Crashworthiness of Aircraft Seat

Martin Plhal, Jan Šplíchal

Faculty of Mechanical Engineering, Brno University of Technology, Institute of Aerospace Engineering
Technická 2 , Brno , 616 69, Czech Republic

Keywords: *Crashworthiness,Aircraft, Dummy,FEM*

Abstract

The dynamic analysis will be demonstrated on the example of a seat for a small 4-seater GA airplane. The airplane is designed according to JAR/FAR-23 regulations. The geometry was subsequently transferred to the MSC.PATRAN preprocessor. This allowed creation of the finite element (FEM) model of the seat.

Model of the seat was created using MSC.PATRAN (pre/postprocessor) and analysed using MSC. DYTRAN (solver). The rest of the seat structure was modeled using 1D elements (BAR2) 2D elements (QUAD4, TRIA3) and 3D elements (HEX8, WED6).

FEM model of the dummy (simulating a pilot) is integrated in MSC.PATRAN/DYTRAN package and the dummy is generated by GEBOD application. The dummy is composed of 2 main parts – ellipsoids and skin.First part is made from RIGID ellipsoids. Ellipsoids are connected together using linkages that simulate articulations. This part of the dummy is responsible for the behavior of the dummy. Second part is the dummy's skin, using 2D elements.

Materials in the seat construction were defined using DMATEP. This material model defines properties of isotropic elasto-plastic material for shell and beam elements. This model enables inclusion of the yield stress, changes in hardening modulus and failure.

Loads were applied in accordance with JAR/FAR 23.562 (b) (1) as the change in velocity in the attachment between the seat/restraint system and the test fixture. Velocities and accelerations in chosen points on the dummy were monitored during computation. Points were chosen where necessary with respect to monitoring of max. accelerations. Such points are in the CG of occupant's head, chest and pelvis. Results were compared with recommendations of regulations for max. accelerations imposed on the human body.

1 Geometrical model

The dynamic analysis will be demonstrated on the example of a seat for a small 4-seater GA airplane. The airplane is designed according to JAR/FAR-23 regulations (JAR regulation is currently in the process of transformation into CS requirements). The geometry was subsequently transferred (via parasolid) to the MSC. PATRAN preprocessor. This allowed creation of the finite element (FEM) model of the seat. The geometry of the seat is on the figure 1.

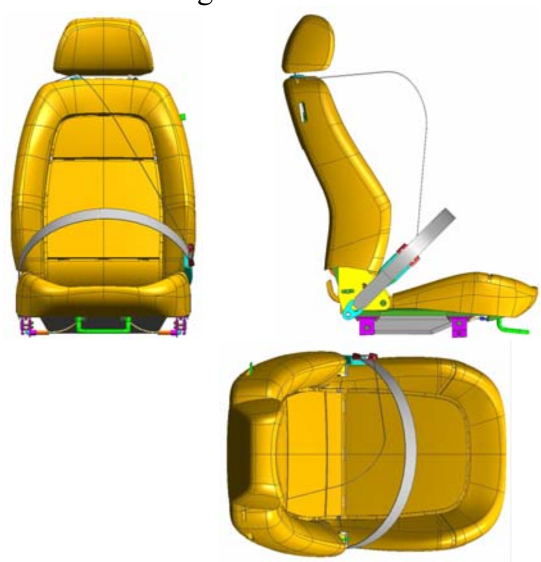

Fig. 1. Geometrical model

2 FEM model

2.1. Seat FEM

Model of the seat was created using MSC. PATRAN (pre/postprocessor) and analysed using MSC. DYTRAN (solver). The rest of the seat structure was modeled using 1D elements (BAR2), 2D elements (QUAD4, TRIA3) and 3D elements (HEX8, WEDGE6). FEM model of the seat has total of 36 561 nodes and 86 802 elements (including 703 1D elements, 20876 2D elements and 65223 3D elements).

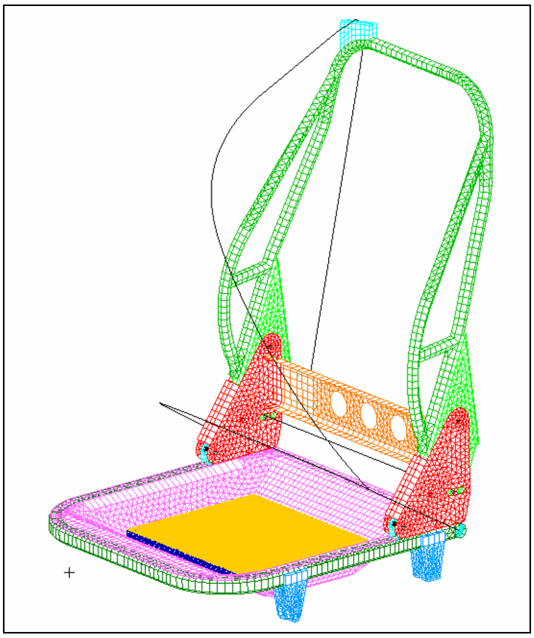

Fig. 2. FEM model the seat

2.2. Dummy

FEM model of the dummy (simulating a pilot) is integrated in MSC. PATRAN/DYTRAN package and the dummy is generated by GEBOD application. The dummy is composed of 2 main parts – ellipsoids and skin:

First part is made from 17 RIGID ellipsoids. Ellipsoids are connected together using linkages that simulate articulations. This part of the dummy is responsible for the behavior of the dummy.

