Use of computational tools for structural analysis and design modification of automobile seat rail structures under various operating conditions

Prof. Raghu Echempati P.E., Kettering University

Professor Echempati is a professor of Mechanical Engineering at Kettering University, (Flint, Mich.). He is a member of ASME, ASEE, and SAE. He has won several academic and technical awards.

Santhosh Sivan Kathiresan, Graduate Student, Kettering University

Master's degree student in Automotive Systems Engineering, graduated from Kettering University, Flint, Michigan, United States.

Use of computational tools for structural analysis and design modification of automobile seat rail structures under various operating conditions

Abstract

This paper is based on, and in continuation of the work previously published in other conferences [1, 2]. This applied research is concerned with a study of an example automotive seating rail structure. Seat structures, one of the key components, as they withstand the weight of passenger, holds the seating foams and other assembled important components such as side airbag and seatbelt systems. The entire seating assembly is supported firmly and attached to the bottom bodywork of the vehicle through the linkage assembly called the seat rails. These seat rails are adjustable in their longitudinal motion (front and back) which plays an important role in giving the passengers enough leg room to make them feel comfortable. Therefore, seat rails under the various operating conditions, should be able to withstand the complete weight of the human with the seating structures, other assembled parts into the seating, and satisfy functional requirements such as crash safety to avoid or to minimize injuries to the occupants.

Keeping the above requirements in view, the goal of this work is to perform studies on the seat rails under different operating conditions through a detailed investigation using SolidWorks CAE simulation tool for structural and vibration (dynamic) analyses with durability using different grades of steel, aluminum, and multi-materials. Based on these studies a slightly modified design of the seat rail structure is proposed. The modified design is anticipated to increase the fatigue life, decrease the damage percentage, increase the resonant frequencies, factor of safety, and increase the amount of absorbed crush energy. Preliminary buckling studies are also performed. Real life experimentation is not carried out in this work. Effective usage of CAD modeling application and analysis techniques are the final learning outcomes of this research and for better understanding to utilize the appropriate analysis features of the chosen CAE tools, considering the example with the design modifications of seat rail structures.

Introduction

Computer software was developed in a way that they have capabilities to handle all operations incorporated, through the programmed codes. However, these specified capabilities performance purely depends on the hardware specifications, controlling the programmed codes to perform the allocated task. Thus, software has been classified and developed into various categories, and user has the freedom to choose and operate depending on the task needed to get performed. In the present work, SolidWorks and LS-DYNA CAE tools were chosen to perform the design and structural analysis of the seat rail structures in various operating conditions. LS-DYNA software is based on open architecture that allows the level of customization option to suit the platform on which the user can do the tasks. Understanding the functionalities and intricacies of the chosen software tool is very critical to obtain meaningful results. This paper briefly outlines the importance of the computer usage to study the structural behavior of the seating structures.

From an academic point of view, concepts of free body diagrams, solid mechanics and machine design were incorporated for understanding the assumptions made in material properties, load and constraint conditions to obtain factor of safety. Numerical simulation using finite element method is performed for the analysis. Seat structure is analyzed for stress, deformation, fatigue (durability), vibration simulation, and buckling behavior.

Literature Review

Prior research on papers for utilizing computers in educational purpose, made sure that SolidWorks is one of the most identified name among students for the projects works. SolidWorks Simulation was used to perform static analysis project for go-karts in determining the maximum deflection [3], implementing the finite element analysis method using SolidWorks Simulation [4], performing fatigue analysis of die casting machine in SolidWorks Simulation [5]. All the studies [3, 4, 5] provided pathway for students to use SolidWorks simulation for the project assignments, making to understand the vital concept of simulation and in analyzing the engineering problems within virtual environment using computers. Thus, for this work, SolidWorks 2019-x64 bit Student Version has been adopted.

