Using a Drop Tower test to Dynamically Validate an ABAQUS model of an Automotive Seat for Side Impact Crash Simulation

H.Hartmann[1], M. Socko[2], and Dr. R. Hanley[3]

[1] Faurecia Autositze GmbH, [2] Faurecia Fotele Samochodowe Sp. z o.o, [3] BMW AG

Abstract: In contrast to frontal crash, the occupant sits directly in the deformation zone of the vehicle structure during a side impact event. The goal in occupant protection is not one of restraining the occupant relative to the interior of the vehicle but one of protecting the occupant from the high energies of the intruding surface.

Apart from the seat's main role of connecting the occupant to the vehicle during normal driving, it also plays a large role during a side impact event. Here, the lateral movement of the occupant relative to the seat can strongly influence the energy levels experienced by the occupant. Thus, the local deformation of the seat structure is of great interest in the analysis of such a side impact event.

The importance of the seat in the side impact event has become increasingly more significant in recent years with the advent of seat integrated side protection airbag systems, lateral comfort systems and safety belt systems. The seat now performs a multitude of functions in the vehicle, all of which add to its mass, its stiffness and the complexity of its construction.

This paper describes a study undertaken to establish a means of dynamically validating a typical finite element seat model used in the virtual development of side impact protection systems at BMW.

The investigation which led to the definition of the validation test is described, as are the test itself and the ABAQUS model. Finally, the correlation of the model to the hardware test is analyzed and the measures taken to improve the correlation discussed.

Keywords: Automotive, Crashworthiness, Impact, Safety, Seat.

1. Introduction

Experience shows that although the global behavior of finite element (FE) seat models corresponds relatively well to that of the hardware seat seen in crash tests, there is still a degree of uncertainty as to the local deformation behavior of the metal seat structure. This may influence the interaction between the occupant and the seat during a side impact crash event. Due to the practical limitations of monitoring the local behavior of the seat during a full vehicle crash test and indeed the expense involved in such tests, it is not feasible to validate the seat model by means of full vehicle test alone. For the validation of the FE seat model repeatable component tests are needed to allow the detailed analysis of the behavior of the seat hardware in isolation and at acceptable costs.

1.1 Foregoing Investigation

Previous efforts to validate FE seat models have been restricted to quasi-static loading of the seat back in the lateral (Y) direction (representing the loading of the seat by the intruding vehicle structure). In such tests the lower part of the seat is also supported on the opposite/unstruck side with a rigid plate (representing the center console or transmission tunnel). Although a helpful first approximation for the stiffness of the seat model, these tests are not sufficient to validate the model for such dynamic events as a side impact crash.

In order to define the boundary conditions for a dynamic component test it was first necessary to analyze the dynamic deformation experienced by seats in vehicles within the BMW product range during the various legislative and consumer group crash configurations. This was done by taking the respective FE simulations of the crash events and calculating the relative lateral (Y) deformation and deformation rate between nominal points on the seat back frame (Upper) and the seat rails (Lower). Figure 1 illustrates the calculated seat deformation characteristics for a range of vehicles in the IIHS test configuration, a US Consumer group test, which is known to result in the highest vehicle intrusion characteristics.

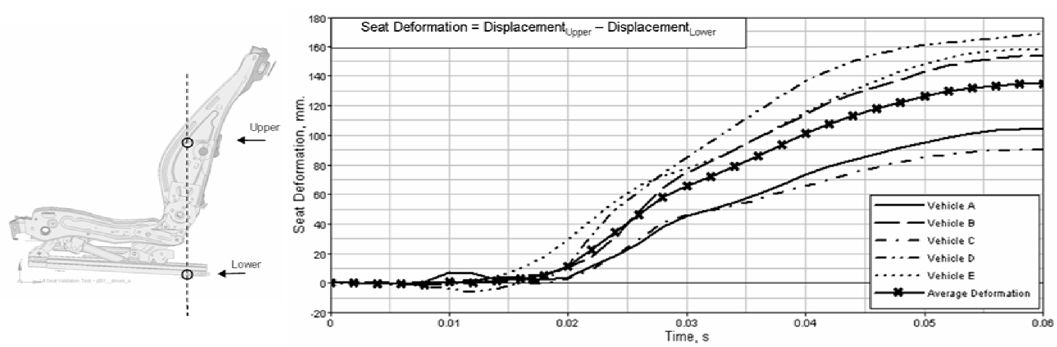


Figure 1. Seat lateral (Y) deformation characteristics from a range of vehicles.

These characteristics are seen to vary from vehicle to vehicle and load case to load case, which is to be expected given the unique boundary conditions within each configuration. It is however reasonable to deduce that the average of these seat deformation curves gives a good indication of the degree and rate of deformation to be expected in a severe crash event and thus is a good basis upon which to specify a component test.

1.2 Selection of Seat to be Investigated

For the purposes of this investigation it was decided to focus on one basis seat design. The seat, from the current X5, was selected for the two main reasons:

- The seat had recently gone into series production and is therefore available at acceptable costs while also being at the leading edge in terms of the design and function.
- The seat is the basis for the next generation mid to large size vehicles from BMW and these vehicles are the first which will be developed using ABAQUS alone for crashworthiness analysis.

1.3 Definition of Component Test

Having identified the approximate degree and rate of deformation to be exerted on the seat, and selected the seat to be investigated, the next task was to define a representative test configuration.

The load exerted on the seat during the crash event is the result of coupled deformation of the surrounding vehicle structure and the barrier; the seat has very little, if any, reverse influence on this deformation of the vehicle or barrier. In simulation it is relatively simple to recreate this loading by means of a rigid impactor plate, driven by displacement function with the average curve described in Section 1.1 as its input and a supporting plate on the unstruck side to represent the transmission tunnel and centre console. This simulation is schematically described in Figure 2. In this ideal configuration, the seat has absolutely no influence on the motion of the impactor.

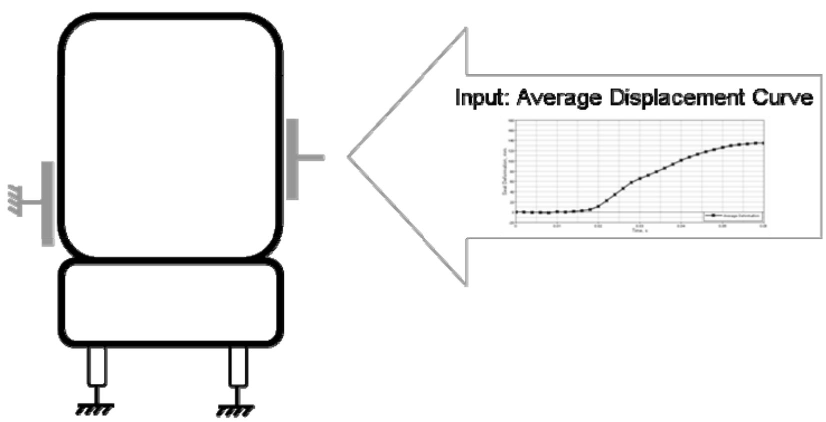


Figure 2. Schematic of driven displacement simulation.

This configuration is however not ideal for the purposes of validation as it requires the use of a high powered hydraulic ram or something similar where even though the motion of the impactor is constrained by the ram, there is always some degree of reverse coupling between the seat and the impactor motion. This constrained motion with coupling is very difficult to capture in simulation.

A more straight forward method is the use of a drop tower or pendulum whereby the seat is impacted by a falling mass. This method differs from the actual loading in the vehicle and in the driven impactor simulation in that the motion of the impactor is directly influenced by the seat and the energy/velocity of the impactor will be diminishing from the first instant of contact on. This difference is however acceptable as with suitably selected impactor mass and velocity, similar degrees and rates of deformation are achievable.

A simulation based investigation was carried out in order to establish the desired mass and velocity of the impactor. It was found that a mass of 100kg with an impact velocity of 7m/s delivers similar deformations and strain distributions to the driven impactor simulation, Figure 3.