Second part is the dummy's skin (using 2D elements).

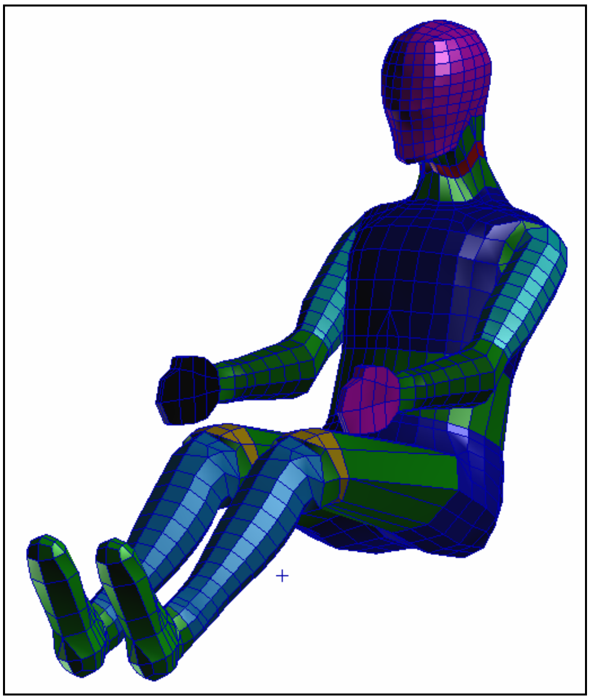

Fig.3. FEM model the dummy's skin

3 Material models

Materials in the seat construction were defined using DMATEP (this material model defines properties of isotropic elasto-plastic material for shell and beam elements). This model enables inclusion of the yield stress, changes in hardening modulus and failure.

4 Loads

Loads were applied in accordance with JAR/FAR 23.562 (b) (1) as the change in velocity in the attachment between the seat/restraint system and the test fixture (see fig. 5).“The change in velocity with the time” of the whole model is in the fig. 4 (seat with an occupant decelerated from this initial velocity).

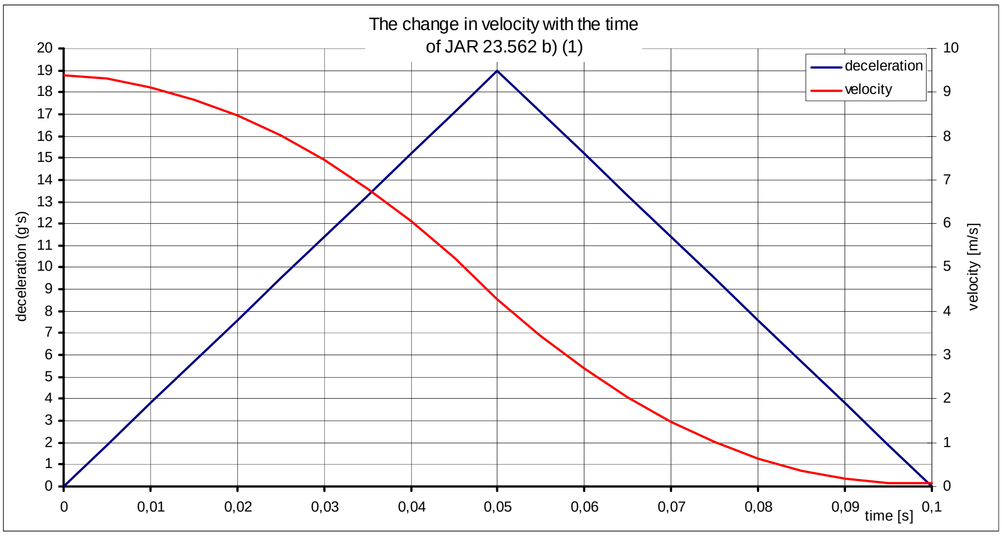

Fig. 4. Loads

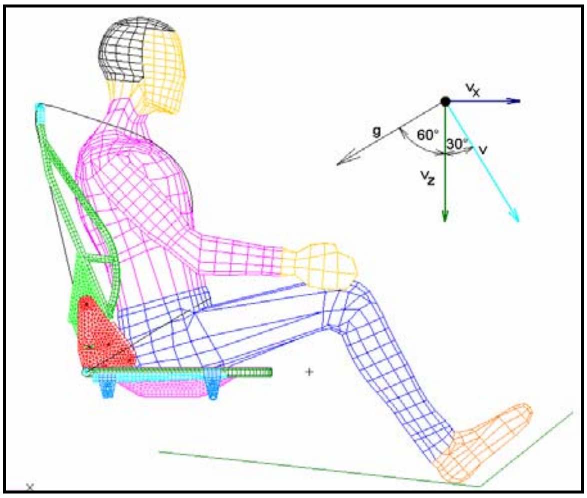

Fig. 5.