SolidWorks is one of the many solid modeling computer-aided design (CAD) and computer-aided engineering (CAE) programs developed by Dassault Systèmes [6]. Siemens NX is another tool that is also available to students for exploring the functionalities and level of the tool depending on the usage needs in academic courses or for research usage. All the design modifications and structural analysis were performed using this software

LS-DYNA is an advanced general-purpose multiphysics simulation software package which is most primarily opted in performing crash testing [7] in virtual environment using computers, developed by Livermore Software Technology Corporation (LSTC) [8]. This software has the capability to perform analysis of complex analysis problems in the virtual simulation environment. For this work, LS-DYNA 2018 with PrePost 4.6 x64 bit was opted to perform the crash safety analysis and SolidWorks is used to study the buckling analysis of the seat rail structures.

Computer Usage: SolidWorks

From the design point of view in using SolidWorks, the following procedural steps were performed for the present work: Model Design of the seat structure, *Operating Conditions, Materials Considered, Boundary Conditions, External Loads, Mesh Model, Solver Settings, Analysis, and Results*. In this paper, all these steps will be explained in brief.

Model design of the seat assembly is the first step taken in this work for the analysis purpose. Since the real seat shown in Figure 1a is difficult to model in CAD, a similar seat frame geometry available in GrabCAD [9] shown in Figure 1b is adopted. For this research work only the computer 3D CAD data of the seating rail assembly is used from the GrabCAD, by extracting from the remaining child parts of the seat as shown in Figure 2.

a. Physical complete seat assembly   b. CAD complete seat structure assembly
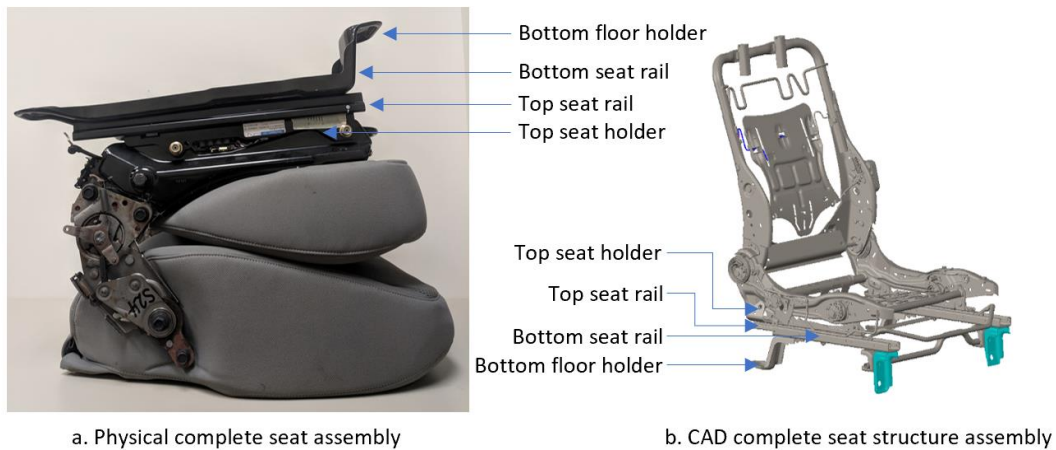
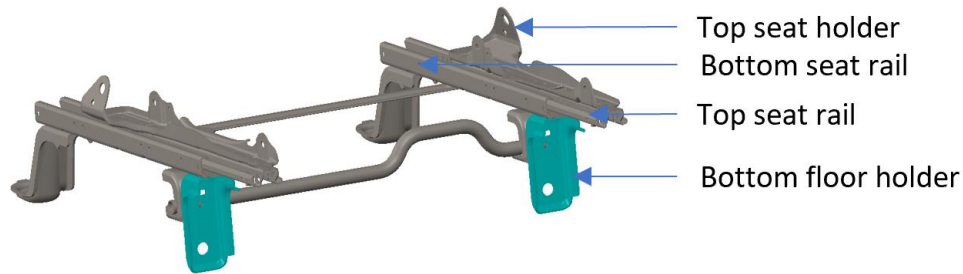Figure 1. Complete physical/CAD seat labelled with seat rail assembly parts [9]

