9th ANKARA INTERNATIONAL AEROSPACE CONFERENCE AIAC-2017-148
20-22 September 2017 - METU, Ankara TURKEY

ENERGY ABSORPTION MECHANISMS AND CRASH ANALYSIS OF HELICOPTER SEATS

Gülce ÖZTÜRK[1] and Prof. Dr. Altan KAYRAN[2]
Middle East Technical University
Ankara, Turkey

ABSTRACT

Crashworthiness is the survivability of occupants inside a vehicle during a crash. In helicopters, crashworthiness is ensured by three subsystems; the landing gear, floor structure and the seats. The objective of this paper is to investigate the method of the energy absorbing mechanisms used in helicopter seats. European Aviation Safety Agency (EASA) established CS 29.562 emergency landing dynamic conditions and requirements in which survivable loads and crash conditions are defined for large rotorcraft. In this paper, these conditions are implemented using the explicit finite element (FE) code in order to simulate how the seat energy absorbing mechanism works. Detailed FE model for the seating system with the structural loading is set up in ABAQUS. The seating system is composed of the main seat structure (seat bucket), seat legs and the damping mechanism. Analyses are done with and without the absorption mechanism in order to see the effectiveness of the mechanism on the human survivability by comparing the G loads on the seat bucket with the acceptable loads given by the administration.

INTRODUCTION

Air transportation takes the place of other transportations since it is time efficient. In 2012, the number of people traveling on airplanes reached 2,957 million, which was 4.7%more than the previous year, which is equivalent to 42% of the world's population [Renner, 2013]. In response to this growing demand, regulations did also grow in order to make sure that the air travel is safe for the passengers.

EASA defines the crash conditions for the seat as well as the fuselage of the rotorcraft in order to reduce the G loads that come to the occupants to the acceptable levels. The focus about the dynamic conditions is mainly on the energy absorbing mechanisms of the seats.

To verify capability of the energy absorbing system, the dynamic behavior of the seat is generally predicted by simulation. Analysis of complex systems by numerical simulation is more efficient than the actual full-scale testing in the aspects of time and cost. Repetition of test is inevitable in case of failure, which means even more time and money lost. This is why "Certification by analysis" is very popular in the industry. LS-DYNA [LSTC, 2015], DYTRAN [MSC Software, 2013], PAMCRASH [ESI Group, 1985] and ABAQUS [ABAQUS Inc., 1978] are very useful finite element codes in order to simulate the behavior of systems under dynamic conditions.

[1] M.S. Student in METU Aerospace Engineering Department, Email: gulce.ozturk@tai.com.tr
[2] Prof. Dr. in METU Aerospace Engineering Department, Email: altan.kayran@ae.metu.edu.tr

Bhonge argued that dynamic testing in support of seat certification is expensive and methods such as finite element analysis reduce the cost and give chance to change the design parameters repetitively and understand the effect of each parameter on the design [Bhonge, 2008]. In Bhonge's study a dynamic finite element analysis (DFEA) model of the passenger seat is developed and challenges in simulation are revealed. Direct impact effects on the seat structure are evaluated from different point of views such as comfort, crashworthiness and manufacturing. Moreover, validation of the FE model is also made by comparing the DFEA results with the actual test results, and it is concluded that computer models are useful and less expensive tools for crash analysis.

Simitcioglu & Dogan states that using computer aided dynamic finite element techniques instead of actual tests provides cost and time saving [Simitcioglu & Dogan, 2013]. They supported their argument with an analysis and the meshed seat geometry used in their paper is shown in Figure 1.

Figure 1: Meshed seat geometry [Simitcioglu & Dogan, 2013]

In this study, the simulation mainly focuses on occupants' safety and rotorcraft seat structures under CS 29.562 dynamic test conditions by using ABAQUS. Analysis shows how the energy attenuation system works by comparing the crash results with and without the mechanism.

METHOD

Dynamic Impact Test Parameters

A minimum of two dynamic tests is required to assess the performance of a rotorcraft seat, restraints, and the related interior system. CS 29.562 defines test conditions as [EASA, 2012]:

1. The rotorcraft's longitudinal axis is canted upward 60°, with respect to the impact velocity vector, and the rotorcraft's lateral axis is perpendicular to a vertical plane containing the impact velocity vector and the rotorcraft's longitudinal axis, as seen in Figure 2.
    - Peak floor deceleration must occur in not more than 0.031 seconds after impact and must reach a minimum of 30 g.

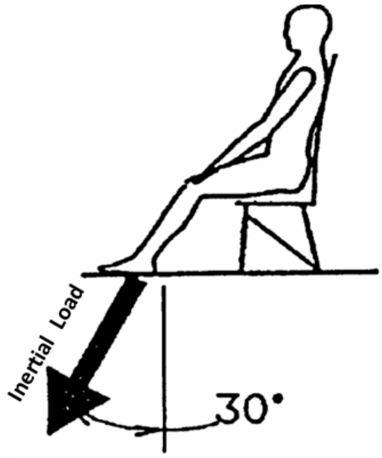


Figure 2: Seat/Restraint System Dynamic Test 1

2. The rotorcraft's longitudinal axis is yawed 10°, either right or left of the impact velocity vector (whichever would cause the greatest load on the shoulder harness), the rotorcraft's lateral axis is contained in a horizontal plane containing the impact velocity vector, and the rotorcraft's vertical axis is perpendicular to a horizontal plane containing the impact velocity vector as seen in Figure 3.
    - Peak floor deceleration must occur in not more than 0.071 seconds after impact and must reach a minimum of 18.4 g.

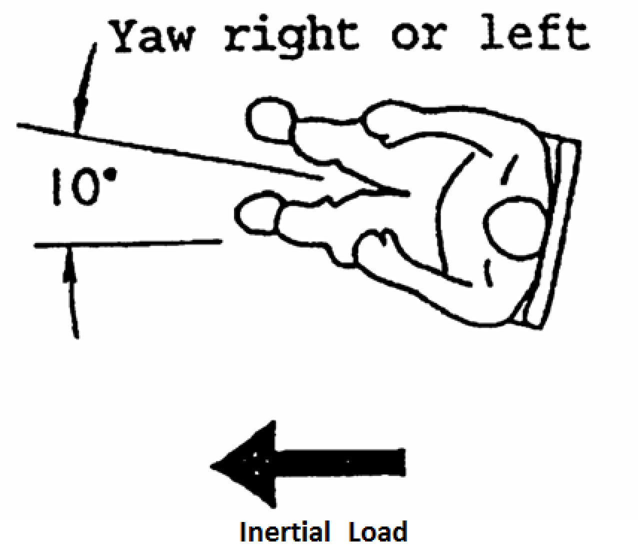


Figure 3: Seat/Restraint System Dynamic Test 2

Tests also require that the floor is deformed by at least 10° in pitch and 10° in roll in order to simulate the actual crash case.

In this study, the simulation mainly focuses on occupants' safety and rotorcraft seat structures. Only downward test definition is implemented on the crash analysis among the test definitions listed above. By implementing this test condition on crash, one can;

i. Evaluate the structural integrity of the seat and
ii. Track data on seat bucket displacement, velocity, and acceleration time histories [SAE AS8049, 2005].

The survivability of human from a crash depends on the loads that come to the lumbar and pelvis area, shown in Figure 4, from the seat bucket. For the prediction of occupant safety, there is a triangle in EASA requirements that shows the G load and its time interval, which mainly defines the test conditions. The triangle says that peak floor deceleration must occur in not more than 0.031 seconds after impact and must reach a minimum of 30 g.

