Safety Analysis of the New Actros Megaspace Cabin According to ECE-R29/02

Author:

Horst Raich
DaimlerChrysler AG, Stuttgart, Germany

Correspondence:

Horst Raich
DaimlerChrysler AG
HPC: B209
D-70546 Stuttgart
Germany

Tel: +49-(0)711-17-24268
Fax: +49-(0)0711-17-23721
e-mail: horst.raich@daimlerchrysler.com

Keywords:
Commercial Vehicles, Safety analysis, ECE-R29,
finite element modeling,

Abstract

During the development of truck cabins the safety of the driver and the front seat passenger in an accident is considered. The cab must be designed in such a way that in an accident a sufficient survival space is guaranteed. The legal requirements of cabin safety are fixed in Europe in the regulation ECE-R29.
In order to reduce the number of iteration loops during the development process, a computational simulation method for the load cases roof strength test, front impact test and rear wall strength test of the ECE-R29 was introduced.
The explicit finite element program LS-DYNA was used for that purpose. The deformations of the driver‘s cab and the loads of the individual components within the elastic and plastic range of the material behaviour can be determined before the first tests are carried out. These tests can then be limited to a minimum by the numeric simulation.
In this paper, the application of this numerical method by the example of the new of ACTROS Megaspace cab is presented and compared to the results from the acceptance test according to ECE-R29.

1. Introduction

The risk of injury for truck occupants is statistically seen relatively low, compared with the high road performances. Nevertheless the automobile industry is endeavored to further increase safety for the occupants. For this purpose specific design measures are necessary.
In several FAT studies [1,2] the accident details of commercial vehicles were examined. According to these studies the highest risk exists in frontal collisions, which lead in approximately three from four accidents to injuries of the truck occupants. Here seat belts and airbags can protect against injuries and reinforced cab structures can reduce the risk of getting jammed. In order to exclude the danger of injury of the occupants to a large extent, the driver‘s cab must be dimensioned in such a way that in case of a rear end collision, rolling over of the vehicle on the side or on the roof, or by load slipping in the case of a front impact, the strength and stiffness of the cab structure is sufficient enough to secure the necessary survival space for the occupants.

From investigations of the accident details with commercial motor vehicles different characteristic test loads were derived, which the driver‘s cabs must withstand. The legal requirements of driver‘s cab safety are fixed in Europe in the regulation ECE-R29 and in Sweden in the state-specific VVFS 1994:22.

For weight- and cost-optimized dimensioning of truck cabins the numeric finite element simulation is suited very well already in the design phase. Before a first test vehicle is built, the behaviour of cabs under most different load conditions can be examined. Thus the number of tests and possibly necessary cost- and time-intensive tool changes can be substantially lowered.

In this article a computational simulation method is presented which was developed by the numerical analysis department commercial vehicles of the DaimlerChrysler AG to analyse the load cases of the ECE-R29 numerically . The effectiveness of the method is shown on the basis of a comparison with the results from the acceptance test according to ECE-R29.

2. Legal requirements of the ECE-R29 Regulation

The legal requirements of cabin safety are fixed in Europe in the regulation ECE-R29. As from 1, October 2002 ECE–R29 approvals can only be granted, when the requirements as specified by the 02 series of amendments are fullfilled. A short description of the tests demanded in this regulation and the requirements for the vehicle for fulfilling these tests is given in this paragraph.

This European regulation ECE-R29 /02 series of amendments contains a three-part test of the cab:

- **Front impact test (A)**
- **Roof strength test (B)**
- **Rear wall strength test (C)**