In Figure 4 the motions and energies for the two simulations are compared. By applying a 20ms time shift to the drop tower curves, one can directly compare the deformation and work rates. In the driven deformation simulation the impactor has no initial velocity or kinetic energy where as the impactor in the drop tower does. Nevertheless, it can be clearly seen that the peak displacements and velocities correspond well between the driven deformation and drop tower simulations, as does the internal energy (work done) in the seat.

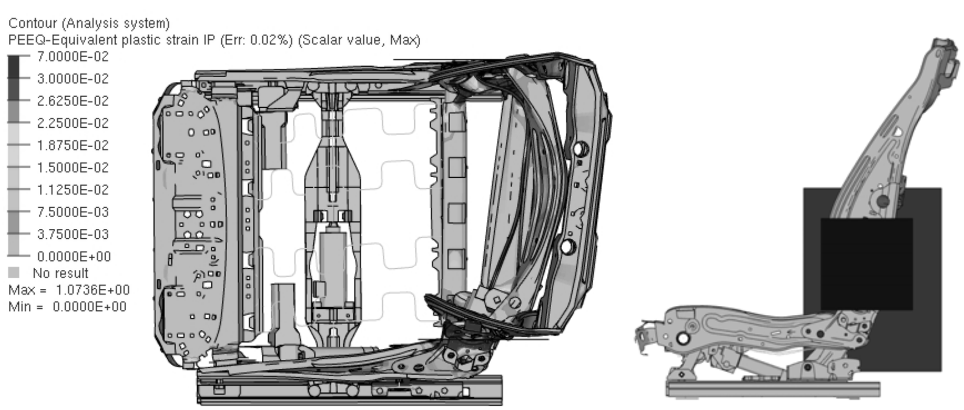


Figure 3. Deformation behavior, impactor mass 100 kg, velocity 7m/s.

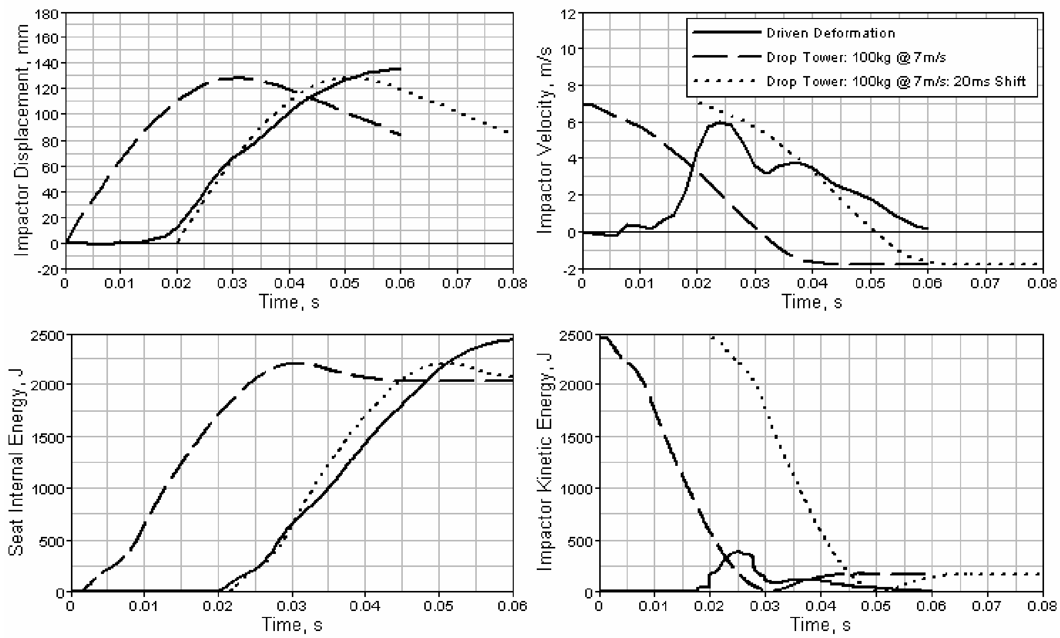


Figure 4. Comparison of driven displacement and drop tower simulations.

On the basis of this simulation based investigation, it was decided that drop tower type component test with the appropriate mass and impact velocities are acceptable for the dynamic validation of FE seat models.

As a partner in the development of seats for BMW, Faurecia were able to offer use of their in-house drop tower facilities to conduct such tests. This drop tower will be described in detail in the Section 3.

2. Introduction to the Complete Seat and its Components

In order to satisfy the high demands placed on the seat system in the full vehicle, the finite element method is today a key tool very early in the seat development process. Seat suppliers and vehicle manufacturers work hand in hand to optimize the seat design for functionality, stability and weight.

As suppliers of complete seat systems, Faurecia are increasingly responsible for the preparation and delivery of validated complete seat models in defined stages to support this process. The complete seat structure is illustrated in Figure 5 and is comprised of metal, plastic and soft foam parts.

Occupant protection systems such as thorax airbag systems are now commonly mounted directly on the side member of backrest and under the seat foam/trim. As a system supplier Faurecia may

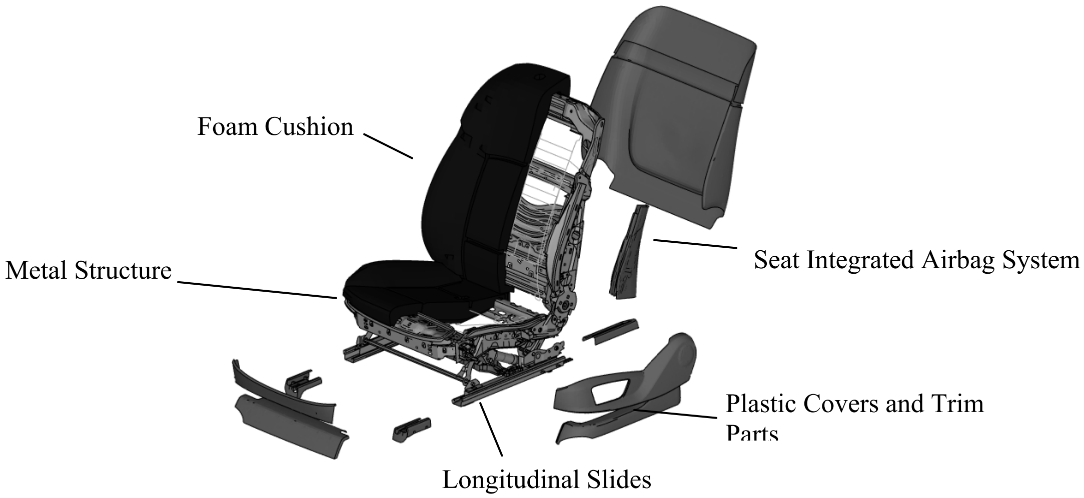


Figure 5. Complete seat finite element model.

also have the responsibility to incorporate these into the FE seat model and to validate the complete system with airbag deployment tests.

As mentioned in Section 1.1, the lateral stiffness of metal structure has previously been validated with quasi-static compression tests, however, dynamic validation tests are now considered necessary. Similar validation procedures are currently in development to validate the seat models for frontal crash purposes.

For the purposes of this investigation only the metal seat structure will we tested and simulated. The models of the foam and plastic parts are generally validated in isolated component and material tests and would only serve to bring uncertainty into this investigation.