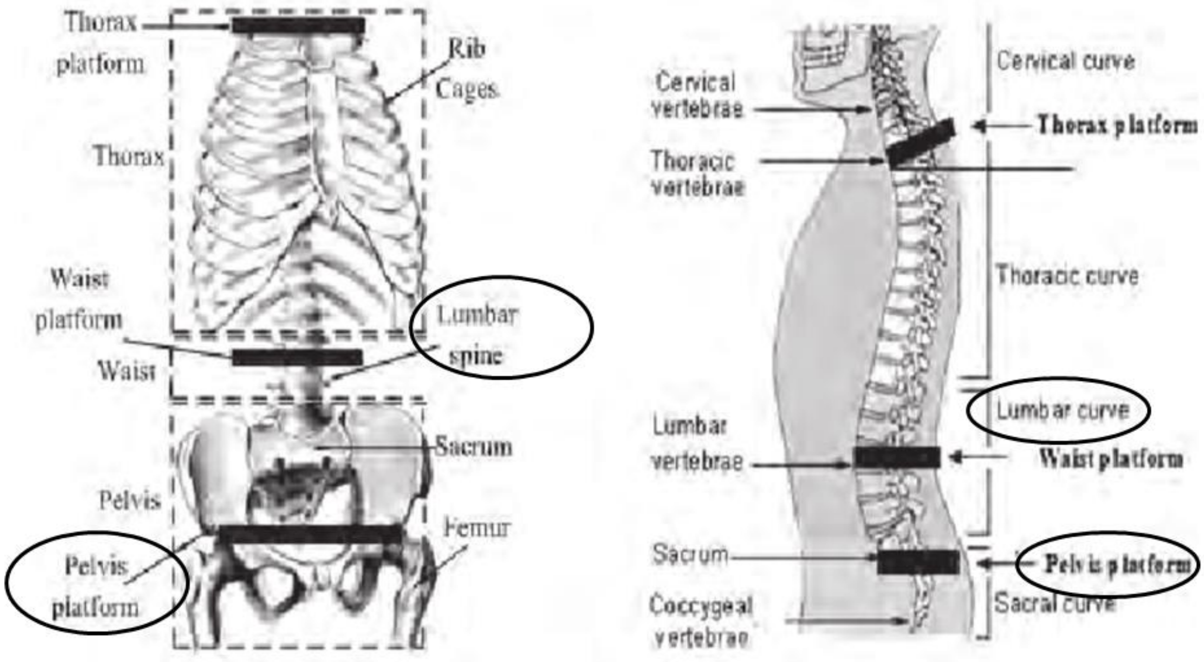


Figure 4: Schemes of human body

In this study two seat models are used. Model-1 is the seat structure in which the damping mechanism is removed and only seat bucket and legs are included. Model-2, on the other hand, includes the seat bucket, legs, lower damping part including the sled and upper damping part that moves into the sled and deforms the column inside the lower part. The main purpose is to show how effective the seat damping system is in an event of crash.

Seat and Energy Absorption Mechanism Model

For the purposes of the present study, a seat is designed as shown in Figure 5 by using CATIA V5. Two different seat models are created. The Model-1 is created as a seat structure in which the damping mechanism is removed and only the seat bucket and the legs are included. Model-2, on the other hand, designed such that it includes the seat bucket, legs, the lower damping part including the sled and upper damping part that moves into the sled and deforms the column inside the lower part. The most important part is the absorbing system of the Model-2 seat, which has to be effective in load reduction that comes to occupant.

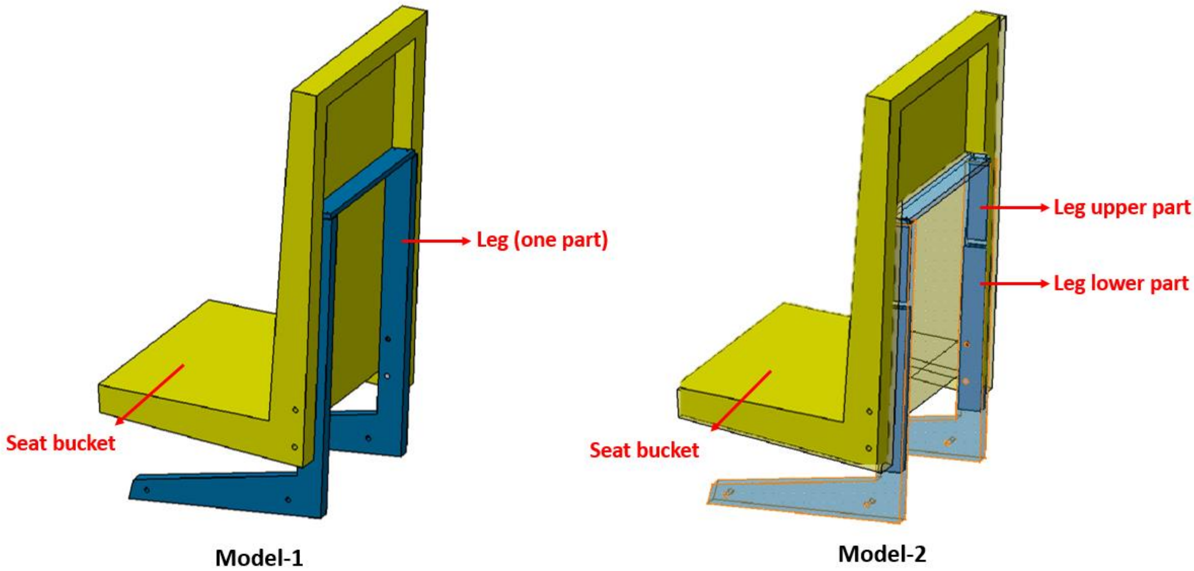


Figure 5: Seat model-1 and model-2

There are a lots of absorption mechanism used in the field. Some of them are listed below:

- Crushable Column
- Rolling Torus
- Inversion Tube
- Cutting or Slitting
- Tube and Die
- Rolling/Flattening a Tube
- Strap, Rod, or Wire Bender
- Wire-Through-Platen
- Deformable Links

The one that is used in this study is very similar to the "Crushable Column" Method. The load-deflection characteristics were produced by crushing the column. The device was first used in landing struts for the Sikorsky S-58, S-61, and the S-62 helicopters. Later it was adapted as the energy absorber for the pilot seat. These seats were used on the early models of the Bell 222 Light Twin Helicopter [Desjardins, 2003].

In this study, the crushable absorber system is defined as shown in Figure 6 and Figure 7. In this system, there is a little sled on the lower part of the seat leg, which is a path for the upper part of the system. In the crash moment, upper part accelerates and slides down into the sled. With its velocity, lower part is deformed and crushed. By this way, the crash energy is absorbed.