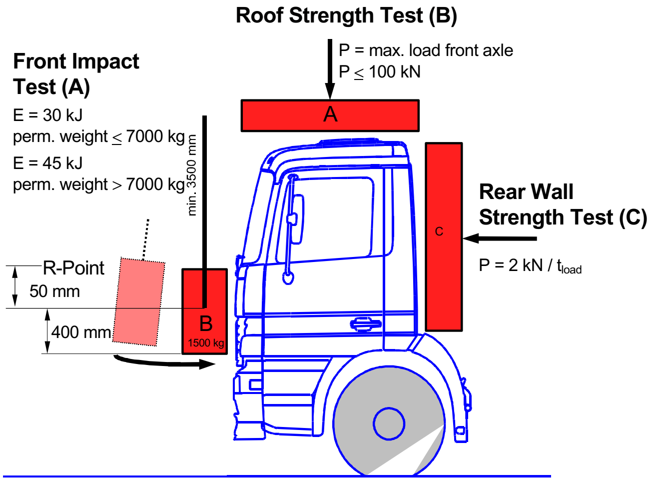

Fig 1: Cabin safety tests according to ECE-R29

2.1. Front impact test (A)

The rigid pendulum with a striking surface of 2500 mm x 800 mm and a mass of 1500 kg ± 250 kg must be so postioned, that in it's vertical position the centre of gravity is 50 +5/-0 mm below the R-Point of the driver's seat. This is different to the preceding version of this regulation where the vertical position of the centre of gravity was 150 +5/-0 mm below the R-Point of the driver's seat with a maximum height above ground of 1400 mm. This change leads to the fact that the pendulum now impacts the front panel of the cab with most vehicle versions, while in the preceeding version of ECE-R29 mostly the cab suspension or the frame front end was impacted. Fig. 2a and 2b show the different pendulum positions according to ECE-R29, by the example of the ACTROS-Megaspace cab, which are prescribed after and before 1, October 2002.

The impact energy of the pendulum has to be 30 KJ for vehicles of a permissible maximum weight up to 7000 kg and 45 kJ for vehicles for which the permissible maximum weight exceeds this value.

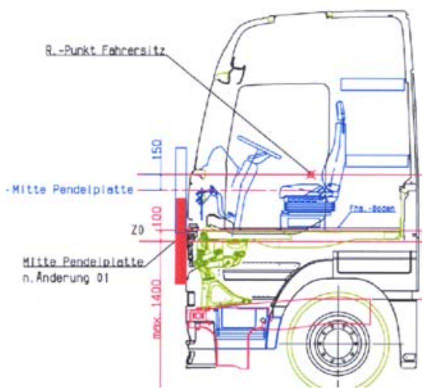

Fig. 2a: Pendulum position according to ECE-R29/01

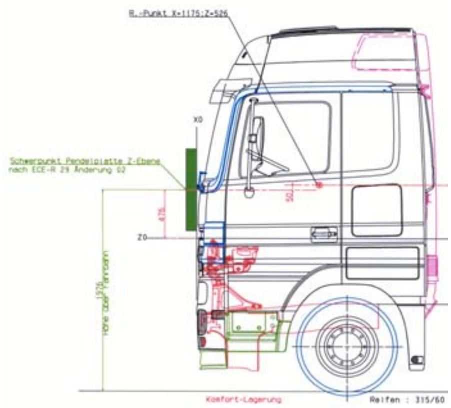

Fig. 2b: Pendulim position according to ECE-R29/02

2.2. Roof strength test (B)

The roof of the cab has to withstand a static load corresponding to the maximum load authorized for the front axle of the vehicle, subject to a maximum of 10 tonnes. This load is to be distributed uniformly over all the bearing members of the roof structure by means of a rigid plate. Deformation of the cab suspension shall be eliminated by means of rigid members.

2.3. Rear wall strength test (C)

The rear wall of the cab must withstand a static load of 2kN per tonne of the vehicle's permissible payload. This load shall be applied by means of a rigid barrier perpendicular to the longitudinal median axis of the vehicle, covering at least the whole of the cab rear wall situated above the chassis frame and moving parallel to that axis.
It is left to the manufacturer whether all three tests A, B and C or only the tests A and B are carried out. Furthermore the tests can be carried out successively on the same cabin or in each case with a new cab.