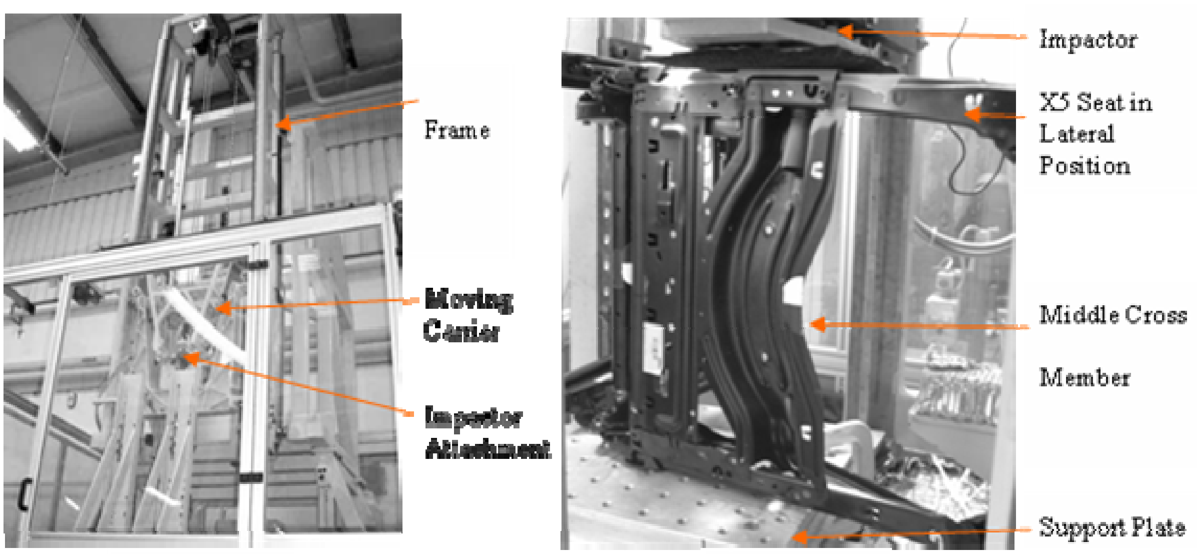
3. Description of the Drop Tower

The drop tower and additional test equipment used are located at the Faurecia validation facility in Brières, France. On the drop tower rig, the side impact event is reproduced by the free fall of a rigid impactor (fixed on a moving carrier) under gravity, Figure 6.

The standard attachment fixtures on the seat rails are mounted to specially prepared mounting blocks on the drop tower frame. A digital measuring arm is used to confirm the position of the seat in and relative to the drop tower rig.

In addition to the static deformation of structure after test the following measurements are made during the impactor fall:

- Displacement, speed and acceleration of impactor.
- Digital optical tracking of displacement of targets fixed on the structure, Figure 7.



(a) (b)

Figure 6: a) Drop tower and b) Seat mounted laterally in the drop tower.

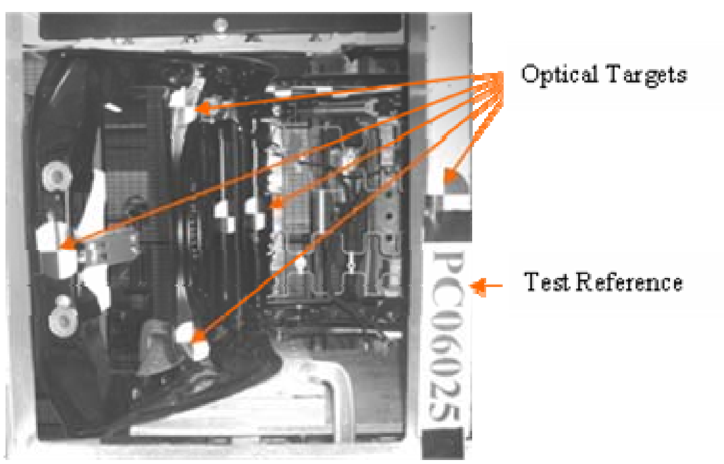


Figure 7. Optical targets and test reference.

The physical operating limits of the drop tower, i.e. the drop height, impactor mass are somewhat curtailed by safety ratings, these are detailed in Table 1.

In order to minimize the setup time and risk of damage to the test rig, the boundary conditions (mass, velocity and position of impactor) for the real drop test were defined by means of pre-simulation with ABAQUS Explicit.

Max. Dimension of Impactor	400 mm* 400 mm* 500 mm
Max. Drop Height	2.5 m
Max. Impact Speed	7 ms-1
Max. Carrier Mass	70 kg
Max. Impact Energy	1225 J

Table 1. Safety rated limits of the drop tower.

3.1 Impactor Mass and Velocity

As described in Section 1.3, an impactor mass of 100kg with an impact velocity of 7m/s (i.e. impactor energy of 2450 J) delivers comparable deformation levels to the driven impact configuration, Figure 4. However, as the safety rated impact energy of the drop test tower (1225J) does not permit such energy levels, analysis was conducted to determine whether lower energy levels would also deliver the desired degree of deformation.

As detailed in Table 2, a combination of the maximum mass (69 kg) and velocity (7 m/s) velocity leads to an impact energy of 1690 J, again significantly higher that the rated energy. However, a combination with an impactor mass of 69 kg and a velocity of 6.1 m/s results in an energy of 1284 J which, although significantly lower than the desired energy, is nevertheless acceptable as the dynamic nature of the test is similar with an impact velocity of only 1m/s less.

The main assembly of the seat backrest consists of two side members and three cross members (lower, middle, upper). In the results of the foregoing investigation, Figure 3, a significant plastic [permanent] deformation is seen in the center of the middle cross member. In the safety limited configuration, illustrated in Figure 8, where the impact energy is considerably less, the middle cross member is seen not to deform to the same extent.

Test Configuration	Impactor Mass [kg]	Impact Velocity [m/s]	Impact Energy [J]
Ideal	100	7,0	2450,00
Physical Limit	69	7,0	1690,50
Safety Limit	69	6,1	1283,75

Table 2. Drop tower impact energy

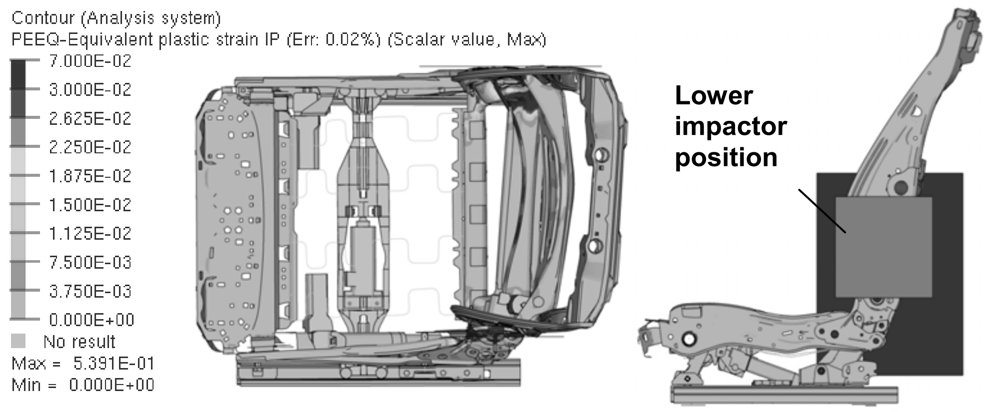


Figure 8. Deformation behavior, 69 kg impactor @ 6.1m/s, lower impactor position.

3.2 Impactor position

The position at which the impactor contacts the seat side member in both the foregoing investigation and these pre-simulations was chosen to approximate the position that the airbag and/or door trim pushes on the seat. Given the safety limits on the drop tower capabilities, it was decided to vary the configuration by moving the impactor upwards along the seat side member in order to reduce the loading of the very stiff lower cross member and increase that of the middle cross member, Figure 9. The deformation and stress distribution in the middle cross member are now very similar to those seen in the higher energy simulation.