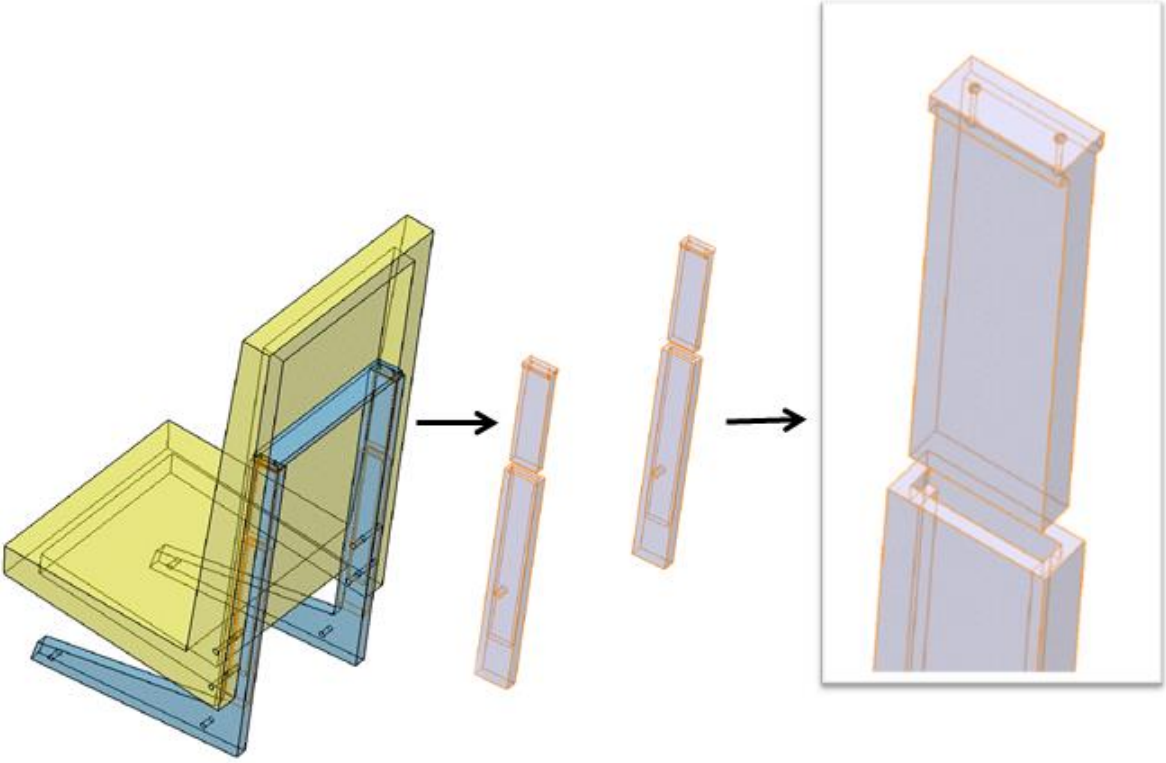


Figure 6: Energy Absorption Concept of the Model-2 Seat

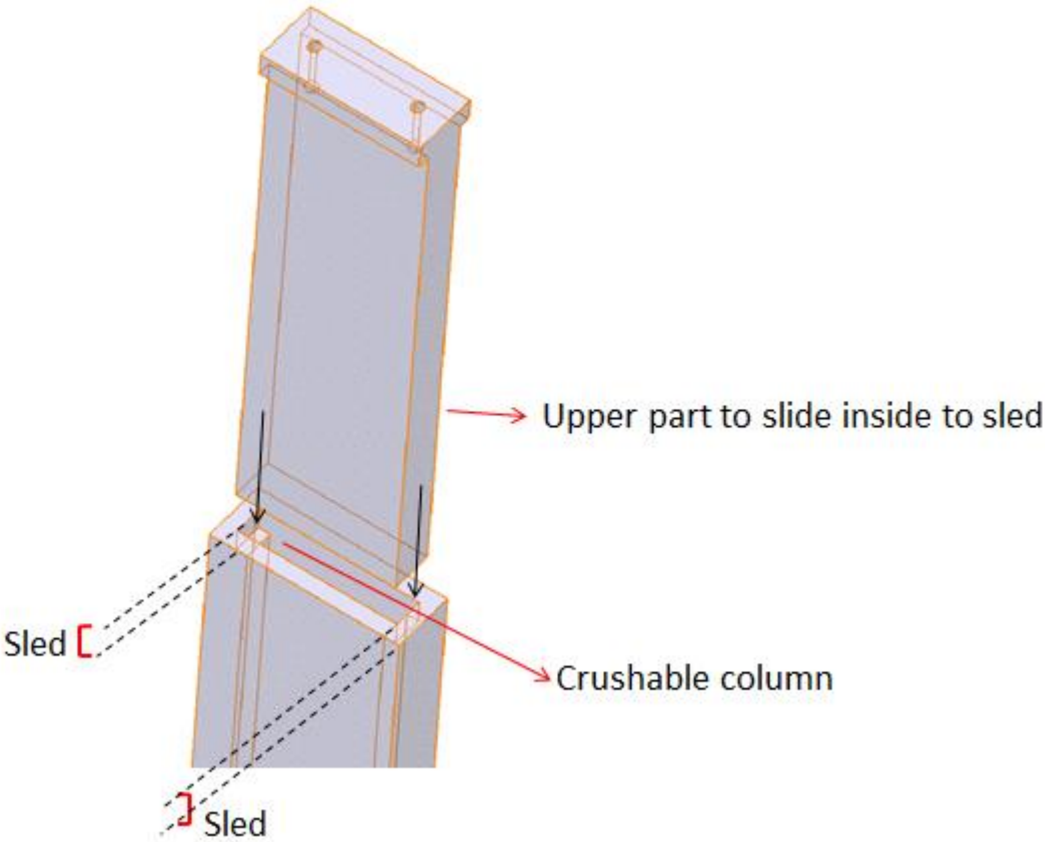


Figure 7: Energy Absorption Concept Detail

Finite Element Method

Finite element model for the test is generated such that it simulates the crash test as close as possible. For this specific crash simulation analysis ABAQUS Explicit software is used for both seat model study case.

ABAQUS is a general-purpose finite element program capable of simulating complex problems. The code's origins lie in highly nonlinear, transient dynamic finite element analysis using explicit time integration.

"Nonlinear" means at least one of the following complications:

- Changing boundary conditions (such as contact between parts that changes over time)
- Large deformations (for example the crumpling of sheet metal parts)
- Nonlinear materials that do not exhibit ideally elastic behavior (for example thermoplastic polymers)

The ability of ABAQUs/Explicit to effectively handle severely nonlinear behavior such as contact makes it very attractive for the simulation. This finite element analysis product is generally used to simulate brief transient dynamic events such as ballistic impact or automotive crashworthiness.

Model Definition

While preparing the ABAQUS analysis, CATIA V5 software is used for seat modelling and geometry is implemented in to the model as step file. While importing the geometries all seat related bodies are imported as 3D deformable solid bodies. The measure units of the model are given in Table 1.

Table 1: Units of measure

Measure	Unit
Length	mm
Time	second
Velocity	mm/s
Acceleration	mm/s^2
Force	Newton
Weight	ton
Energy	N.mm

The finite element modeling procedure starts with a generation of cad model as mentioned in previous chapter. After that step, the one should clean up the geometry and prepare for meshing. Cleaning the geometry means that the complex geometries and sharp edges are removed from the model in order to get simple mesh geometry. Simple mesh geometry is important to reduce the analysis time.

The next step is determining the connections of the seat parts. It is important for the meshing procedure because around the hole locations dense meshing is applied.

After determining the connections, FE mesh is created and model quality is checked in terms of askewness, warping, mesh size and aspect ratio. Aspect ratio of the mesh should be less than 3:1 in order to avoid the instability of stress wave.

For the mesh of the both seat models, tetrahedron (TET10 with 10 nodal points) mesh is used. The mesh is adjusted for explicit analysis. For all the parts minimum 5 mm of mesh seed is used. This is due to nature of the explicit analysis and the length of the computation. The generated mesh can be seen in Figure 8.