5 Computation and results

Initial parameters of the computation:

END TIME 0,15 time of computation
INISTEP,1e-7 initial time step
MINSTEP,1e-9 minimal time step

Velocities and accelerations in chosen points on the dummy were monitored during computation. Points were chosen where necessary with respect to monitoring of max. accelerations. Such points are in the CG of occupant's head, chest and pelvis. Results were compared with recommendations of the regulations for max. accelerations imposed on the human body

6 Conclusion

Requirements of the regulation:FAR 23.562 (c) (7) The compression load measured between the pelvis and the lumbar spine of the ATD (anthropomorphic test dummy) may not exceed 680kg (1,500 pounds)

Load measured between the pelvis and the lumbar spine has max. value of 621 kg. This value is lower than max. allowable load specified in the requirement JAR 23.562 (c) (7), which permits 680 kg. This implies that, according to the results of simulation, pilot seat complies with requirements of regulation.2.4 Diagrams and Figures

References

[1] FAR Part 25: Airworthiness standards: Transport category airplanes. Federal Aviation Administration, Washington, D.C. 7/2002

[2] MSC.DYTRAN, Users manual, Version2004, U.S.A., 2004

[3] Sedláček R. ,Šperka P., Využití systému MSC.DYTRAN a MSC.NASTRAN v oblasti výpočtů leteckých konstrukcí,2001

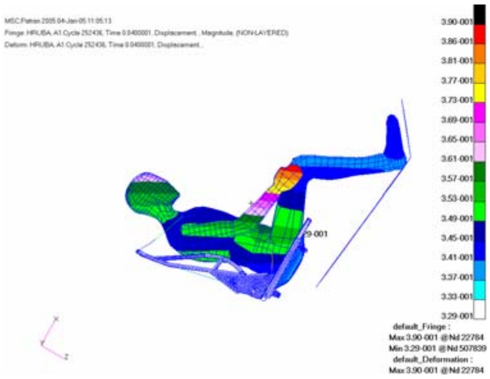

Fig. 6. – time 0.04[s]

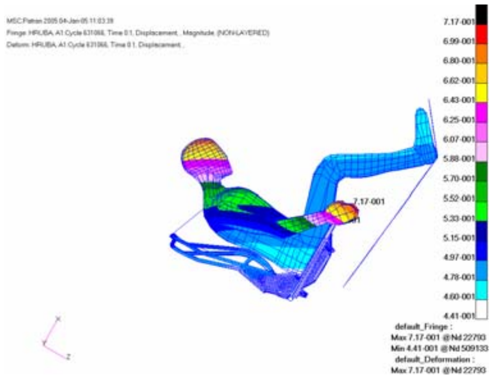

Fig. 7. – time 0.1[s]

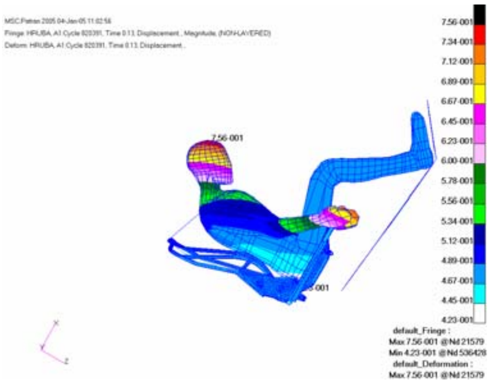

Fig. 8. – time 0.13[s]

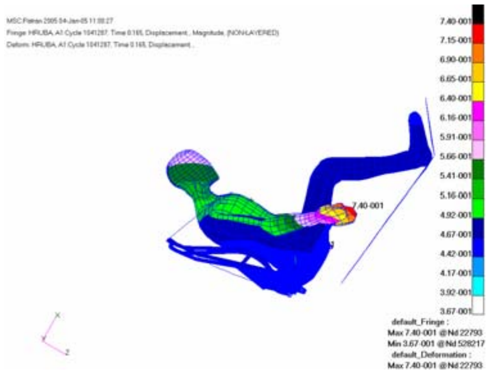

Fig. 9. – time 0.165[s]