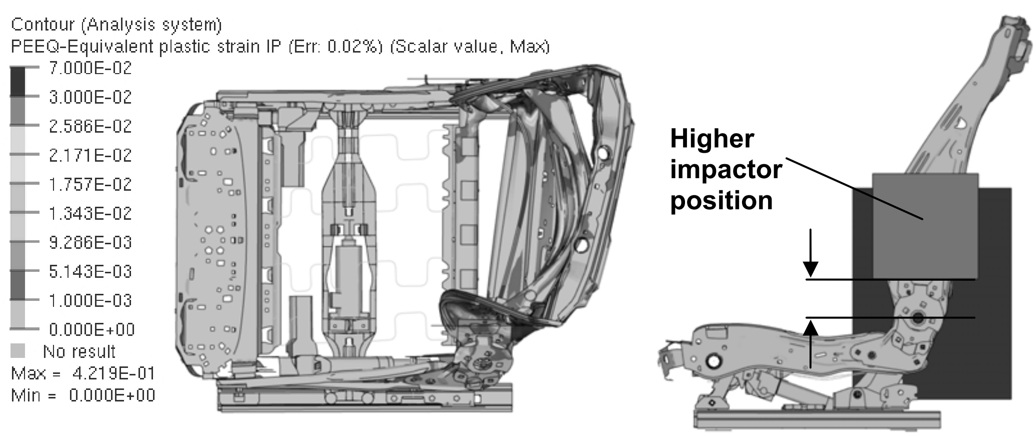


Figure 9. Deformation behavior, 69 kg impactor @ 6.1m/s, higher impactor position.

4. FE Abaqus Seat Model

The FE seat model was created for ABAQUS Explicit v6.6. The following simulations were performed with the v6.6-EF1 Release.

4.1 Discretization, Time Step & Mass Scaling

Faurecia experience has shown that the best ratio of accuracy of the results to the computational time for component tests is assured for time step of approximately 0.6 µs. However, due to the large model sizes involved, BMW uses an approximate time step of 1.25 µs in full vehicle crash simulations. The use of such a finely meshed component model in the full vehicle simulation is not desirable as it leads to unacceptable mass scaling in the seat structure.

Table 3 shows the details of the three models built up to investigate the influence of the mesh density on the result of the component test. An illustration of the difference in the mesh density on the lower cross member in also given.

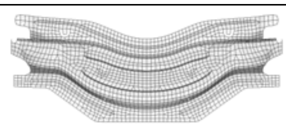
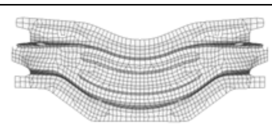
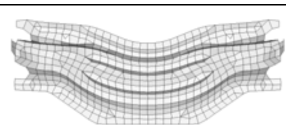
	Nominal Time Step (NTS) (µs)	**Average Element Size (mm)**	**Total Number of Elements**	**Mass Increase (%) @ NTS**	**Mass Increase (kg) @ NTS**
 **Fine Model**	0.6	5.5	33807	0.5	0.078
 **Medium Model**	0.9	6.5	27451	7.7	1.15
 **Sparse Model**	1.25	10.5	15412	12.3	1.892

Table 3. FE model details used to investigate effect of mesh density.

The mass increase for simulations with the 'nominal' time step (NTS) is also listed in Table 3, and is seen to be significantly greater in the sparser models. Most of this mass is applied at the longitudinal slides, Figure 5, where the seat is attached to the vehicle; here small elements have to be kept due to functional issues. The added mass does not have significant influence on the results of this component test, as these slides are rigidly constrained.

4.2 Element Formulation

Traditionally, elements with reduced integration (S3R/S4R) are used in explicit analysis in order to increase computational efficiency and avoid problems with shear locking which are sometimes experienced with fully integrated elements (S3/S4). The test was simulated using both of these element formulations in order to investigate the effect of the results and the robustness/stability of the model.

4.3 Connections

There are several types of connections in the automotive seat metal structure:

- spot welds
- welding lines
- screws and bolts
- rotational pivot connections

In order to accurately model the seat structure, the type of connection must be carefully considered, whether it may be modeled in a simplified way or not. For each type of physical connection an appropriate modeling method is required.

- Spot welds – modeled with Fasteners. Fasteners may have the capability of failure with progressive damage. In this investigation no failure models were implemented.
- Welding lines – modeled with Tie function, without failure.
- Screws and bolts – modeled with MPC Beam 'star' at the holes, connected with Connector type 'Cartesian, Cardan' to measure transmitted forces.
- Rotational pivot connections – modeled in a similar manner to screws, but the rotational capability assures Connector type Hinge.

4.4 Constraints/Boundary conditions

As in hardware, the seat model is constrained at the mounting points of the slides. The impactor is modeled as a rigid plate with appropriate mass and an initial velocity, Table 4, it is also constrained not to move and rotate in any other direction than the direction of the imposed velocity.

Although no bending of the impactor plate was allowed and no energy absorption (friction) between impactor and seat was considered, the imposed boundary conditions are sufficient to accurately reflect the real conditions. The supporting plate on the unstruck side is modeled as a fixed rigid body. Gravity was applied in the direction of the imposed velocity. A complete model is shown in Figure 10

Dimension of Impactor	200 mm * 200 mm
Drop Height	2.5 m
Impact Velocity	6.1 ms-1
Impact Energy	1283 J
Carrier Mass	69 kg

Table 4. Boundary conditions defined through ABAQUS pre-simulation.

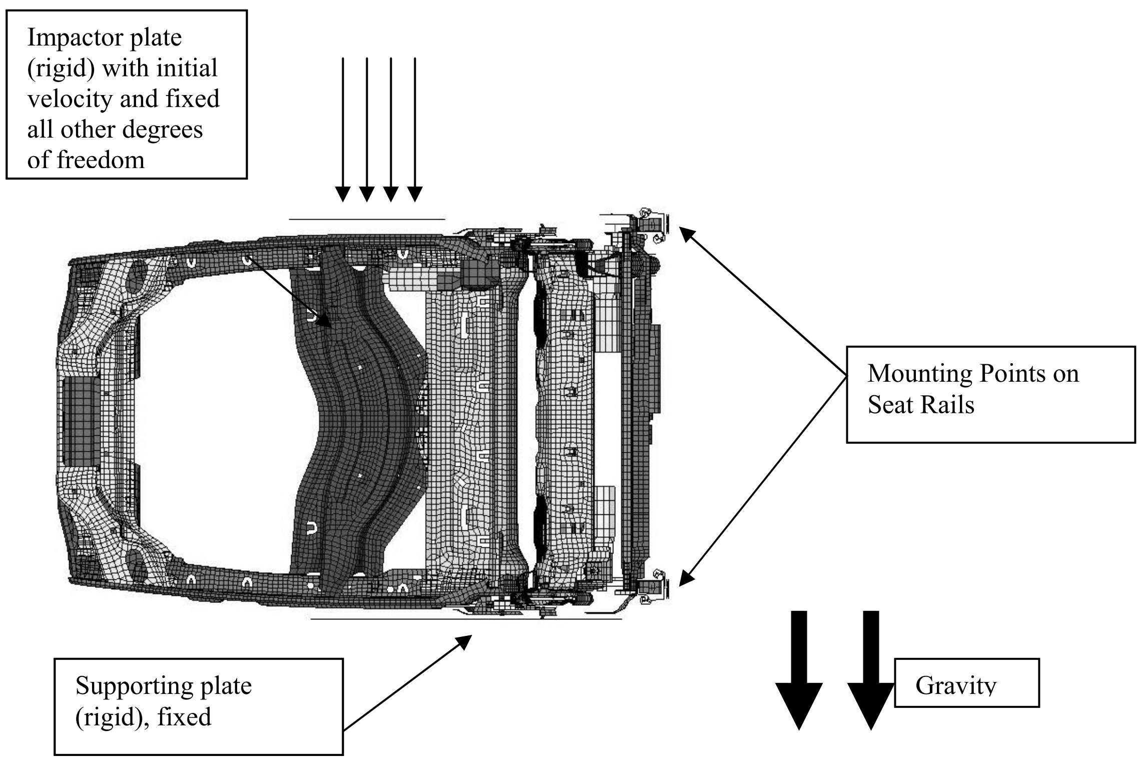


Figure 10. Complete FE model – boundary conditions.