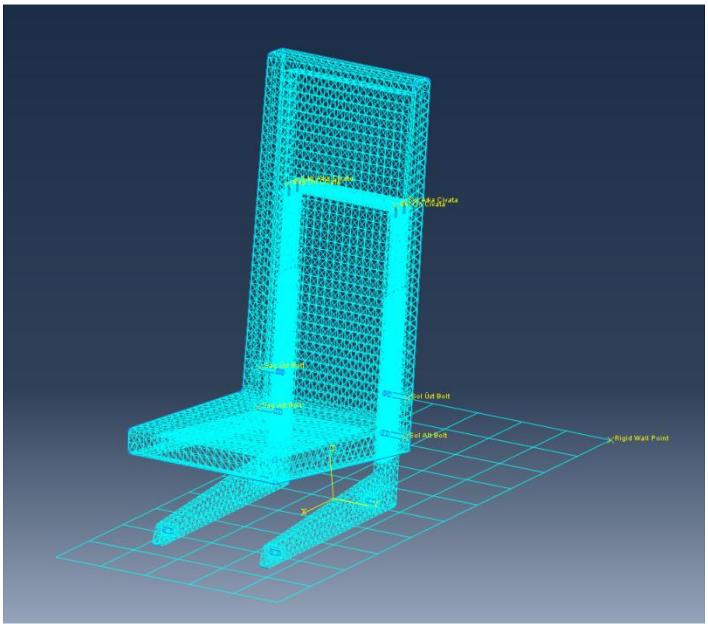


Figure 8: Meshed Seat Geometry

At the end of the mesh creation, materials are assigned using the section assignment. After generating the mesh, a reference point (RP) is created for the rigid wall which is later used to fix the wall and create a boundary condition for the seat to be mounted to wall from the legs.

After the part meshing is completed, the seat assembly is created in ABAQUS. Since the step file is used and imported directly to ABAQUS from CATIA assembly, all parts are assembled in correct places as in the CATIA step file. Assembly procedure also contains the Reference Point, RP, assignments.

For all the holes and bolted joints, shown in Figure 9, a RP is created to be used in force transition points between different parts. These RP's are used in constraint definitions and all constraints are defined via the coupling method. Coupling method is used to bind the holes to each other between a reference point (a nodal set) and geometric or nodal sets. Kinematic coupling is used in order to transfer both translational and rotational degrees of freedom which is the case in bolted joints and generally used in force transition points.

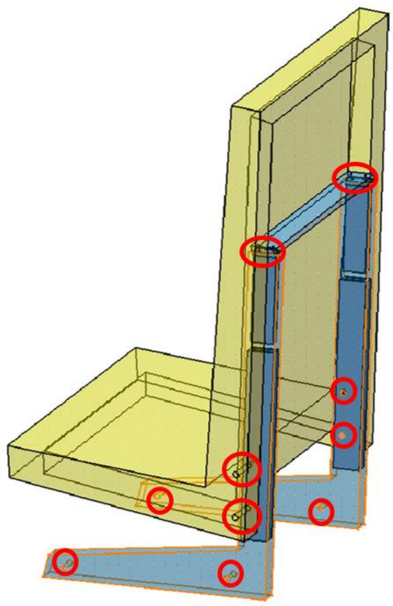


Figure 9: Seat joint locations

Since this model tries to simulate the material deformation between parts of the seat; an interaction should be generated in order to define the contact points. As it can be seen in most of the crash analysis, this is done by using the general explicit interaction command in ABAQUS. This generic command creates an interaction surface relation when two or more mesh surface gets into contact in a specific time step. For the interaction properties, tangential penalty and normal behavior is used to simulate the material flow for plastic deformation and normal load transfer for frictional contact.

The material used for the seat leg structure is aluminum. Material Al 2024 with elastic and plastic behavior is used for seat leg structure and the true stress-strain curve of Al 2024 given in Figure 10 is used for the material model. In addition to stress-strain behavior, frictional coefficient is also added as an analysis parameter, which is 0.3 for Al 2024.

Figure 10: AL2024 Stress-Strain Curve

To simulate the seat bucket, on the other hand, a user defined material is defined with the same elastic behavior as Al 2024 but with different density to adjust the total weight of the seat according to the real rotorcraft seat. The total weight of the seat system is 17.8 kg in which 13.8 kg is for the seat bucket and 4 kg is for the seat legs.

To simulate the crash event, 2D discrete rigid planar shell geometry is used to represent the rigid wall.

ABAQUS Explicit solver needs time step definition and frequency for output collection from the model. History output is used to collect energy related values from the system; whereas field output is used to collect force, acceleration, velocity and strain related values. To specify these values before completing the explicit model, simple frequency analysis is done to find the natural frequency of the interested deformation shape. Time step value defined in explicit step should be at least the inverse of the natural frequency of the desired mode shape of deformation.

Dynamic model contains a few very small elements that will force ABAQUS to integrate the entire model. These small elements may be the result of a mesh generation task and may increase analysis time significantly, although the effect on the overall dynamic behavior of the model may be negligible. For that reason, mass scaling is used to speed up the analysis time. By scaling the masses of these controlling elements at the beginning of the step, the stable time increment can be adjusted. The criteria while creating the mass scaling is the condition that the stable time step should be at least 10^{-7} seconds.

If this criteria is not satisfied, model automatically scales the system in order to have a stable time period.

Boundary Conditions

The boundary condition is used to define the fixed non-deformable rigid wall. While modelling the wall, a single RP is placed to a corner of the surface in order to define this BC as shown in Figure 11. By using the ENCESTRE command in ABAQUS BC choices; it is specified that this point cannot translate or rotate.

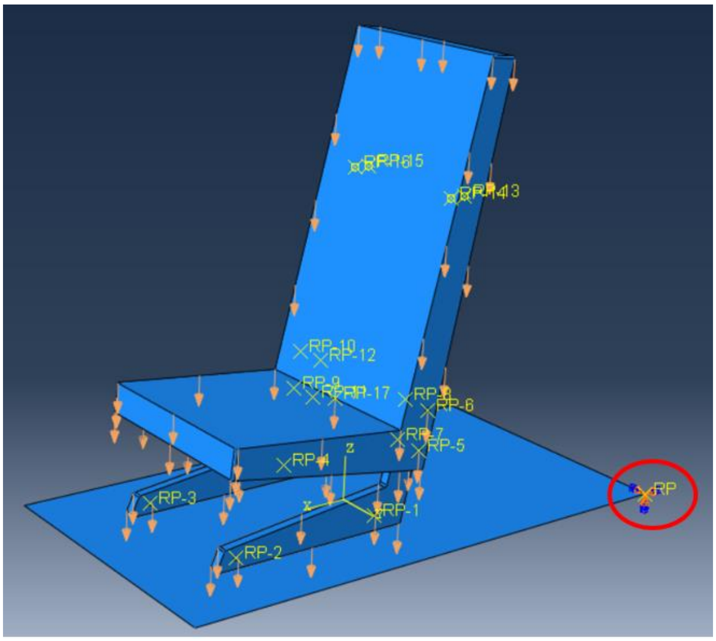


Figure 11: Rigid wall boundary condition point

The predefined field section is used in order to define the impact velocity of the seat. In this model, since the seat is modelled as it is coupled to rigid wall from the 4 holes, which can be seen on Figure 9, the model simulates the impact with seat having initial crash velocity 9100 mm/s, which is defined in CS29.562 EASA requirement, at time zero.