2.4. Requirements

The cab of the vehicle must be so designed and so attached to the vehicle as to eliminate to the greatest possible extent the risk of injury to the occupants in the event of an accident.
After undergoing each of the tests referred to above a survival space has to be present, allowing accomodation of the test dummy defined in ECE-R29 on the seat in the centre position, without contact between the test dummy and non-resilient parts. The survival space so defined has to be verified for every seat provided by the manufacturer.
During the tests the parts with which the cab is fastened to the chassis frame may deform or break, as long as the cab remains connected with the frame. The doors may not open during the tests, but the doors shall not be required to be opened after testing.

3. Numerical Simulation of the ECE-R29 tests

With the example of the ACTROS Megaspace cab the simulation method used at the DaimlerChrysler AG is presented.

3.1. Description of the model

3.1.1. Geometry and Boundary constraints

The cab structure, the cab suspension, the tilt cylinders, the steering system, the instrument panel, the structural frame of the front-end flap, the front part of the vehicle frame with engine and radiator and the subframe with the chains, usually used during the testing for clamping the test set-up were meshed with finite elements (Fig. 3). The overall model consists of 428.000 nodes, 320.000 shell and 172.000 solid elements.

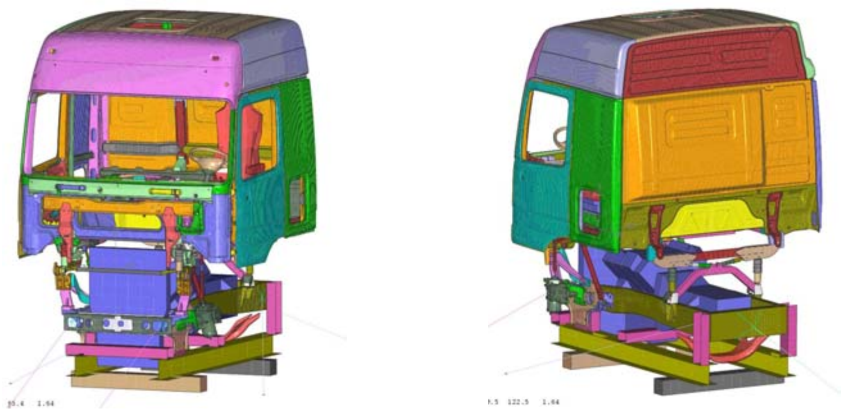

Fig. 3: FE-Model of the Test Set-Up

The underintegrated Belytchko Tsay shell element (Type 2) was mainly used for the shell structure. However these elements show a noticable lower bending stiffness than fully integrated elements, caused by their single integration point. Underintegrated shell elements are unably to carry in plane bending loads and can be sensitive to hourglass modes under arbitrary loading. Therefore load-bearing parts were modelled with the fully integrated shell element (Type 16), especially when the recommended using of 3 underintegrated elements per side of any open or closed section [3] could not be used, due to time step reduction. Additionally 5 integration points through the thickness were selected for all shell elements in order to guarantee a correct elasto-plastic behaviour for out-of-plane bending.

Solid structures like bearing brackets are meshed with solid elements. If hexahedrons are mixed with tetrahedrons and pentahedrons under the same part id, degenerate tetrahedrons and pentahedrons are used. These elements, especially pentahedrons, sometimes occured to be unstable and led to error termination of the job, propably caused by an uneven mass distribution in the element [4]. It was tried to mesh the whole part with eight-node solid hexahedron

elements, avoiding pentahedrons at all costs. For some solid properties fully integrated solids (Type 2) had to be used due to hourglass problems.
Some solid components however were taken over from existing FE-models which were modelled as pure tetrahedron mesh. For these parts the four node tetrahedron element with one integration point (Type 10) was used.
An AUTOMATIC_SINGLE_SURFACE (Type 13) Contact was used for general contact conditions. In some regions of the model an AUTOMATIC_GENERAL Contact (Type 26) had to be introduced to ensure that edge to edge contacts work properly.
Spotwelds were meshed as element to element connections with beam elements (Type 9, MAT_SPOTWELD) and are conected to the corresponding shell structure via a CONTACT_TIED_SHELL_EDGE_TO_SURFACE (Type 7) contact.

