Strength Analysis of Seat Belt Anchorage According to ECE R14 and FMVSS

Author:

Klaus Hessenberger
DaimlerChrysler AG,Stuttgart, Germany

Correspondence:

Klaus Hessenberger
DaimlerChrysler AG
HPC B209
D-70546 Stuttgart
Germany

Tel: +49-(0)711-1724281
Fax: +49-(0)0711-1723721
e-mail: klaus.hessenberger@daimlerchrysler.com

Keywords:
Strength analysis, nonlinear structural mechanics,
finite element modeling

ABSTRACT

To guarantee proper function of the seat belt system, belt anchorages have to resist defined static test loads that represent an vehicular impact. ECE R14 and FMVSS210 are tests to ensure sufficient strength of all anchorage points. In these tests high forces are applied to the seatbelts over loading devices. All components of the sytems, namely seats, seat and belt anchorages have to resist the defined loads without damage. The loads are applied slowly and are sustained over a long period of time, so one can assume a quasi static test.
The correct modelling and simulation of the complex load application system is essential for significant and accurate computational results.
The experimental test with an existing drivers cab according to FVMSS 210 was simulated with Abaqus Standard (implicit) and LS-Dyna (explicit). During the application of both tools, problems specific to each system were encountered.
In Abaqus, problems were caused by large deformations of the sheet structure and possible local buckling phenomenons. In the LS-Dyna calculations the presence of dynamic effects have to be minimized to yield a good correlation with the quasi static tests. The problems encountered and the approach used are presented and a comparison between test and analysis will be given.

TEST SPECIFICATION

ECE R14 and FMVSS 210 are tests to ensure the strength of the seats, the seatbelts and the anchorage points. Therefore, test loads are applied over loading devices, so called body blocks, see Figure 1, and transferred by the seatbelts to the vehicle structure.

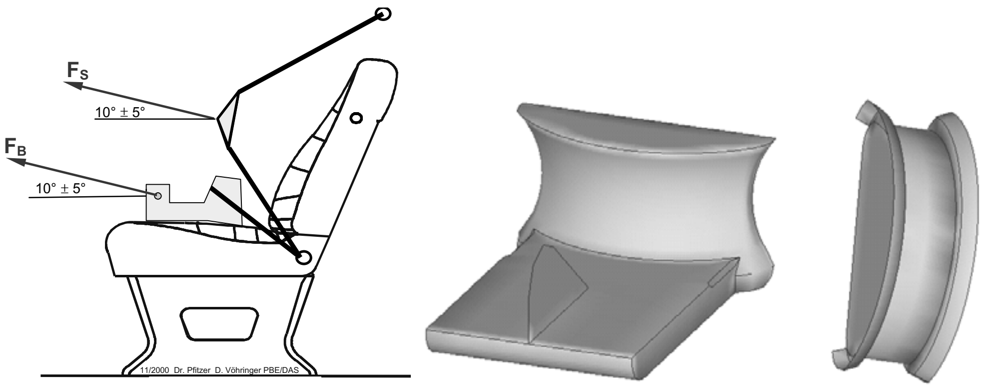

Figure 1 Sketch of the load application and shape of lap and shoulder body block

Because the loading devices are not tied to the seatbelts or the seats, contact and slipping between all parts can occur. Therefore these parts (seat, seatbelt, slipring, loading device) build a complex kinematic system and the configuration under load determines the distribution of the applied loads to the anchorage points. Hence a correct modeling of the kinematics is essential for significant and accurate computational results.
There are mainly two differences between the European ECE R14 and the NAFTA FMVSS 210. The ECE R14 classifies the vehicles on basis of their maximum allowed weights and requires them to sustain different loads dependent on their weight (see

Table 1), whereas in tests according to FMVSS 210 the same loads are applied to all vehicles. Because the tested drivers cab belongs to a class N2 vehicle in Europe, the applied loads are 6.75 kN on each block, whereas in the NAFTA countries it has to sustain the full 13.5 kN on each body blocks.

	Classification		
	N1: m < 3.5 t	N2: 3.5 < m < 12 t	N3: m > 12 t
Shoulder Block	13.5 kN	6.75 kN	4.5 kN
Lap Block	13.5 kN	6.75 kN	4.5 kN
Seat	20 x seat weight	10 x seat weight	6.6 x seat weight

Table 1 ECE R14 test loads

The second main difference is the velocity of load increase and the time the vehicle has to sustain the maximum load. While ECE R14 requires the load to be increased as fast as possible and the anchorages have to withstand at least 0.2 seconds, the FMVSS 210 requires a loading ramp between 1 and 30 seconds and the structure have to sustain the loads 10 seconds. Therefore the FMVSS test can be viewed as a static test.