In the analysis model, gravity force is defined also, which will contribute to the acceleration of the seat.

Finally, in order to simulate the passenger weight, which is given as 77kg by EASA, weight is defined on the seat. This weight is given to a single RP, shown in Figure 13, as a concentrated force, which is the load case for the model. This RP is also coupled to the seat surface where the passenger is placed. The force given to the RP is 755.37 N (0.077ton x 9810mm/s^2) in -Z direction.

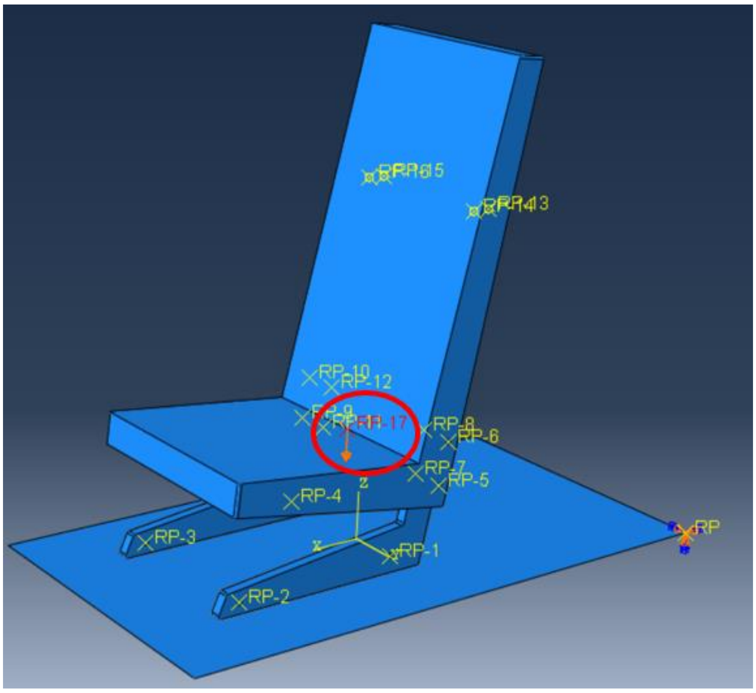


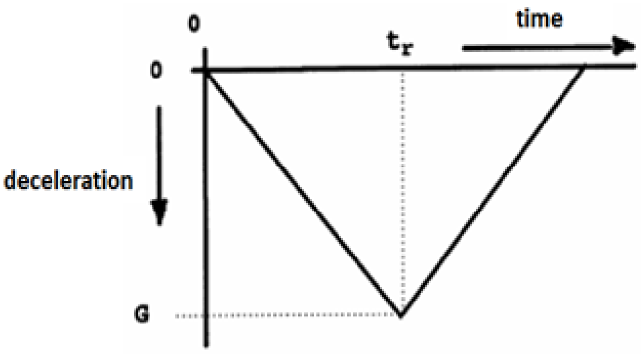
Figure 12: Seat load application point

After these steps, the model is complete and job is created to submit for analysis run. While submitting the model, 8 cores solving method is used for faster computing.

It is important to see the complete behavior of the seat in the analysis. To specify the analysis run time, simple frequency analysis is done to find the natural frequency of the interested deformation shape. Then, analysis run time is given as 0.05 seconds for this study.

RESULTS

Acceleration pulse behavior of the helicopter seats that are used in the field are expected to fit the graph given for the downward crash in CS 29.562 that defines the pass/fail criterion [EASA, 2012] as shown in Figure 13. In the table of Figure 13, V is impact velocity, G is deceleration measured on the sled and tr stands for the rise time. For this study, g load is measured on the seat bucket where the passenger sits in order to see how the most critical part, pelvis part, of the passenger is affected.



	Test 1	*Test 2*
Min. V (m/s)	*9.14*	*12.80*
Max . tr (s)	*0.031*	*0.071*
Min.G	*30*	*18.4*

Figure 13: Test pulse simulating rotorcraft floor deceleration-time history [SAE AS8049, 2005]

In this study two seat models are used. Model-1 is the seat structure in which the damping mechanism is removed and only seat bucket and legs are included. Model-2, on the other hand, includes the seat bucket, legs, lower damping part including the sled and upper damping part that moves into the sled and deforms the column inside the lower part, as shown in Figure 5.

This study aims to compare the acceleration pulse behavior of the two seat models and observe the difference between the peak accelerations.

In the crash analysis, proper contact regions are defined for the damping mechanism parts and elastic and plastic material model is applied to Al 2024.

FE seat models are exposed to dynamic crash testing loads by giving 9100 mm/s crash velocity to the seat at time zero as shown in Figure 14 and hit the fixed rigid wall.

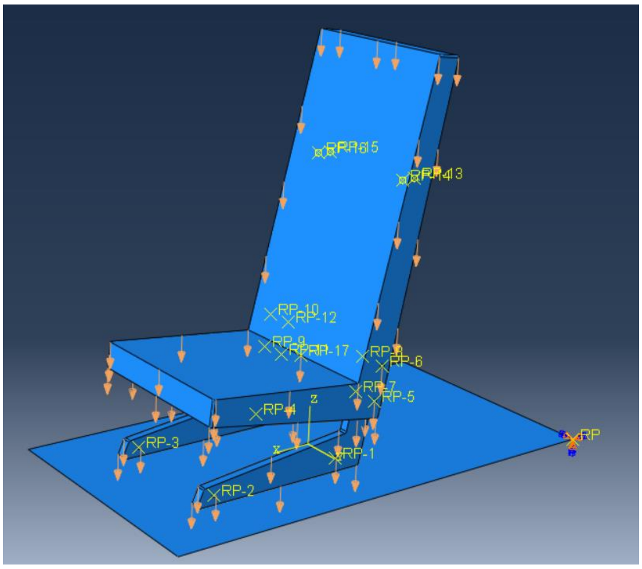


Figure 14: Seat at t=0 moment with initial velocity

Field output is used to collect acceleration, displacement and velocity values. The node location where displacement field output is tracked is the location shown in Figure 15. Figure 15 and Figure 16 shows the displacement levels of the Model-2 seat, which has damping mechanism.

