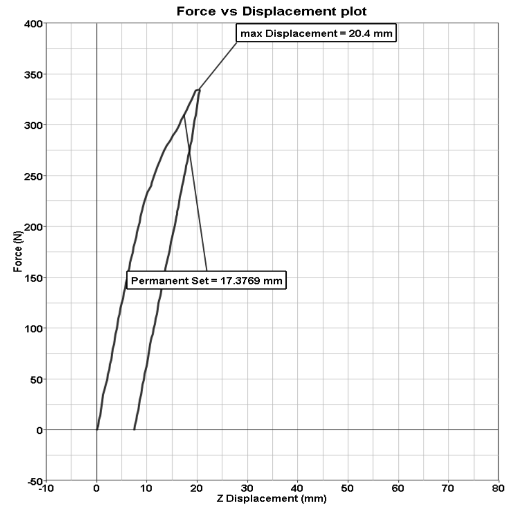
Case 1

Load applied: 360 N
Thickness of hinged bracket: 2mm



Graph 1: energy plot of the arm rest

The graph shows energy plot of the armrest when loaded. The total energy should not be in negative value. If there is any negative value then there should be penetration, or fail in the connection

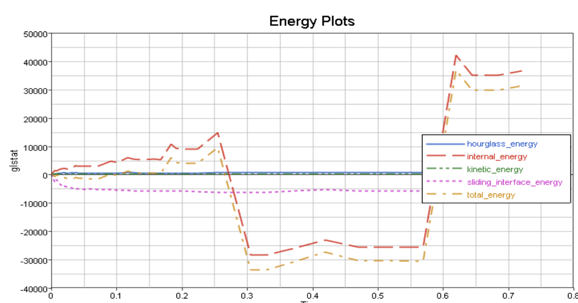


Graph 2: Displacement plot

The graph 7.3 shows a displacement plot. Where the permanent set can be seen as 17.3mm

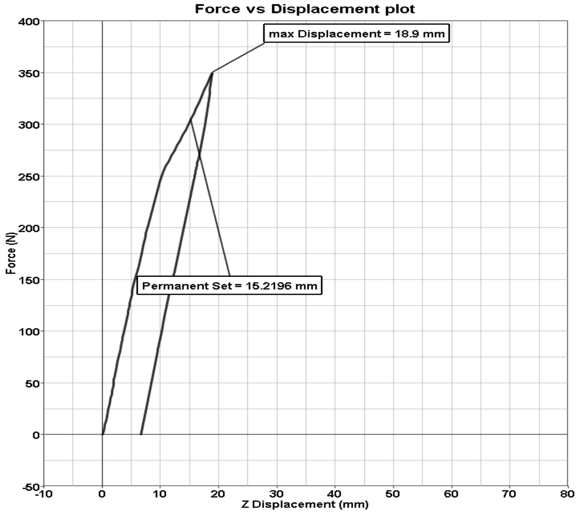
Case 2:

Load 360 N
Hinged bracket with no beads



Graph 3: Energy plot

The graph shows the energy plot where we can see that energy in negative, which shows that loss of energy, takes place due to sliding

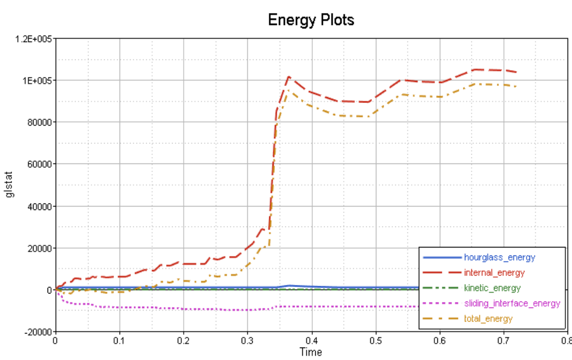


Graph 4: Permanent set

The figure shows the permanent set of 15.2 mm is observed in the current structure

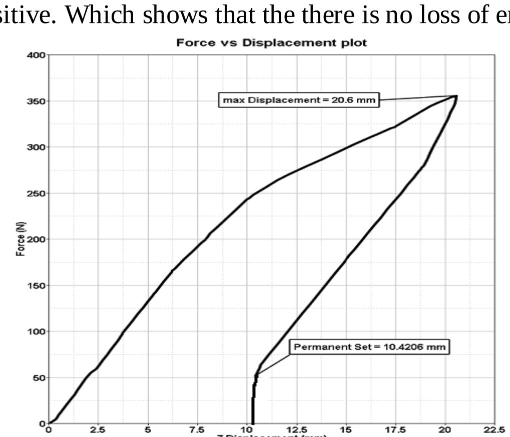
Case 3:

Hinged bracket thickness 2.5mm
Hinged brackets with beads



Graph 5: Energy plot

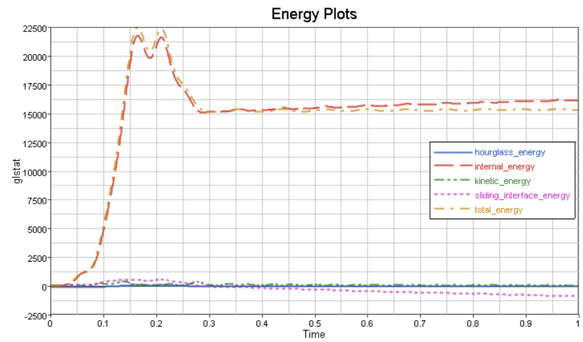
The figure shows the energy induced in the armrest in the positive. Which shows that the there is no loss of energy