3.1.2. Material Model

The isotropic elastic-platic material model (MAT_PIECEWISE_LINEAR_PLASTICITY, MAT 24) was used. Strain rate effects were considered by stress versus strain curves for various strain rates, when available. This material model describes the deformation behaviour occurring with the safety tests with an appropriate fine mesh density for sheet metal materials with good accuracy.
This material model was used also for the solid components made of cast aluminum and grey cast iron. For this component/material combinations no suitable material models were available in order to better predict cracks, crack propagation and fracture in the components. Shear failure was considered in the material models for bolt connections.

3.1.3. Load Conditions

Front Impact Test
The pendulum was idealized as a shell structure, which was set rigid with the MAT_RIGID card. At the same time only the rotation around the y-axis was set free. The energy of the pendulum was controlled via the PART_INERTIA card. The INERTIA Option allows the inertial properties and initial conditions to be defined rather than calculated from the finite element mesh. This applies to rigid bodies only. The correct length of the pendulum arms was set via the input of the centre of gravity for the rigid body in this card.

Roof and Rear Wall Strength Test
The plate which applies the roof load and/or rear wall load to the structure was also meshed as a rigid shell structure.
The roof crush test was carried out at very low speed and should be regarded as a quasistatic test. In terms of the analysis this means ignoring strain rate effects and applyinig the fixed velocity to the ram plate at sufficient low velocity not to induce dynamic effects. Several analyses showed that by applying the load with a velocity of 1 m/s, a very good agreement with the appropriate test results is achieved. The BOUNDARY_PRESCRIBED_MOTION_RIGID card was used for this purpose. The cab’s roof/rear wall load was output by using a CONTACT_FORCE_TRANSDUCER_PENALTY (Type 25) contact in the rcforc file. Modelling with these specifications achieved a workable compromise between accurate deformations and CPU time.

3.2. Results of the Numerical Simulation

3.2.1. Front Impact Test

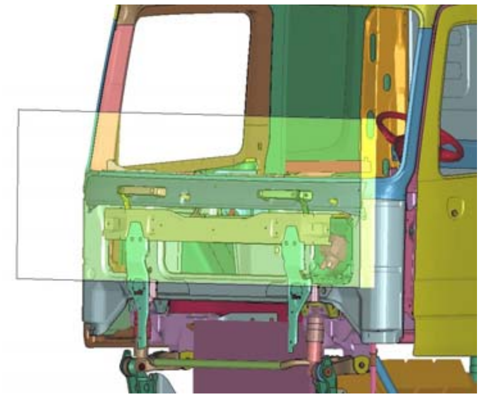

Fig. 4: Position of pendulum in FE-model

Due to the change for the pendulum position in the 02 series of amendments of the ECE-R29 the centre of gravity of the pendulum is now, as in paragraph 2 already mentioned, 50 mm below the R-point of the driver's seat in centre position. With this arrangement the pendulum impacts with most vehicle versions the front wall and the front wall crossbar above the front cab suspension.

To verify the influence of this change of the pendulum level on the survival space, the front impact test was numerically simulated in a first analysis without any changes made to the cab.

It was shown that the survival space for the driver was not sufficient. The pendulum first strikes the hand grips and windshield wiper axles of the vehicle, over which the energy of the pendulum is applied into the upper part of the front wall. This causes a tilting of the front wall around the y-axis. As the steering unit is fastened to the front panel, a strong lowering of the steering wheel rim towards the driver's seat was caused. This resulted in the steering wheel rim contacting with the driver's seat. Thus the dummy could not be inserted anymore.
An effective solution had to be found, in order to ensure the survival space for the driver.
For the harmonization of the deformations within the front panel area, the corresponding contact zones and stiffness distribution were optimized in such a way that the front panel was now pushed inward relatively parallel and thus strong tilting of the steering wheel was prevented. This was caused by the fact that the pendulum hits the stiffness and strength optimized front panel briefly after the contact with hand grips and wiper axles.