SIMULATION WITH ABAQUS

There are two main difficulties in using Abaqus for simulation of the seat anchorage tests. The first is the correct modelling of the seatbelts, sliprings and the body blocks, because Abaqus provides no tools for modelling these kind of structural components. To circumvent these problems a special user element was implemented in Abaqus, that can slip through a fixed midpoint and uses a tension only material. With these elements the sliprings and the contact between the body blocks and the seatbelts can be realistically modeled.
The second difficulty arises from the use of a static implicit simulation procedure for a problem where local instabilities, like buckling, can occure. In cases where the built-in stabilization procedure is not sufficient, due to very small load increments and the consequential long run times, one has to manually introduce damping devices in the model, such as dashpots.
Due to the lack of a automatic contact searching, the definition of all necessary contact interfaces is a very time consuming and error sensitive part of the process.
But if one has built the finite element model correctly a simulation with Abaqus will yield accurate results, without the danger of overestimating dynamic effects.

SIMULATION WITH LS-DYNA

Compared to Abaqus the main difficulty in LS-Dyna simulations is not the modelling of the loading system, though these have to done with care too, but to suppress unwanted dynamic effects. Setting a global damping constant is an efficient method for these purpose, but the damping value has to be chosen with respect to the relevant eigenmodes of the structure. If the load application time, the holding time and the damping constant are properly chosen a balanced state can be reached.

With LS-Dyna one can use the built-in capabilities to model the complete load application system. Seatbelt and slipring elements are used to model the belts not in contact with other parts of the structure. The region where belts and loading devices are in contact, have to be handled with special care. As mentioned earlier, slip between the loading device and the seatbelt is allowed, so that the load has to be transferred over a contact condition between belt and body block. A numerically more robust method of modelling this contact than with seatbelt elements is the use of membrane elements for the belt in this region only. Figure 2 shows an example for the modelling of the complete load application system consisting of seatbelt, body block, sliprings and seat.

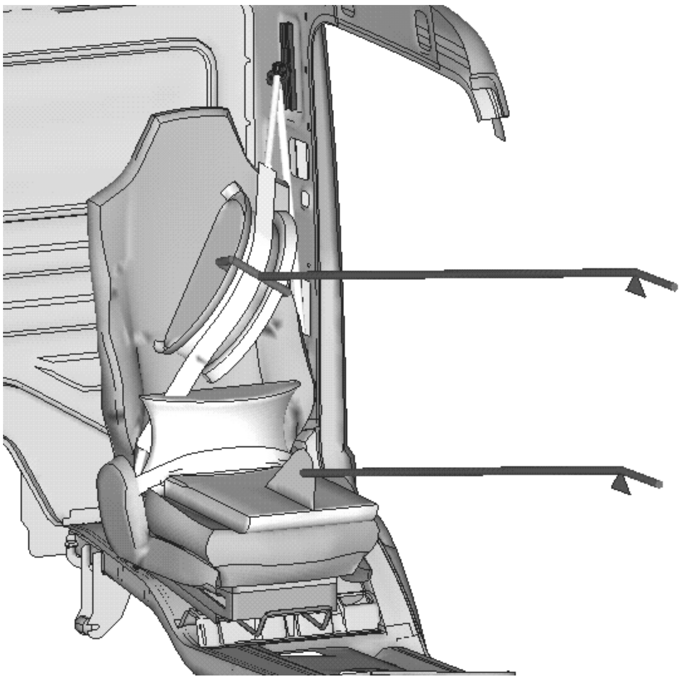

Figure 2 Modeling of seatbelts and loading devices in LS-Dyna

COMPARISON WITH TEST RESULTS

For the comparison between computations and test results a existing drivers cab was chosen. In FMVSS 210 no classification by vehicle weight is provided, which means that the full loads of 13.5 kN have to be applied regardless of the maximum vehicle weight. Because this are stronger requirements than the 6.75 kN ECE R14 demands, only this test was performed. Due to the high loads one can expect large deformations of the body in white. Indeed, as Figure 1 shows, large deformations at the slip ring occur, but no breakdown of the structure. The comparison between LS-Dyna simulations and the test results show a good correlation of the overall deformations of the cab and also of the local deformation at the anchorage points of seatbelts and seats. A direct comparison of the slipring anchorage is given in Figure 3 which shows a nearly perfect agreement between simulation and test.

The drivers cab passed the FMVSS 210 test with minor modifications at the lower belt anchorage points.

Figure 3 Undeformed state and comparison between LS-Dyna simulation results and experimental results for the FMVSS test

SUMMARY

Both methods, the implicit static an the explicit dynamic, can produce realistic results for the strength analysis of seat belt anchorage. Using the implicit method, one has to take special care of local instabilities, like buckling, which may occur in the computation. This is usually no problem in the explicit method. The fact that in explicit computations the loads are applied much faster than in reality, can be outweighted by the proper choice of global damping.
The program of choice not only depends on the numerical properties of the method, but also on the modeling possibilities of the available computational codes and the CAE-Environment. For realistic computational results one has to describe contact conditions between various parts of the structure. Using the automatic contact capabilities of LS-Dyna this is much easier than defining the contact interfaces one-by-one, as is necessary in Abaqus. Furthermore, using tested seat models, which are normally available as LS-Dyna models only, will result in time saving for the overall process of modeling, simulation and correction.
Due to the above reasons, the switch from Abaqus to LS-Dyna can yield a much higher throughput.