Figure 2. Extracted seat rail assembly from the complete seat assembly

Two operating conditions have been analyzed: one with the seat in the full-back position, and other with the seat in the full-front position. Figures 3a and 3b show both these positions.
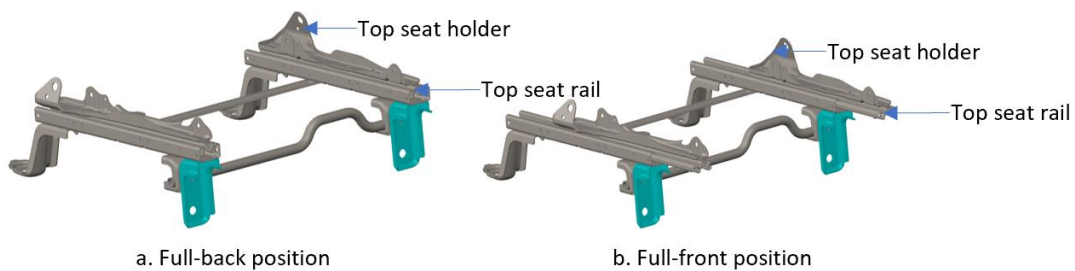

a. Full-back position   b. Full-front position

Figure 3. Top seat holder and rail in full-back/front position

Materials used for the analysis of stress, displacement, durability, and vibration are: high-strength alloy steel only (with yield strength of 620 MPa), 6063 aluminum alloy only (with yield strength of 240 MPa), and multi-materials (combination of alloy steel and 6063 aluminum alloy).

Boundary conditions are defined by fixing the faces of the seat rail bottom mounting points to the vehicle floor body. The remaining child parts in the seat rail are carefully mated and defined with

their contact sets of each sub assembly parts. Figures 4a and 4b show the fixed boundary conditions in the mounting points of the seating rail which are fixed to the vehicle floor body.
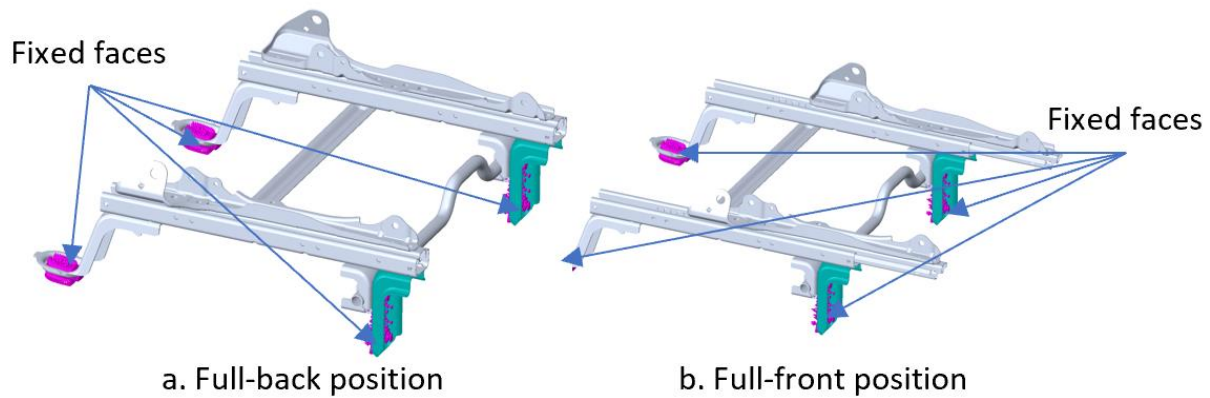