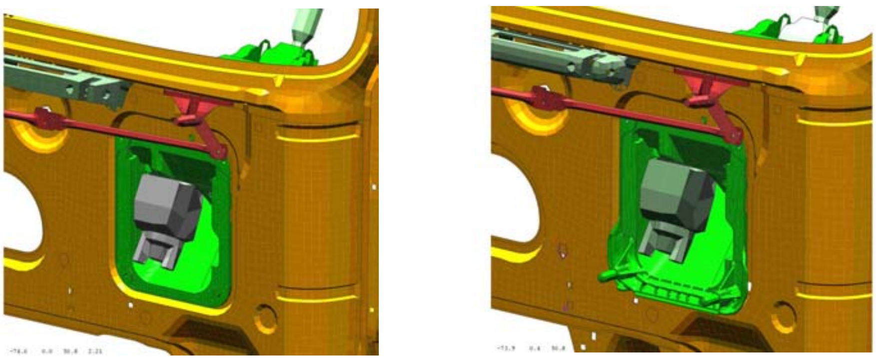

Fig. 5: Front Panel and Pedal box before and after Optimization

Figures 6a and 6b show the effect of the new versus the old pedal box during the simulation process.

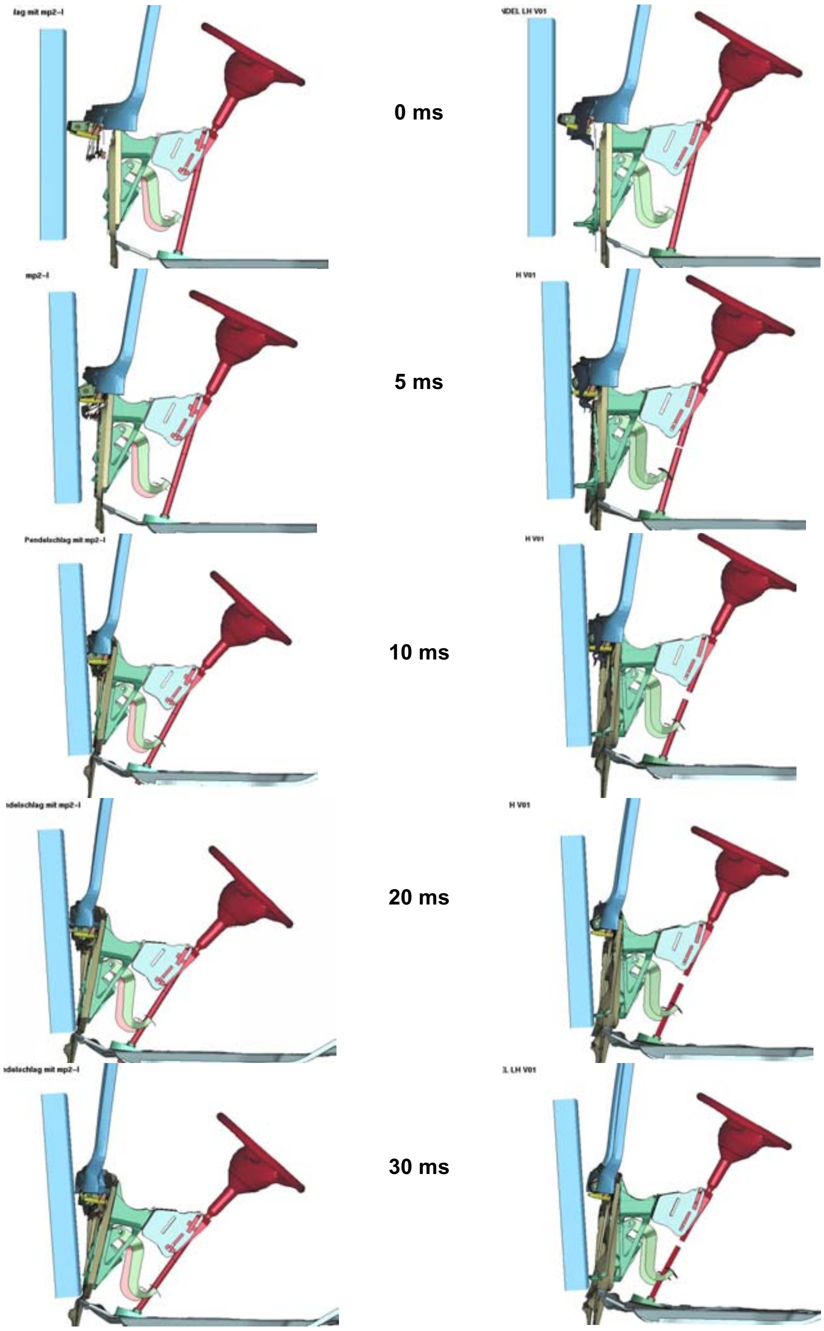

Fig. 6a: Deformation Sequence with Original Pedal Box

Fig. 6b: Deformation Sequence after Integration Of the Stiffness Optimized Front Panel

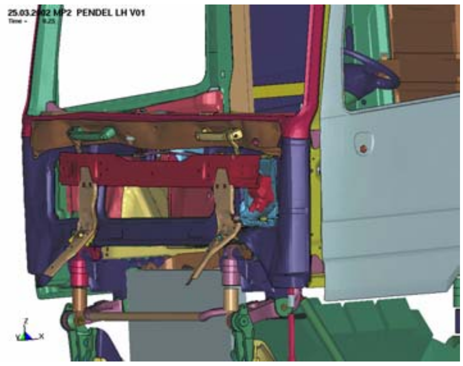

Fig. 7 : Deformed Structure after Pendulum Impact

Figure 7 shows the deformed structure of the front end area of the cab after the pendulum impact. One can recognize the hand grips and the structural frame of the front-end flap distorted strongly by the impact of the pendulum. Also the front wall which was moved parallel to the rear by the effect of the optimized front panel and pedal box and thus the reduction of the tilting of the steering column towards the drivers's seat can be seen. Apart from the movement of the steering column the pedal intrusion was also considered. Both remained small enough and so the survival space for the driver remained sufficient (fig. 8). A good matching for the remaining survival space of the driver between analysis and acceptance test results was seen.

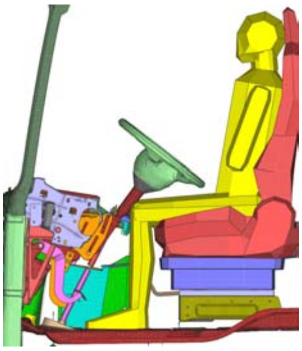

Fig. 8: Survival Space Resulting from Numerical Simulation