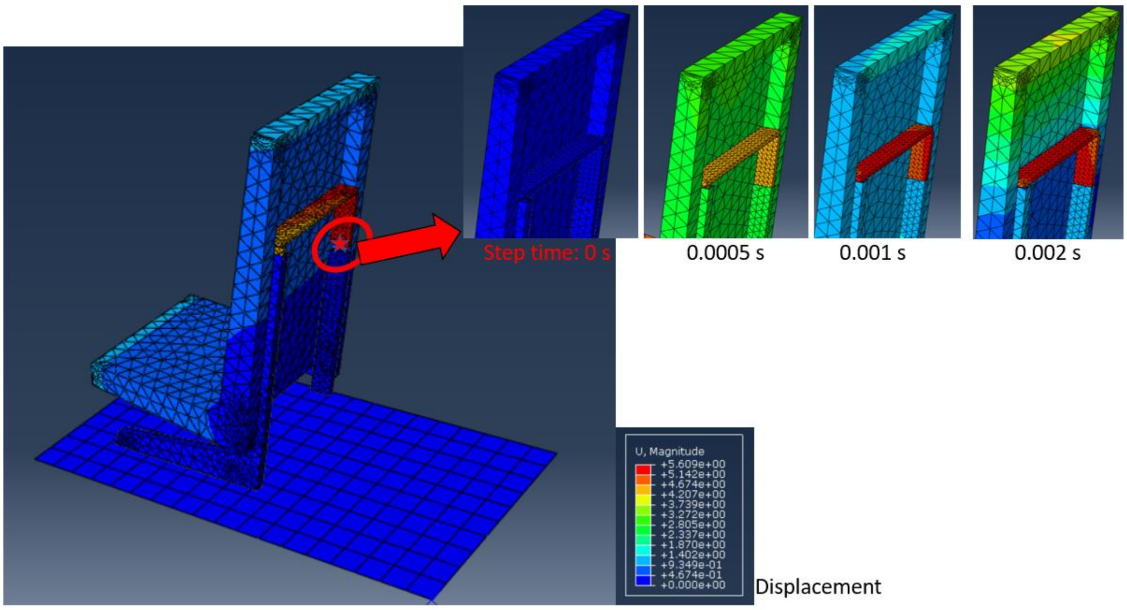


Figure 15: Model-2 seat leg upper part displacement

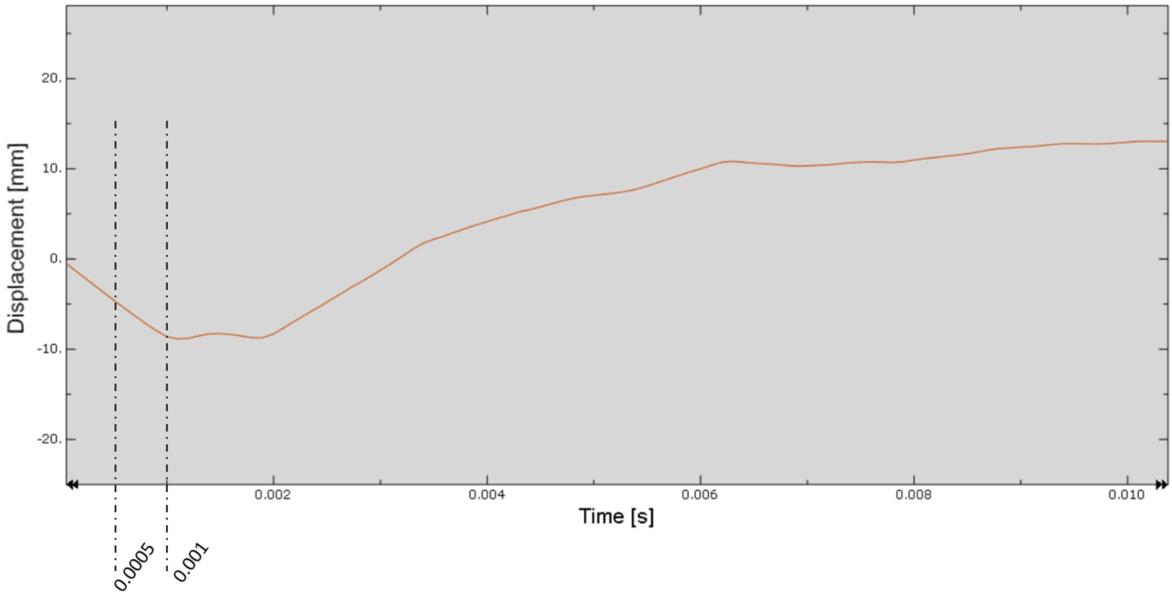


Figure 16: Model-2 seat leg upper part displacement graph

IAs shown in Figure 16, upper part displacement is nearly 10 mm. t is observed that the upper part moves towards the sleds on the lower part and crushes it. Then it goes back and moves upward as a reaction.

Field output node location to collect acceleration and velocity values is shown in Figure 17. Selected point is the point where the passenger pelvis is in contact on the seat.

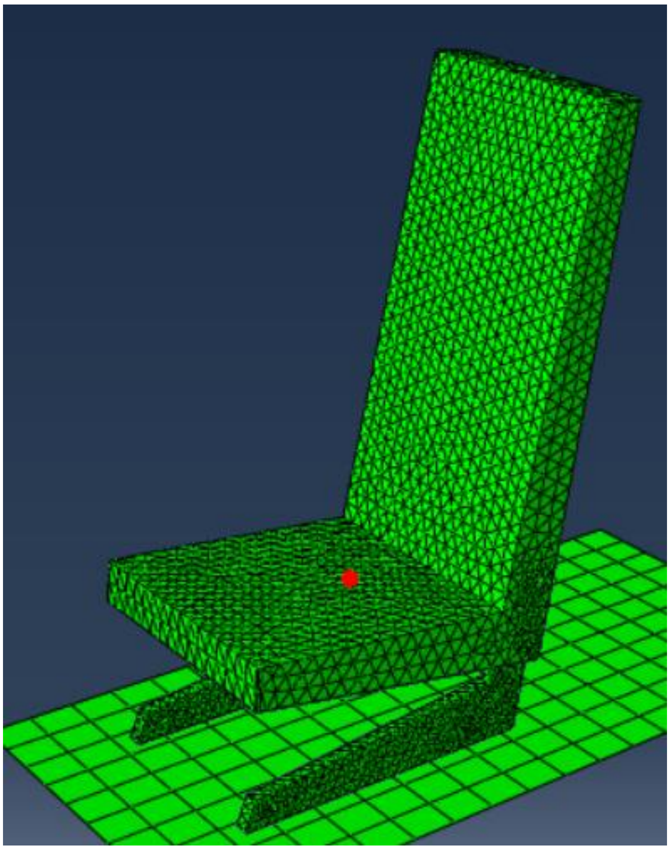


Figure 17: Seat field output location

Figure 18, Figure 19 and Figure 20 show the acceleration,velocity and kinetic enrgy outputs of the model-1 seat analysis.

