INTEGRATION OF VIRTUAL TESTING METHODS IN COMMERCIAL VEHICLE: VALIDATION OF SAFETY COMPONENTS FOR APPROVAL PURPOSE

Carine Teixeira
Manuel Laso
IDIADA Automotive Technology.

1. ABSTRACT

Within commercial vehicle development, the integration of virtual testing methodologies has become increasingly significant for safety components evaluation. The accuracy of virtual testing relies heavily on the correct replication of physical test into the virtual model, as for example to assign the realistic material properties from the structural components and following the boundary conditions coming in real conditions. In this presentation some regulations are used as examples. These regulations correspond to components for commercial vehicles, like rear underrun protection which corresponds to UN R58.03, the front underrun protection which corresponds to regulation UN R93.00, and the lateral protection corresponds to regulation UN R73.01
This study investigates the correlation between physical tests and numerical simulations approaches in the context of the safety component and homologation process. It explains how the simulations were performed, what technical aspects were considered for the result correlation with physical tests and which steps were followed for the methodology validation process.

2. INTRODUCTION

The application of virtual testing tools gives the possibility to manufacturers to simulate the same boundary conditions represented in physical test scenarios and gives options to different applications which can solve the difficulties that sometimes the physical tests cannot solve. This alternative method is technically and cost-efficient because it reduces the need for prototypes, it keeps all information of structural behaviour, so the possible countermeasures applied will be more effective, and finally shortens time to market. Technological advancements in Computer-Aided Engineering (CAE) in recent years have significantly contributed to the development of safer and higher-performing vehicles, while also enhancing flexibility for the representation of several types of scenarios.

However, CAE and homologation engineers face significant communication challenges due to their different technical languages and approaches to data interpretation. While CAE professionals use complex mathematical terminology and generate probabilistic results, homologation specialists operate with regulatory frameworks and require binary pass/fail decisions. To overcome this divergence, it is essential to establish a structured communicative framework based on three elements: nomenclatural standardization, development of simulation methodologies previously validated by homologation criteria, and creation of consensual evaluation metrics that efficiently translate the results into objective determinations of regulatory compliance, thus promoting a technical integration that meets the needs of both specialties.

But with these new technologies, facilities and benefits offered, the use of virtual testing methodologies in the homologation process is becoming increasingly popular.

The EU Regulation 2018/858 currently stablishes requirements for vehicle approval and market surveillance, ensuring safety, environmental protection, and technical harmonization across all vehicle categories, including cars, buses, trucks, and trailers [1]. Annex VIII of this regulation contains a list of regulatory acts linked to specific technical provisions where virtual tools can serve as alternatives to physical tests within the approval process. However, some regulations are not still included in this list. Probably in the future the new technological advancements and the new requirements evolving safety and efficiency needs will increasingly incorporate more regulations that will be allowed for the use of virtual testing in the vehicle homologation process.

3. REQUIREMENTS FOR USE OF VIRTUAL TESTING

After the methodology has been created by the manufacturer, the technical service evaluates first the quality of the model which has been created following a procedure (method) and after that they validate that the result from simulation gives realistic, conservative, and stable results. To assure this condition, a result comparability between physical and virtual need to be performed.

In the next picture the appendix 3 of annex VIII from EU Regulation 2018/858 shows a flowchart with two main columns: In the left-hand column it shows the validation process for virtual methodology and in the right hand it shows the approval process once the methodology has been validated [1].