4.5 Contact

As many seat components may interact with each other during the crash event, predefining individual contact couples would involve considerable effort. However, ABAQUS Explicit offers very useful general contact capabilities which allow all parts to be defined in one contact domain, which means considerably less effort in the predefinition of the contacts.

Considerable care is taken to avoid initial penetrations in the model, however, here again the ABAQUS capability to store offsets in the initialization phase means that initial penetrations pose less of a problem. The penetrating nodes are not initially adjusted but their required offsets are stored in memory, and resolved during the calculation.

For the parts in the backrest of the seat, where there the highest deformation is experienced, all edges, and not just those on the perimeters, are considered in contact interaction.

The friction coefficient was taken as 0.1 as all of the contact partners are of finished steel.

5. Comparison of Numerical and Test Results

5.1 Test Matrix

Having defined the test conditions through pre-simulation, it was decided to carry out the eight tests in all, as described in Table 5.

	Lower Impactor Position	**Higher Impactor Position**
Basis Seat	2 Tests	2 Tests
Multifunction Seat	2 Tests	2 Tests

Table 5. Test Matrix

Two impactor positions (upper and lower) were selected in order to assess the actual influence of the impactor position on the backrest deformation. Furthermore, the tests were carried out on both the basis seat structure and the multifunction variant, which has extra linkages in the backrest for comfort adjustment. For the purposes of this paper we will focus on the tests with the basis seat and the higher impactor position, where the deformation of the middle cross member corresponds to that in the foregoing investigation, as discussed in Section 3.2.

5.2 Comparison of results between test and FEA

Based on Faurecia experience, a baseline model with a time step of 0.6 µs, using S3R/S4R elements and without strain mapping was prepared. This model will be identified as ‘Ref’ in the following diagrams.

Investigations were then carried out to establish the influence of differing element types, time step and strain mapping on the correlation of the model to the test. To assess the correlation, two main criteria were used:

- Deformation image – visual assessment of the resulting deformation of the structure.
- Numerical assessment –displacement, and acceleration of the impactor and critical points in the seat.

Figure 11 illustrates the deformation comparison after test. Although the general deformation is similar, there are differences in the behavior of middle cross member of the backrest. In the hardware a distinct buckling is seen in the middle of the cross member, whereby a smoother bending deformation is seen in the simulation.

The optical measurement of the displacement at five points of interest on the seat will be discussed in the following section. These measurement points are shown in Figure 12.

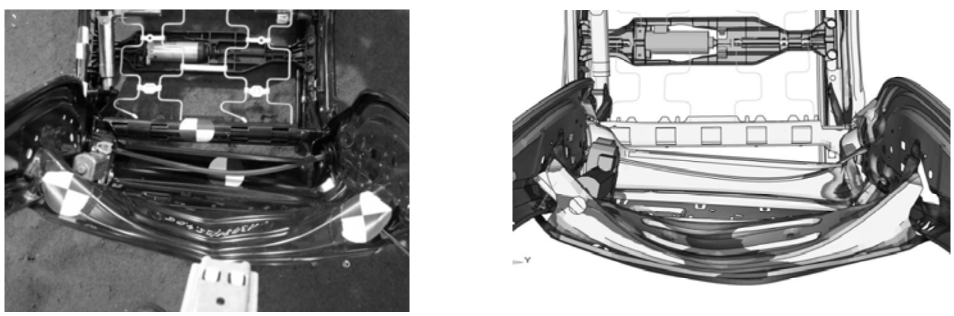
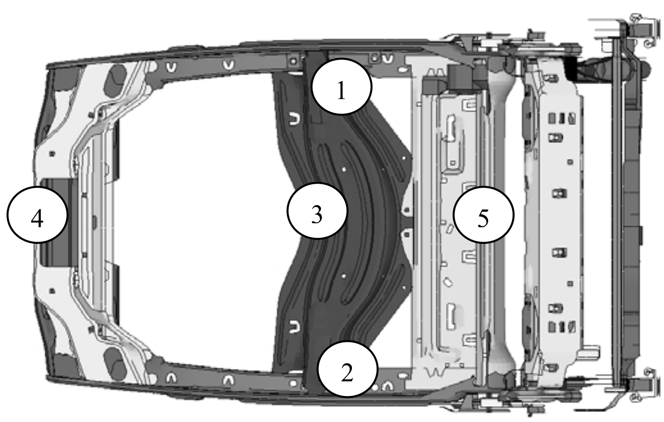


Figure 11. Comparison of real and simulated deformation in the seat backrest.



.Figure 12. Optical measurement points.

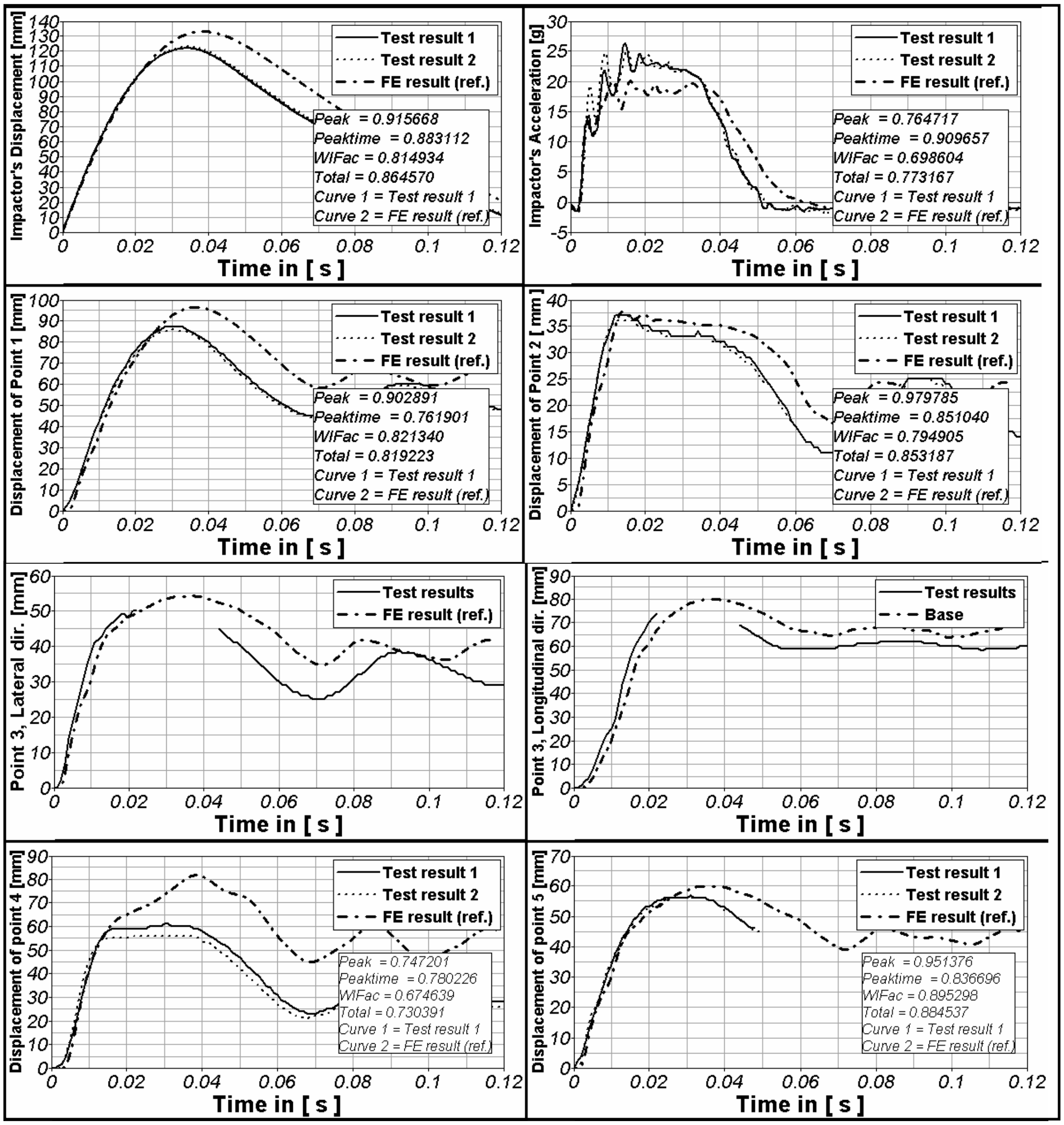


Figure 13. Comparison of results between test and FE (reference simulation).