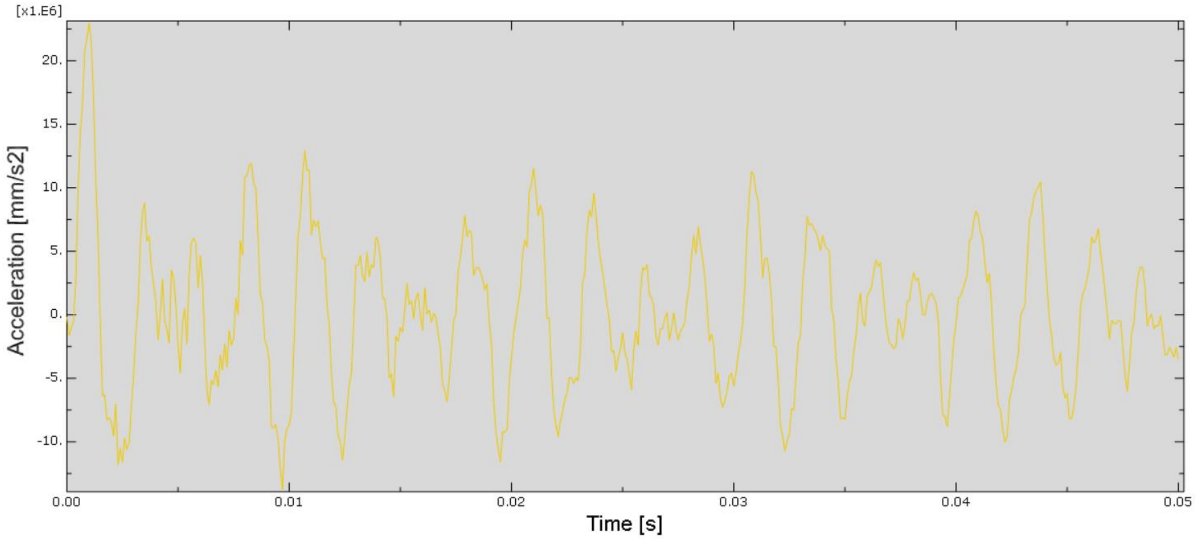


Figure 18: Acceleration output from the model-1 seat

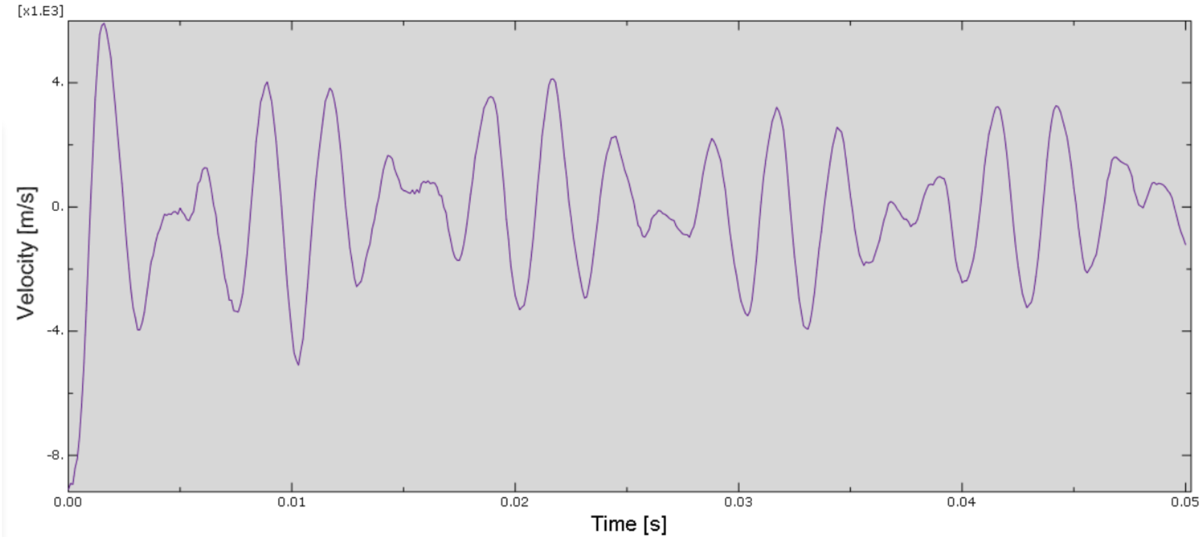


Figure 19: Velocity output from the model-1 seat

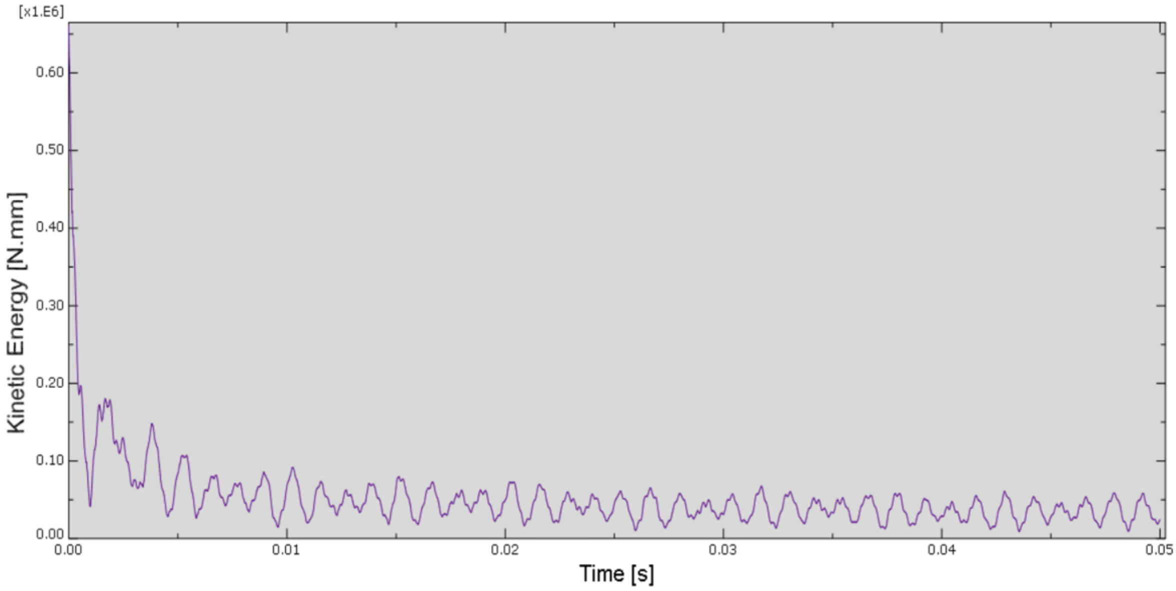


Figure 20: Kinetic energy output from the model-1 seat

Figure 21, Figure 22 and Figure 23 show the acceleration, velocity and kinetic energy outputs of the model-2 seat analysis.