Figure 4. Fixed faces for full-back/front position

To input external loads, the weight of the full frame assembly above the seat rail (10.88 kgs or 24 lbs.), and the passenger sitting on the seat (100 kgs or 22-5 lbs) are considered. Therefore, in this analysis the total force on the top of the seat rail assembly acting through the seat mounting points is 110.88 kgs (around 244.46 lbs.). This is equivalent to a force of 1088 N (with passenger). The loading conditions on the seat rail are shown in Figures 5a and 5b.
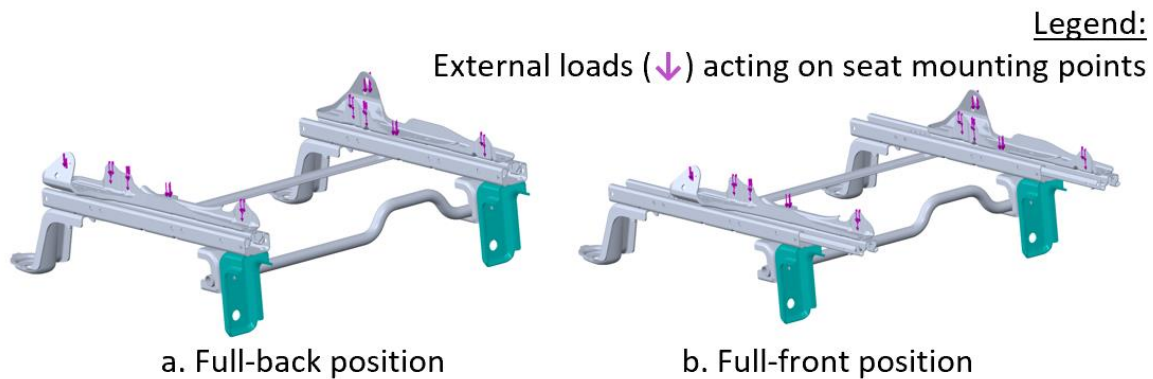

Figure 5. External loads for full-back/front position

For mesh model, the complete seat rail assembly is meshed with the available parabolic tetrahedral elements having 2nd order polynomial function. The meshed results and information are shown in Figures 6a, 6b, and 7. Parabolic elements are also called higher-order elements and defined to have the corner nodes, mid nodes, and edge nodes. Parabolic elements yield better results than linear elements because: 1) they represent curved boundaries more accurately, and 2) they produce better mathematical approximations. However, parabolic elements require greater computational resources and time than linear elements. The minimum size of the elements in the curved regions is around 4.2 mm, while coarse elements of around 21 mm are used elsewhere. Several mesh sizes

have been tried to obtain acceptable results in the fast simulation time. The total number of elements for the assembly are around 150,000.
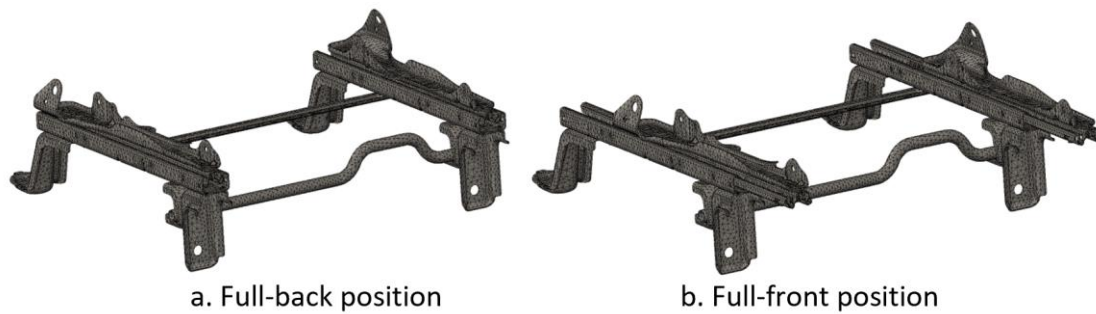