The curves of displacement and acceleration, Figure 13, were compared with the results of real test using a rating tool, which numerically defines the convergence factors for the peak values, peak time and curve shape, and computes the total convergence factor. The impactor acceleration curves, both for the real test and for FE simulation were filtered with CFC 180 Filter. The results for the optical measurement points were not filtered. For point 3, the measurement was taken in lateral and longitudinal direction. However, as the target was partially obscured in the longitudinal direction by the test, the measurements taken in test were not complete (a gap in the curve).

It can be seen that simulation curves from the reference model fit the test with at least 80% accuracy (Total Correlation Parameter), which is acknowledged as well acceptable by Faurecia internal standards. Importantly, the loading phase of each curve correlated very well with the test, however, the peak deformation allowed appears to be too high, suggesting the model is not stiff enough. However, given the overall correlation, the use of this model as the baseline is justified.

5.3 Influence of Element Formulation

The remaining discussion deals with the sensitivity of the model to the various modeling options and, in the interests of brevity, will focus on the impactor displacement and the displacement of measurement points 1, 4, and 5.

The comparison of results for different element types, shown in Figure 14, indicates that the influence of the fully integrated elements S4 is not so significant in the loading phase, although the peak displacements do differ slightly. However, the computational time with fully integrated elements was seen to be approximately 3 times that with reduced integration. The global deformation behavior of the strongly deformed parts (e.g. middle cross member) did not change substantially.

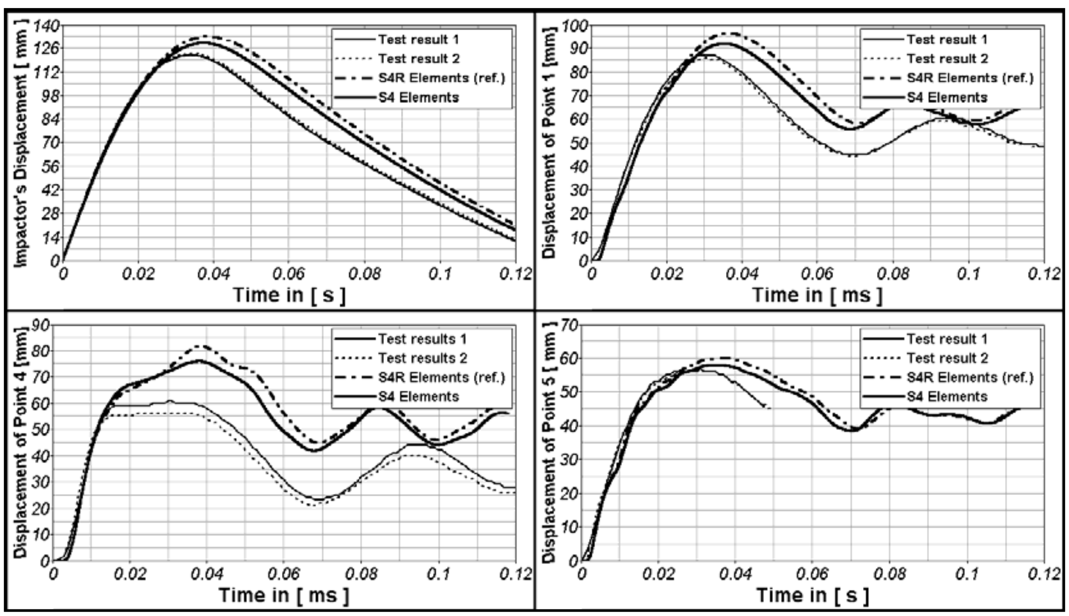


Figure 14. Comparison of results between element types.

Reduced integration elements (S3R/S4R) suffer from hourglassing modes, which introduce non-physical deformation. Fully integrated elements are not affected by this behavior. To avoid hourglassing ABAQUS implements an artificial controlling force on the element which can lead to over stiff response of the structure if the control coefficients are set too high.

The default value for these control coefficients as advised by ABAQUS is 1.0 and eliminates the hourglassing modes completely, but influences the deformation behavior excessively, with the artificial work done exceeding 10% of the overall internal energy. For the reference model above, coefficients of 0.2 were taken, resulting in artificial work done of less than 5%.

5.4 Influence of Mesh Density

The influence of mesh density (and thus stable time increment) on the model performance is shown in Figure 15. All three models were computed with S4R elements, non-default hourglass control coefficients of 0.2 and the respective nominal time steps.

The model with the sparse mesh predicts a significantly stiffer response, up to 15% less displacement of the impactor, whereby the 'middle' variant assures the results still similar to the Reference model and to the results of real test. This is a clear indication that for large deformation of the highly drawn components such as those in the seat structure, a stable time step of 1.25 µs (element length – 10 mm) is insufficient. However, the middle model, with a nominal time step of 0.9 µs is seen to produce comparable results to the fine model.

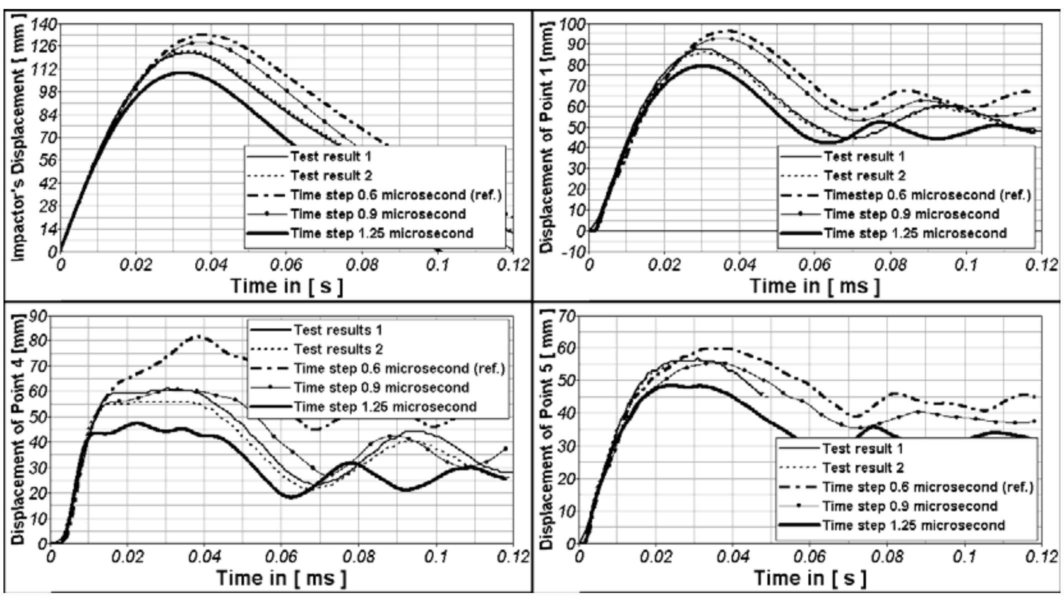


Figure 15. Comparison of results between time step 0.6 µs and 1.25 µs.