The rear bolted connection of the adapter console with the frame fails as desired (fig.9). All other bolted connections of the cab suspension and the tilting cylinder remain however intact, thus the cab remains connected with the chassis frame. The bolt connections of the cab suspension with the frame are however highly loaded. In order to guarantee that these bolt connections do not fail in any case, these were overdimensioned. In order to make more exact predictions, the influence of the pre-load should be considered in future analyses.

The stresses and strains for the components of the front and rear cab suspension as well as the tilting cylinders coming out from the pendulum impact did not result in exceeding of the permissible limit values.

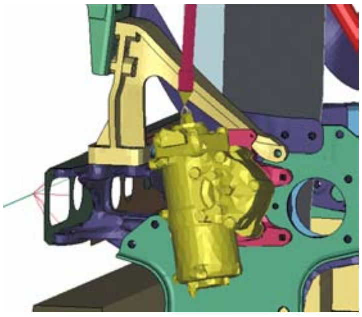


Fig. 9: Comparison of Bolt Failiure Resulting from Numerical Simulation and Acceptance Test

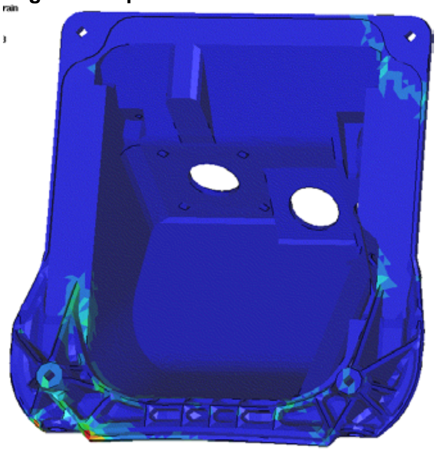

Fig. 10: Strain Distribution in Optimized Pedal Box

The pedal box this is highly loaded as a result of the impact of the pendulum on the domes. For this reason the material of the pedal box was changed from from EN AC—AlSi8Cu3 to Magsimal 59 (GD-AlMg5Si2Mn). This alloy was developed for innovative constructions of safety components using pressure casting.