a. Full-back position   b. Full-front position

Figure 6. Meshed model for full-back/front positions

Legend:
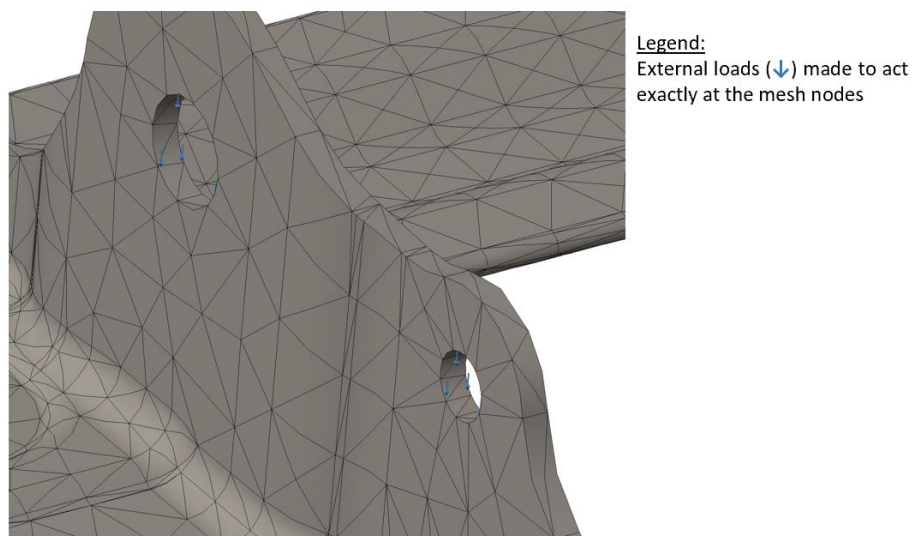External loads (↓) made to act exactly at the mesh nodes

Figure 7. Detailed meshed part showing loads acting on its nodes

For the solver, accurate bonding option available in SolidWorks solver settings has been chosen.

Results

The results of the numerical simulation for the seating rail assembly have been obtained for 6 different cases, 3 for full-back position and 3 for full-front position. Each case is solved for stress and displacement using the materials specified earlier. These results are published in references [1 and 2]. The results show that a factor of safety of 2.0 and above have been obtained for either material usage, and under the assigned operating conditions. Slight design modifications have been done to the seat rail by providing extra material, where fatigue damage is envisioned from the simulations and thus resulted in small incremental weight to the assembly. Also, manufacturing the suggested design feature will involve further machining operations compared to one-step stamping of the seat rail. For the study of buckling, hand calculation were performed first using Euler buckling formula as shown in Figures 8 and 9, and Tables 1 and 2, before performing the numerical simulation using the SolidWorks package. The hand calculations have not matched with

the computer results, as the computer simulation is performed with 3D elasticity equations, while hand calculations use simple 1D Euler beam theory. Due to space limitations, computer results are not shown in this paper.
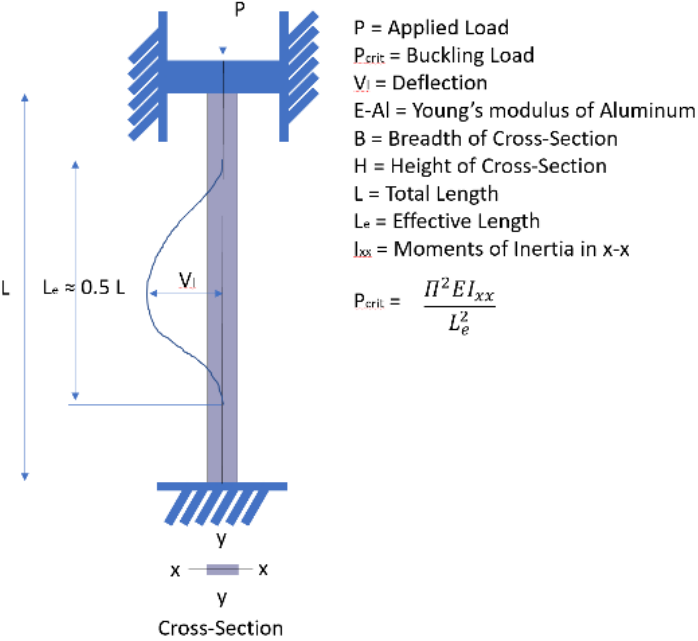