	Mass Increase at time step = 1.25 µs
Fine Model (Nominal time step = 0.6 µs)	17.5 kg
Middle Model(Nominal time step = 0.9 µs)	8.73 kg

Table 6. Initial Mass Scaling when calculated with a time step of 1.25 µs

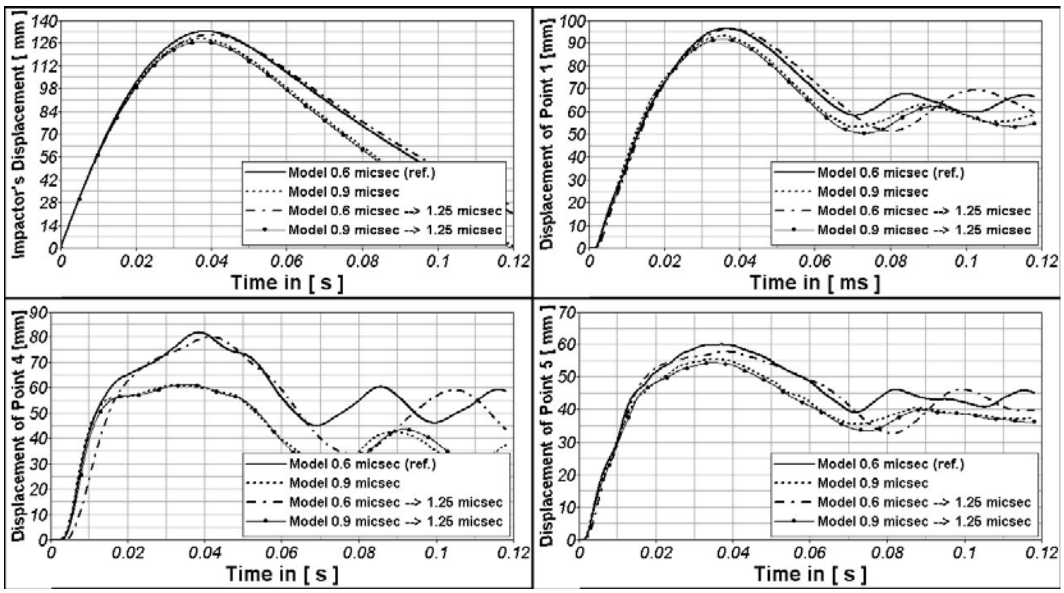


Figure 16. Comparison of fine and medium models calculated with nominal and increased time step (1.25 µs).

Also of interest is how the finer models behave when calculated with a time step of 1.25 µs as used in the BMW full vehicle crash simulations. Table 6 details the mass increase through initial mass scaling which, as described in Section 4.1, occurs mainly in the longitudinal slides.

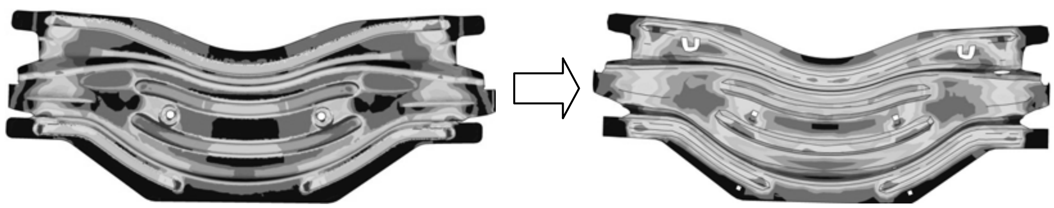
Figure 16 shows the influence of the higher time step on the performance of the fine and medium meshed models. It can be seen that the overall deformation is not significantly influenced by the time step used for the calculation; again, this is to be expected as only a small percentage of the added mass is added to the seat back area.

5.5 Influence of Pre-Strain Mapping

As mentioned in Section 5.4, the components of the seat structure are subjected to very deep drawing in the stamping process, resulting in strain hardening and thinning of the formed part. In the following section the influence of these pre-strains on the performance of the seat model will be investigated.

The results of forming simulations for the middle backrest cross member were obtained from a presimulation with a one step solver (Forming Suite) using three integration points through the thickness. A HyperWorks tool was used to map effective plastic strains (realized with *INITIAL CONDITIONS, TYPE=PLASTIC STRAIN option) and effective thickness (realized with *NODAL THICKNESS option) to the more sparsely meshed ABAQUS model, Figure 17.

Although, the one step solver approach assures only approximate values for stamping simulations, the results obtained with pre-strains for only one part show that the influence of stamping for highly redrawn parts is potentially significant, particularly at Point 4, Figure 18. A more extensive investigation of this topic is planned for the future.



One Step Solver Result

ABAQUS Input

Figure 17. Pre-strain mapping applied to the middle cross member.

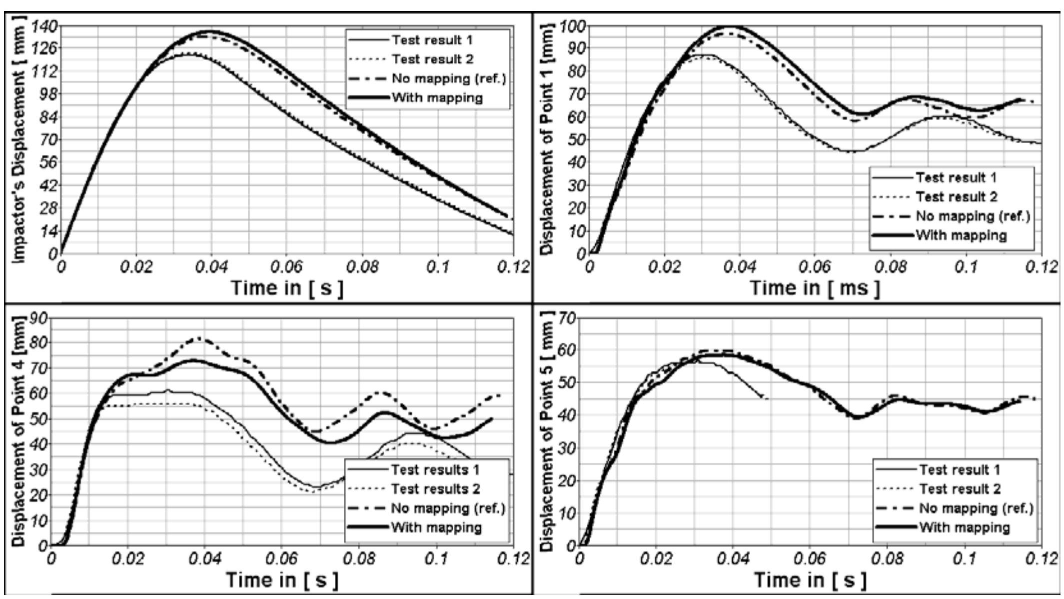


Figure 18. Comparison of results between mapped and reference model.

5.6 Influence of Scatter in the Material Properties

A common problem in validating FE simulation is the influence of scatter in material properties and geometric tolerances. In most cases a database of typical materials with nominal properties is used. However, the material properties of real parts may differ significantly due to production batches, stamping processes, surface finishing processes etc.

For the purposes of this research, the part with the most severe influence on the results, the middle cross member, was tested in the material lab. The laboratory test revealed that the actual part tested had significantly stiffer material than the nominal. The tensile strength was over 30 % higher (480 MPa instead of 340 MPa). In addition, the real thickness of the part was 1.1 mm instead of 1.0 mm.

The influence of these measured parameters is illustrated in Figure 19. The convergence with the test results is much better than for the base model, with point 4 showing particularly better correlation. The practical implication of this is that although the nominal values of the material and geometric parameters are generally valid, the tolerance bandwidth with which such components are manufactured has a significant influence on the results of FE simulation and should be considered in validation exercises.