In several steps the ripping of the pedal box was optimized. This optimized pedal box made of Magsimal 59 has an acceptable stress and strain behaviour (fig. 10).

3.2.2. Roof Strength Test

The maximally permissible front axle load for the ACTROS with megaspace cab amounts to 9 tons. This corresponds to a roof load of 88.29 KN, which has to be reached in the acceptance test. As in the test, the rear shock absorbers in the computational model were replaced by rigid components.

The test load causes the roof panel to intrude into the cab in the roof lid area. The backside of the roof and the lateral roof panels showed buckling. A roof load of 90 kN could be achieved together with the sufficient survival space required by the ECE-R29 (fig. 11, 12). Deformations resulting from the computational results matched very well with the deformations seen in the acceptance test after ECE-R29 (fig. 13).

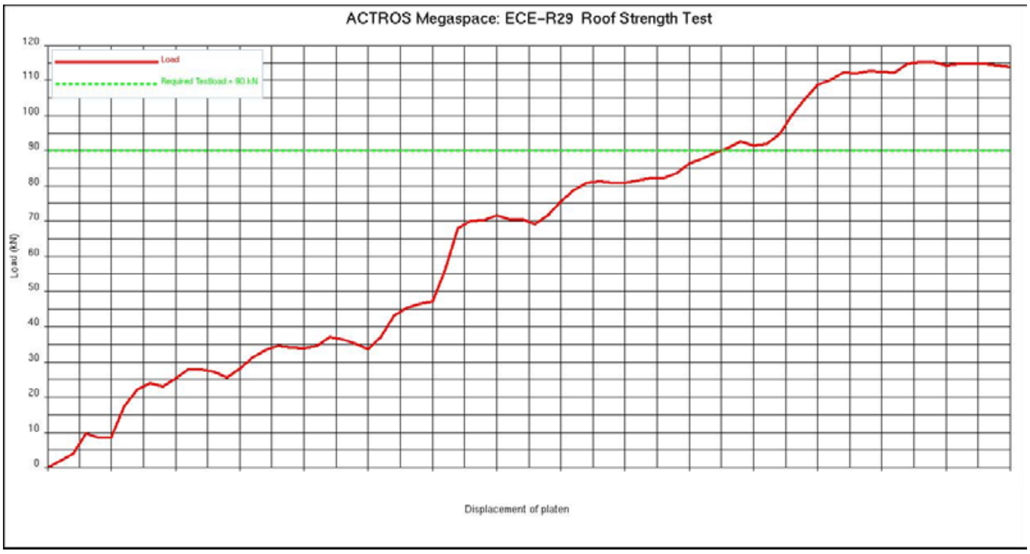

Fig. 11: Roof Load Curve Resulting from Numerical Simulation

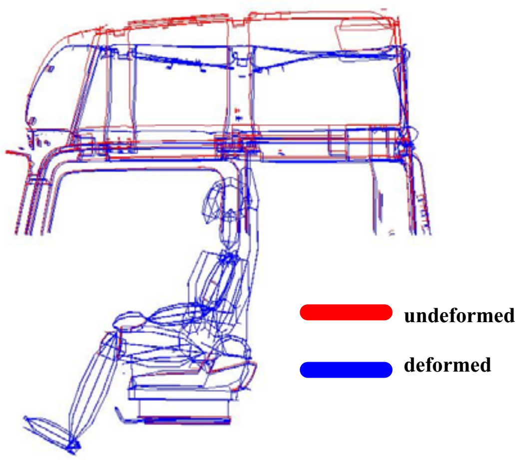

Fig. 12: Deformation of Roof Structure Resulting from Numerical Simulation

Fig. 13: Comparison of Roof Deformation Resulting from Numerical Simulation and Acceptance Test