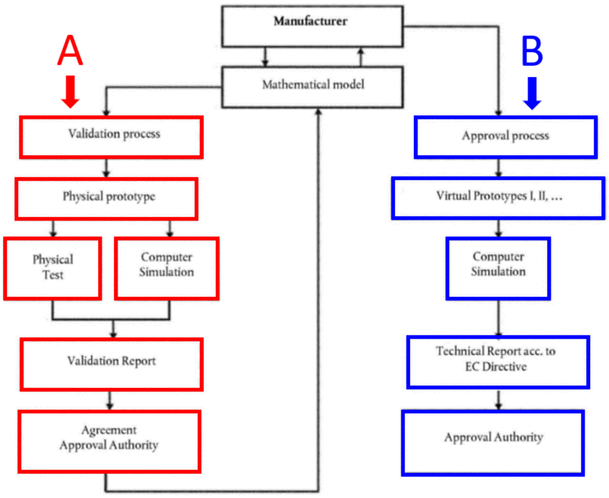


Figure 1. Flowchart of validation process from annex VIII, Regulation EU R2018/858

Validation process, which is represented in column A, left hand side, in the figure 1: The validation process shall be done independently for each regulation, and limited only for the specific paragraphs.

This process requires the next points:

1. Standardized methodology definition for virtual testing: This ensures consistency across simulations and simplifies the replication of the modelling construction by following the general guidelines. These guidelines shall consider all technical aspects.
2. Model quality procedure: The control of any mistake or misalignment in the model is essential to assure the quality. The use of a checklist tool allows to go through all technical aspects in the pre-processor phase and in the post-processor phase to assure all points are fulfilling the quality requirements.
3. Result comparability between the virtual and physical model will show if correlation has been reached in the way of being realistic results, more conservative and according to stable solutions.
4. Validation report: If the quality of the model is according with the methodology and the result comparability demonstrates that virtual results are realistic, conservative, and stable, then the technical service can deliver a validation report and from that point, the manufacturer is authorized to use simulation results in the vehicle approval process.

Approval process, which is represented in column B, right hand side, in the figure 1: The approval process is related to the validation of the vehicle type in which the manufacturer submit in the process the technical report with the test results, which in this case will be generated by the virtual tool that was previously validated.

The activity process shall be following the next steps:

1. Selection of the representative model: During the submission of the approval the manufacturer there is the interest of including the complete list of variants within the family type of vehicle in the same approval process. This represents in some cases big differences between these variants, and for that reason it is necessary to perform a technical evaluation to select the model variant that would represent the worst-case scenario, meaning that this model will give worst results.

   Worst case means the variant among a group of variants, least likely to withstand the requirements of the specific regulation [1]. If the selected worst-case model fulfils the technical requirements, then the technical service can directly consider the coverage of the rest of the model variants within the same approval and without the performance of more simulations.

2. Computer simulation and analysis: The manufacturer shall use the selected model (worst case) to simulate, and after simulation the result analysis shall demonstrate the fulfilment of the technical requirements.

3. Technical report: This report is delivered by the manufacturer and includes the complete data from the selected model object of the study. In this specific case a structural validation of a rear overrun protection device (RUPD) will be presented. This validation will follow the conditions of Regulation UN R58.03 considering the description of the structural parts, including specifications for each of materials, specifications for how it is connected to the chassis and specifications of load application regarding the category of the truck. The output will include the maximum longitudinal and vertical displacements, the plastic deformations and the section force on the bolts.

   The mathematical model (virtual model) shall be validated in comparison with the actual test conditions. To that effect, a physical test shall be conducted for the purposes of comparing the results obtained when using the mathematical model with the results of a physical test. Comparability of the test results shall be proven. A validation report shall be drafted by the manufacturer or by the technical service and submitted to the Type Approval Authority [2]

4. In addition, a time vs displacement graph can also be included to show in a visual manner the plausibility of the results.

5. Delivery of documentation for approval: The manufacturer will send to the technical service all information documentation, necessary statements, and technical justifications to implement to complete the approval process.

4. SIMULATION ACTIVITIES

The rear underrun protection device (RUPD) shall offer adequate resistance to forces applied parallel to the longitudinal axis of the vehicle. The test setting and sample preparation shall be according with the requirements of the regulation.

In the next image, see the representation of the component's attachment, in which the distance from the RUPD's main attachment point to the rigid test bench must not be less than 500 mm. If a diagonal strut is used to support the RUPD, this distance shall be measured between the foremost fixing point of the strut to the side rail structures and the rigid test bench [2].