P = Applied Load
P_{crit} = Buckling Load
V_l = Deflection
E-Al = Young's modulus of Aluminum
B = Breadth of Cross-Section
H = Height of Cross-Section
L = Total Length
L_e = Effective Length
I_{xx} = Moments of Inertia in x-x

$$P_{crit} = \frac{\Pi^2 E I_{xx}}{L_e^2}$$

Figure 8: Nomenclatures for the rectangular bar of the seat structure using the Euler's Formula [10, 11]

P = Applied Load
P_{crit} = Buckling Load
V_l = Deflection
E-Al = Young's modulus of Aluminum
D = Diameter of Cross-Section
A = Area of Cross-Section
R = Radius of Gyration = Sqrt(I/A)
L = Total Length
L_e = Effective Length
e = Offset distance
I_{xx} = Moments of Inertia in x-x

$$v_{max} = e\left[\sec\left(\sqrt{\frac{P}{EI}}\frac{L}{2}\right) - 1\right]$$
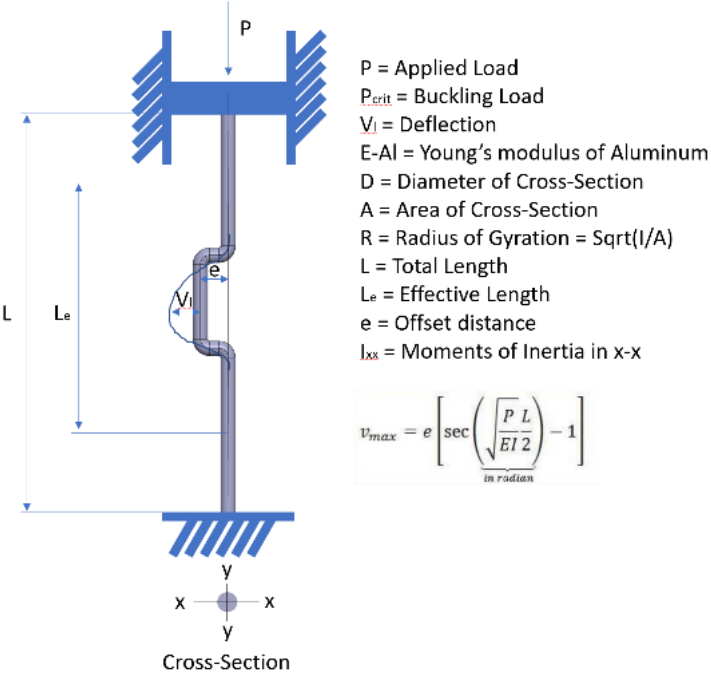in radian

Figure 9: Nomenclatures for the circular bent rod of the seat structure using the Secant Formula [10, 11]

Table 1: Hand-Calculation for the rectangular bar using the Euler buckling Formula

Buckling Analysis of Rectangular Bar			
By Euler Formula			
Pi	3.14		
E-Al	6.90E+04	MPa	
B	33.00	mm	
H	2.60	mm	
Ixx	48.334	mm^4	
L	467	mm	(Original Length)
Le	233.5	mm	(Fixed-Fixed State)
P-Critical (Aluminium)	603.71	N	

Table 2: Hand-Calculation for the circular bent rod using the Secant's Formula

Buckling Analysis of Circular Bent Rod Only			
By Secant Formula			
Pi	3.14		
E-Al	6.90E+04	MPa	
D	16.00	mm	
Ixx	3216.990877	mm^4	
A	201.0619298	mm^2	
R	4	mm	(Radius of Gyration)
e - offset	30	mm	
L	467	mm	(Original Length)
Le	233.5	mm	(Fixed-Fixed State)
P-Critical (Aluminium)	40181.38	N	
Eccentricity Ratio (ec/R^2)	15		
Aluminium	Maximum Deflection (Vmax)-Full length	0.045	mm
Aluminium	Maximum Deflection (Vmax)-Le	0.011	mm