3.2.3. Rear Wall Strength Test

A load of 66.4 kN coressponding to a maximum payload of 33 tons has to be applied for the rear wall strength test. This rear wall load could be achieved together with sufficient survival space for all occupants required by the ECE-R29. The load curve shown in fig. 14 still contains the movements of the cab in the cab suspension as well as elastic deformations of the cab. Again a good matching between numerical and test results was seen.

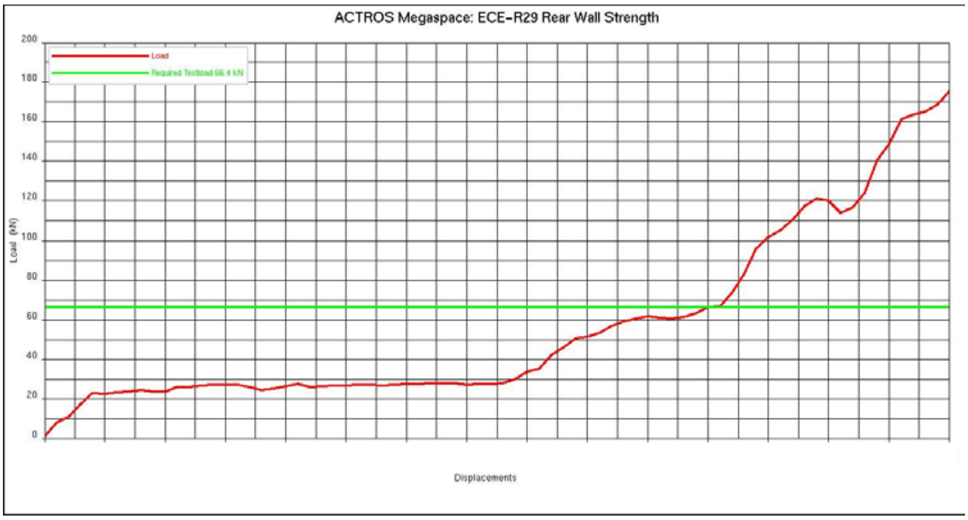

Fig. 14: Rear Wall Load Curve Resulting from Numerical Simulation

4. Summary and Outlook

With the numerical method presented in this paper and the FE model coordinated with it, it is possible to compute safety tests according to ECE-R29 with sufficient accuracy. This was shown with the example of the new of ACTROS megaspace cab.

Within a short time by means of the numeric simulation a solution was found for fullfilling the requirements of the front impact test, which was critical due to the change of the pendulum level according to ECE-R29 series 02 of ammendments.

With the numerical simulation of the safety tests the number of development loops could be reduced leading to a reduction of tests for different design variants. Thus development time can be substantially shortened.

To further improve this numerical method the following steps are planned:

- Using LS-DYNA Implicit for the quasi-static load cases roof load and rear wall load to realize a possible reduction of computing time and costs.
- Development of a suitable material model for the solid parts made of grey cast iron and cast aluminum to better predict damages such as incipient cracks, crack propagation or fracture.
- Introduction of a model to consider the influence of the pre-load in critical bolt connections. First promising investigations to this topic were accomplished in co-operation with DYNAMORE.

References

[1] Langwieder K.: Retrospektive Untersuchung über die innere Sicherheit von Lkw-Fahrerhäusern, FAT-Schriftenreihe Nr. 75, Frankfurt 1988

[2] Grandel J., Niewöhner W.; Untersuchungen zur inneren Sicherheit von Lkw-Fahrerhäusern, FAT-Schriftenreihe Nr. 115, Frankfurt 1994

[3] Paul du Bois: Crashworthiness Engineering with LS-DYNA, 2000

[4] LS-DYNA, Keyword User's Manual Volume 1, Version 960
Livermore Software Technology Corporation, March 2001