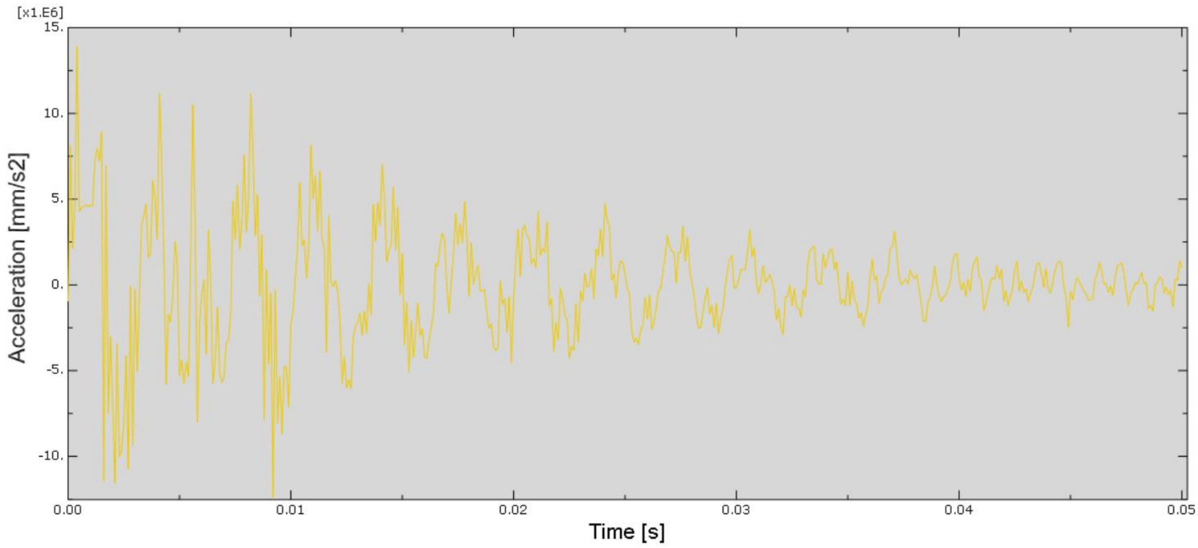


Figure 21: Acceleration output from the model-2 seat

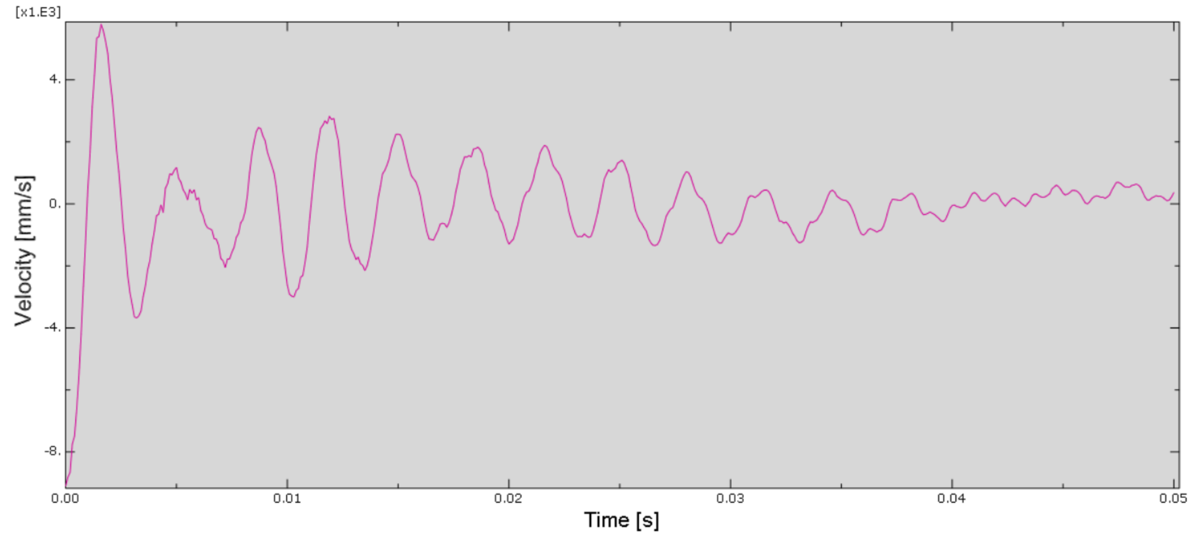


Figure 22: Velocity output from the model-2 seat

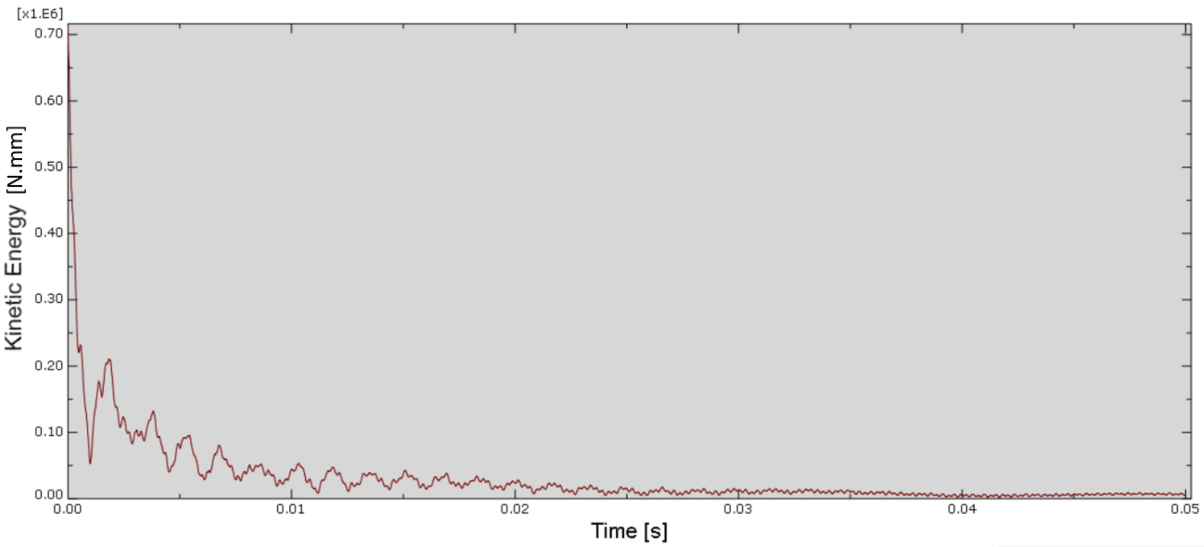


Figure 23: Kinetic energy output from the model-2 seat

After the crash analysis of model-1 seat, maximum acceleration is calculated as 2290 g, as seen in Figure 19. Model-2 seat analysis on the other hand, gives the maximum acceleration as 1275 g, as seen in Figure 22. Peak acceleration results obtained in the preliminary study turned out to be very high and as a continuation of the present study, a thorough check of the model will be made. However, velocity responses of the both models are very reasonable, since the initial impact velocity is read as -9100 mm/s, and after the impact velocity of the output node given in Figure 18 oscillates about zero as expected.

CONCLUSION

In this study, effectiveness of the energy absorbing system based on the crushable column concept is investigated for a rotorcraft seat which undergoes an impact event.

It is shown that by using a material model which takes the plastic deformation of the material into account, “crushable column” concept is implemented successfully and % 44 decrease in the g level is achieved. Although the peak acceleration levels are very high and need to be checked, the model generated for the crushable column concept works and accounts for significant reduction the g level transferred to the occupant.

Inputs and the output graphs of the seat models that with and without the energy attenuation system show that the damping mechanism in seat structure can absorb some of the crash energy and decrease the load that is transferred to the passenger.

The main purpose of this study is to assess the behavior of the seat structure and the occupant safety with a crashworthy seat via FE analyses. The study demonstrates that the energy absorption mechanisms on rotorcraft seats are the key for the -survivability of the occupant.

References

ABAQUS, ABAQUS Inc. (1978), https://www.simulia.com.

Bhonge, P. (2008) *A Methodology for Aircraft Seat Certification by Dynamic Finite Element Analysis,* Doctoral dissertation, Wichita State University, USA, Dec 2008.

Desjardins S. P. (2003), *The Evolution Of Energy Absorption Systems For Crashworthy Helicopter Seats,* PRESIDENT SAFE, INC TEMPE, ARIZONA

Dhatt G, Touzot G. & Lefrançois E. (2012), *Finite Element Method,* London, UK: ISTE Ltd & Hoboken, USA: John Wiley & Sons, Inc.

DYTRAN, MacNeal-Schwendler Corporation (MSC) Software (2013), http://www.mscsoftware.com/.

European Aviation Safety Agency (2012) *Certification Specifications CS-29: Large Rotorcrafts*, December 2012.

Liang C., Ceccarelli M. & Carbone G. (2011) *Design and Simulation of Legged Walking Robots in MATLAB Environment,* LARM: Laboratory of Robotics and Mechatronics, University of Cassino, Italy, pp.477- 478.

LS-DYNA, Livermore Software Technology Corporation (2015), http://www.lstc.com.

PAMCRASH, ESI Group (1985), https://www.esi-group.com/.

Renner, M. (2013) *Global Air Transport Continues to Expand*, Worldwatch Institute (New York: W.W. Norton & Co.), December 2013.

SAE Aerospace (2005), *Performance Standard for Seats in Civil Rotorcraft, Transport Aircraft and General Aviation Aircraft, 2005.*

Simitcioglu G. & Dogan V. (2013), *Dynamic Finite Element Analysis of an Aircraft Seat,* Ankara International Aerospace Conference, Turkey, September 2013.

TNO-MADYMO, https://www.tassinternational.com/madymo.