Computer Usage: LS-DYNA

For performing crash analysis of the seat rail structure, LS-DYNA 2018: PrePost 4.6 x64 bit has been used two crash scenarios – full frontal impact (35 mph) and full side impact (20 mph). These correspond to FMVSS standards for the worst material case scenario using the 6063 aluminum alloy. LS-DYNA uses keyword format which is generated by LS-PrePost module. Many procedural steps are needed to be followed to obtain meaningful results. These are outlined in the LS-DYNA User's Manual [12]. Few of these important steps are outlined in Appendix 1 at the end of this paper. As mentioned before, LS-DYNA uses open architecture and so incorrect results can be obtained due to errors in selecting incorrect material properties, incorrect boundary conditions, incorrect contact algorithm, incorrect damping parameters, and incorrect integration points. Each crash simulation performed in this work took around 15 hours of computational time. Therefore, correct preparation of the model and correct understanding of the usage of the software are critical to save effort and time. Due to space limitations, computer results are not shown in this paper.

Conclusions

This paper presents the use of computers in education by applicating the simulation procedures for numerical analyzing tools such as SolidWorks and LS-DYNA using the example - seat rail structures. Understanding the various assumptions made and the procedural steps followed to perform design iterations using these software tools is critical to obtain accurate results. Use of CAE and math tools allow students to explore 'what if' scenarios and develop critical thinking skills needed in STEM careers. Designing components and assemblies that are lighter and safer saves energy and environment as a whole. Use of recyclable materials such as aluminum contributes to these global issues. To conclude, computer usage in this work was very intensive and key take away for readers in developing critical thinking for incorporating essential simulation parameters by considering from applications point of view for the software - SolidWorks and LS-DYNA. Without the usage of computational and math tools, the research work presented in this paper would not have been possible. More technical details and results of this work are available in the previously published papers by the authors [1, 2].

Acknowledgment

References

[1]   Kathiresan, K. and Echempati, R., *Analysis and Design Optimization of Seat Rail Structures in Various Operating Conditions,* (Grand Rapids, MI: American Society for Engineering Education, North Central Section Conference, 2019).
[2]   Kathiresan, S. and Echempati, R., *Structural Analysis and Design Modification of Seat Rail Structures in Various Operating Conditions,* SAE Technical Paper 2020-01-1101, 2020, https://doi.org/10.4271/2020-01-1101.
[3]   Patil, N.R., Kulkarni, R.R., Mane, B.R. and Malve, S.H., *Static analysis of Go-Kart Chassis frame by Analytical and SolidWorks Simulation*, International Journal of Scientific Engineering and Technology, 3(5), pp. 661-663, 2014.
[4]   Chen, Y.D., Bao, Z.Q., Ren, H.J. and Wang, Y.X., *Finite Element Analysis for Product Design Based on SolidWorks Simulation*, Computer technology and development, 2012-09.
[5]   Li, Q.S., Guo, L.J., Wang, W.B., Fan, J.Z. and Su, R.R., *Fatigue analysis of head board of die casting machine based on SolidWorks Simulation*, Journal of Mechanical & Electrical Engineering, 2010-10.
[6]   SolidWorks Education Edition, https://www.solidworks.com/product/solidworks-education-whats-new.
[7]   Atahan, A.O., *Vehicle crash test simulation of roadside hardware using LS-DYNA: a literature review*, International Journal of Heavy Vehicle Systems, 17(1), pp. 52-75, 2010.
[8]   LS-DYNA: http://www.lstc.com/products/ls-dyna.
[9]   GrabCAD Seating CAD Data: https://grabcad.com/library/frame-of-car-seat-1, 2013.

[10] Buckling Aspects: http://www.engineeringcorecourses.com/solidmechanics2/C5-buckling/overview/, 2016.