Graph 6: Permanent set

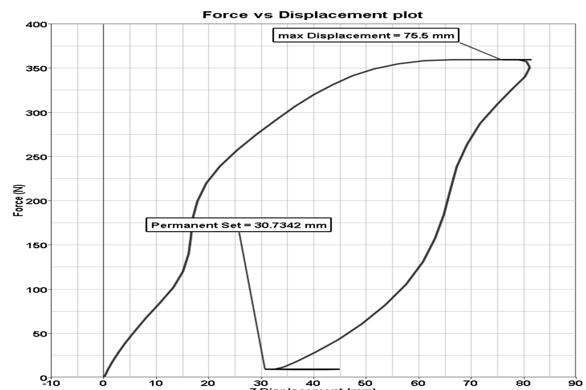
Graph shows the permanent set of 10 .4 mm

Case 3: with explicit method



Graph 7: Energy plot

The graph shows the energy induced in the armrest in the positive. Which shows that the there is no loss of energy



Graph 8: Permanent set

In this graph we can see that permanent set of 30.7 mm

V. CONCLUSION

In this course of study, analysis was conducted on the armrest with abuse loading condition with implicit and explicit method and by optimizing the hinged bracket by varying thickness and design of the bracket. An attempt has been made to obtain suitable method for the analysis and the optimized bracket design. The analysis was carried out to find out the displacement, energy absorbed, permanent deformation within the allowable limit. From the result it can be concluded that stress induced in all the cases are under the allowable limit. The design of hinged bracket with thickness 2.5 mm and with beads is suitable for the armrest. Because it permanent set is under the allowable limit of 12mm specified by OEM. Implicit method is most suitable carrying out the analysis of the small structures as the convergence is more in the implicit method.

REFERENCES

[1] Automobile engineering by Kripal singh.

[2] Hanumant N. Kale, C. L. Dhamejani,"DESIGN PARAMETERS OF DRIVER SEAT IN AN AUTOMOBILE" International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321-7308

[3] The Finite Element Method – Fluid Dynamics by O. C. Zienkiewicz, P. Nithiarasu, R. L. Taylor

[4] Haining Chen, Hao Chen, Liangjie Wang, 'Analysis of Vehicle Seat and Research on Structure Optimization in Front and Rear Impact', Received 21 March 2014; revised 26 April 2014; accepted 3 May 2014

[5] T.C Fai1, F. Delbressine and M. Rauterberg B, 'Vehicle seat design: state of the art and recent development' Proceedings World Engineering Congress 2007 (pp. 51-61), Penang Malaysia.

[6] Rajeshkumar U. More, Dr. R. S. Bindu, 'Comfort Analysis of Passenger Car Vehicle Seat', International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 4, Issue 4, July 2015

[7] J.S. Sun, K.H. Lee, H.P. Lee, Comparison of implicit and explicit finite element methods for dynamic problems, Journal of Materials Processing Technology 105 (2000) 110±118

[8] Alan Mayton, Dean Ambrose, Chris Jobes, Ergonomic and existing seat designs compared on underground mine haulage vehicles, NIOSH, Spokane Research Laboratory, Spokane, WA.

[9] Thomas J.R. Hughes, 'Implicit-explicit finite-element "mesh partitions" are developed for transient problems of nonlinear mechanics'.

[10] W. K. Liu and T. Belytschk, 'Mixed-time implicit-explicit finite Elements for transient analysis'.

[11] Roland Furtado, Steven Reed, Mari Milosic, 'Method of designing automotive seat assemblies for rear impact performance'.

[12] Lee, Ho-Yong; Lim, Joong-Yeon; Bom, Hyung-Taek, 'The Finite Element Analysis of Car Seat Frame According to The FMVSS Strength Test'.

[13] M. Grujicic *, B. Pandurangan, G. Arakere, W.C. Bell, T.He, X. Xie, 'Seat-cushion and soft-tissue material modeling and a finite element investigation of the seating comfort for passenger-vehicle occupants', Materials and Design 30 (2009) 4273–4285.

[14] Song Chen, Yuehui Zhu, 'Quasi-Static Finite Element Analysis (FEA) of an Automobile Seat Latch Using LS-DYNA', Volume 2, Issue 9, September 2014.

[15] Uday M. Jamdade1 , Sandip H. Deshmukh2 , Sanjay S. Deshpande, Modal Analysis of Automotive seating System, IPASJ International Journal of Mechanical Engineering (IIJME), Volume 2, Issue 9, September 2014.

[16] LS-DYNA keyword manual

[17] LS-DYNA theory manual

[18] A M Prior , "Applications of Implicit and Explicit Finite Element Techniques to Metal Forming" J. Mater. Process. Technol. 45 (1994) 649-656

[19] Evaluating the effectiveness of virtual validation methods for automotive seating system by ITC infotech.

[20] Robert Thomson, Mervyn Edwards 'Passenger vehicle crash test procedure developments in the vc-compat Project', VC-Compat Consortium Paper Number 05-0008