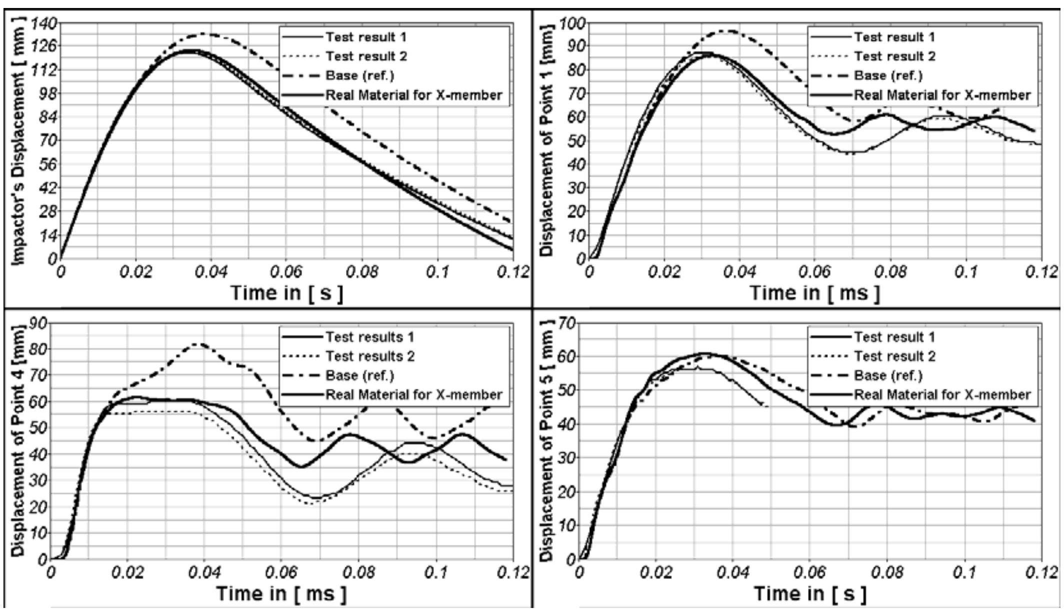


Figure 19. Comparison of results for real material properties.

5.7 Combined Effect of Validation Parameters

The foregoing analysis has highlighted the influence of several individual parameters on the performance of the seat model. In this section a seat model consisting of the optimum combination of parameters for use in a full vehicle simulation will be discussed.

We have seen in Section 5.3 that the use of S4R elements is acceptable for a model of this type and that the Hourglass Control settings should be carefully chosen. Hourglass control coefficients of 0.2 were found to produce the best compromise between stability and artificial stiffness in this model.

Furthermore, Section 5.4 shows us that the use of a relative sparse model with nominal time step of the order of 1.25 μs is not acceptable as the resulting behavior of the seat structure is significantly stiffer than that with finer meshes and that of the hardware. On the other hand, the use of a very fine mesh with a nominal time step of the order of 0.6 μs leads to additional mass of approximately 17kg being added to the mesh. This is quite unacceptable as the meshed structure itself has a mass of approximately 15kg.

However, the ‘middle’ model delivers performance which is only marginally stiffer than the very fine model with only 50% of the additional mass, approximately 9kg. Although still significant, this additional mass is considered acceptable, as in practice a number of electrical and comfort related components which do not influence the stiffness of the seat are not modeled, and the additional mass in the model compensates for their absence in the total mass distribution calculations.

As stated in Section 5.5, the effect of pre-strain mapping of the overall behavior of the seat model is potentially significant; however the method and effects have not been sufficiently investigated for practical application of the model at this time.

Finally, we have seen in Section 5.6 that the scatter in the material properties and geometric tolerances has quite a significant effect on the behavior of the seat model.

In practice, the tolerance definition for a mechanical component such as a seat, involves many minimum stiffness and robustness criteria, such that the tolerance for scatter in the downward direction (either in stiffness or thickness) is very low. However, the upper limits are generally only governed by cost and overall mass. The consequence of these looser upward tolerances is that it is generally found that components lie in the area between the nominal and the upper tolerance levels.

In light of this, it was decided to investigate the behavior of the model optimum parameter combination, detailed in Table 7, calculated with the real material properties, Figure. 20.

It can be seen that the model now exhibits very similar behavior at all 4 points illustrated when compared to the hardware results. Given the level of correlation, this combination of parameters is considered to be a very good compromise of accuracy and computational efficiency.

	Optimum Model
Element Formulation	S4R
Hourglass Control Coefficient	0.25
Element Length	Approx. 6,5 mm
Nominal Time Step	0.9 µs
Calculated Time Step	1.25 µs
Mass Increase	Approx. 9 kg
Calculated Time Step	1.25 µs

Table 7. Details of the 'optimum' model

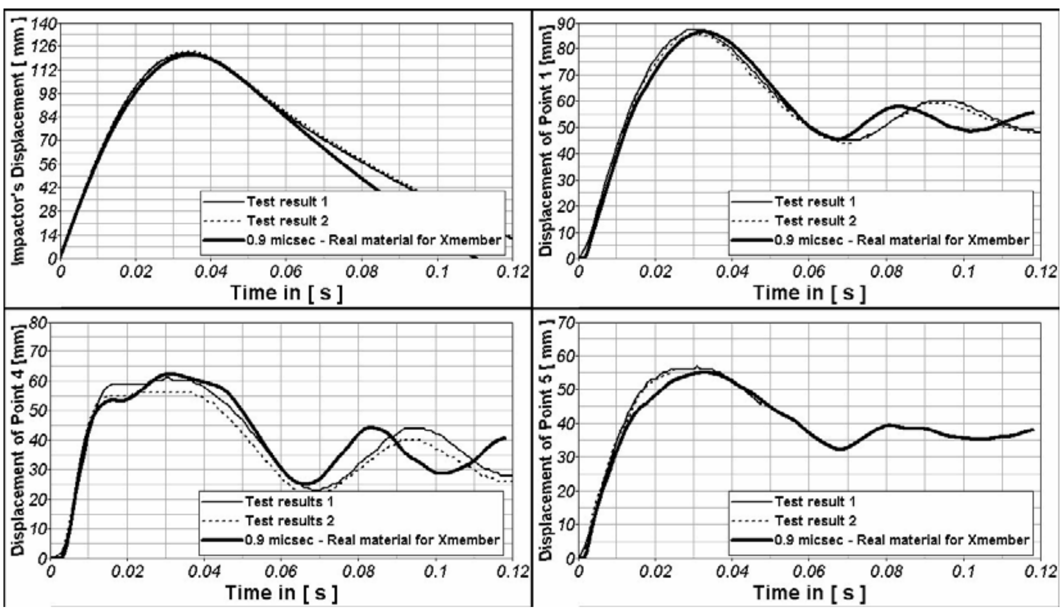


Figure 20. Comparison of the 'optimum' model with the test results.

6. Conclusion

This paper has described the definition, execution and subsequent use of a drop tower test in the validation of a FE seat model for side impact crash analysis.

The applicability of the drop tower test and the boundary conditions used have been discussed in relation to the level and type of loading experienced by the seat in a full vehicle crash situation.

The influence of the various modeling parameters available has also been investigated and it has been illustrated that the use of reduced integrated elements is acceptable, although the hourglassing control coefficients employed should be selected with care.

It has been demonstrated that for such a seat model the use of a mesh with an average element length of 6-7mm delivers acceptably accurate results without incurring excessive mass scaling in the critical areas of the seat structure, even when calculated with the relatively high time step of 1.25 μs as used in full vehicle crash simulation.

Furthermore, it has been shown that the influence of pre-strain in the deeply drawn components of the seat structure has a potentially significant influence on the deformation of the structure.

Finally, the influence of the scatter in the material and geometrical parameters has been exhibited and the need for awareness of this scatter in validating and using such FE seat models highlighted.

Some important questions have been answered though this paper, however, a number of issues require further investigation, for example:

- Is the influence of more detailed pre-strain mapping actually any more significant and if so is the increase in accuracy worth the extra computational effort?
- To what degree is the full vehicle occupant simulation sensitive to the scatter of +/- 15% in overall stiffness resulting from the modeling variations and material/geometry scatter in the hardware parts?

As ABAQUS is now the sole FE code used in crash development for new vehicle projects at BMW, these issues will be the subject of further investigation in the coming months. The long term goal of BMW is to establish industry standards for the validation of such FE models, this process will include consultation with the various component suppliers and indeed other vehicle manufacturers.