[11] Robert, C., Juvinall, and Kurt, M., Marshek, *Fundamentals of Machine Component Design,* 3rd Edition, 2000.

[12] LS-DYNA Manuals: https://www.lstc.com/download/manuals, 2004.

Appendix 1: Procedural steps followed in LS-DYNA

- "Import" the IGES format file of the seat rail assembly. [12]
- "Mesh" the model by selecting the appropriate element size. For this analysis element size 5 is opted. [12]
- Select the "Blank" option to check whether all the meshed elements are formed perfectly.[8]
- Create the nodes on the points where the result measurement needs to happen using the "NodEdit" option. [12]
- Fix the nodes which have been created using the "CreEnt" option under the "Constrained -> Nodal Rigid Body (CNRB)" category. [12]
- Select the "Plane" option to create the rigid or the impact wall. [12]
- Select the "ExtFace" option and define the length value to extend the plane which was created in acting as the rigid or the impact wall. [12]
- Select the "Mesh" and activate the "AutoM" option to create mesh on the plane which was developed. [12]
- Select the "EleTol" option and activate the "Transf" module and translate the plane which was created to the distance needed. [12]
- For side impact analysis make sure to rotate the plane by selecting the "Rotate" option. [12]
- Now select "Model" option and activate the "Keywrd" module. [12]
- Now select the "CreEnt" option and activate the "Spc" option under the "Boundary" tab and allow the plane to move only in the direction which is needed to be. [12]
- In the "Keywrd" option, under the "Mass_Part" section, define the mass. [12]
- In the "Keywrd" option, under the "Velocity_Generation" section, define the velocity with the direction using (+) or (-) sign. [12]
- In the "Keywrd" option, under the "Section_Shell" category, define the ID number. [12]
- In the "Keywrd" option, under the "024-Piecewise_Linear_Plasticity" category, input the material's mass density in "RO", Young's Modulus in "E", Poisson's ratio in "PR", and Yield Stress in "SIGY". [12]
- In the "Keywrd" option, under the "020-Rigid" category, input the wall's mass density in "RO", Young's Modulus in "E", and Poisson's ratio in "PR". [12]
- Select the "PartID" section and connect the "Section ID" with the correct "Material ID" which was formed till now. [12]
- In the "Keywrd" option, under the "ASCII_Option" category, select the required parameters needed, like "GLSTAT" checkbox, "NODOUT" checkbox, "MATSUM" checkbox, "RDCFORC" checkbox. [12]
- In the "Keywrd" option, under the "Binary_D3Plot" category, select the correct ID which is needed. [12]
- In the "Keywrd" option, under the "Control_Termination" category, select the time on which the simulation should finish. [12]

- In the "Keywrd" option, under the "History_Node" category, pick all the nodes on the design model where the reference point needed to be placed, in order for the result to be measured. [12]
- In the "Keywrd" option, under the "Automatic_Single_Surface" and "Automatic_Node_To_Surface" category, define the contact between the design model and the wall surface. [12]
- For side impact analysis make sure to select the correct surface contact from the "Automatic_Single_Surface" option. [12]
- Now under "Application -> Model Checking -> General Checking" look for any errors which are getting listed in the "Keyword Check" tab. [12]
- Now save the Keyword in desired workspace location and run it to anlayse the results. [12]

All parameters in LS-DYNA are considered in the specified units as mentioned below:

- Length in mm
- Mass in kg
- Time in ms
- Temperature in K
- Work and Energy in J
- Acceleration in $mm . ms^{-2}$
- Area in mm^2
- Frequency in ms^{-1}
- Velocity in $mm . ms^{-1}$
- Volume in mm^3
- Angular Acceleration in $rad . ms^{-2} = ms^{-2}$
- Angular Velocity in $rad . ms^{-1} = ms^{-1}$
- Density in $kg . mm^3$
- Pressure, Stress, Young's Modulus in $GPa = kN . mm^2$
- Force in kN
- Moment in kN . mm
- Stiffness in $kN . mm^{-1}$