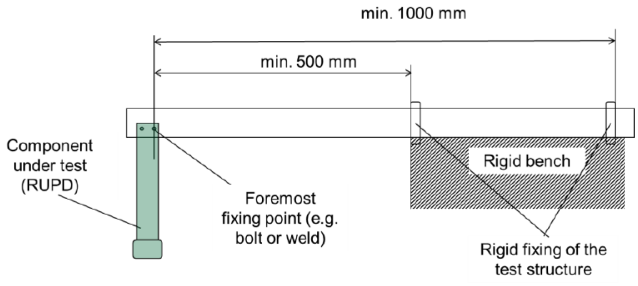


Figure 2. Figure 1 from Annex 5 of UN R58.03

The test conditions establish the application of a horizontal force of 180kN or the 85% of the force generated by the maximum mass of the vehicle, selecting the smaller value. The force shall be applied consecutively to two points situated symmetrically about the centre line of the device or of the vehicle whichever is applicable at a minimum distance apart of 700 mm and a maximum of 1000mm [2].

In addition, a horizontal force of 100 kN or 50 per cent of the force generated by the maximum mass of the vehicle, whichever is the lesser, shall be applied consecutively to two points located 300 ± 25 mm from the longitudinal planes tangential to the outer edges of the wheels on the rear axle or of the RUPD, if it exceeds the width of the rear axle, and to a third point located on the line joining these two points, in the median vertical plane of the vehicle [2].

In this study case, a generic structure of typical rear protection used in truck will be used. Below in figure 3 the sample structure of rear protection is represented. Two different tests will be used considering 2 different loads, as shown in Figure 4.

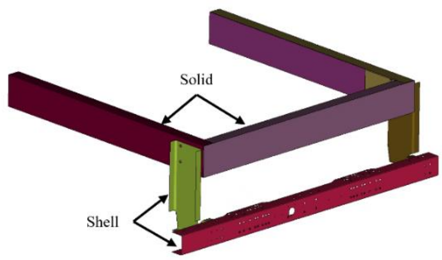


Figure 3. Representation of the test sample used

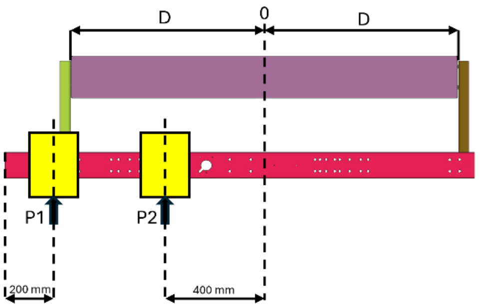


Figure 4. Representation of loads applied in this study

The study considered 2 identical models, one with bolted parts and one with welded parts. For the application of experimental loads, a specific approach was adopted: the bolted model was selected for the implementation of loads P1, while the welded model was employed for the application of load P2.

TEST & SIMULATION MATRIX		
	P1	P2
BOLTED ASSEMBLY	**80kN**	
WELDED ASSEMBLY		**65kN**

Figure 5. Load applied to bolted and welded samples

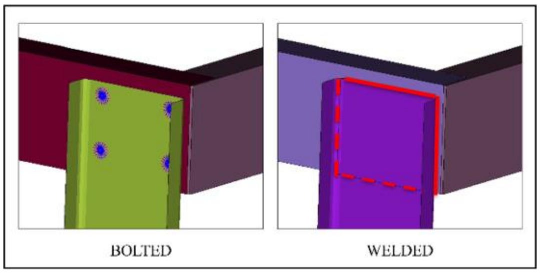


Figure 6. Difference in part connection between bolted and welded model

4.1-Model preparation

The parts were dimensioned considering the specific thickness of each component, a determining factor in selecting the applied mesh methodology: hexahedral solid elements for certain components and shell elements for others (as illustrated in figure 3). The discretization criterion implemented in both the models and weld regions maintained an average dimension of 5 mm, respecting pre-established limits between a maximum of 8 mm and a minimum of 2 mm. The current finite element configuration represents an optimized balance between results accuracy and computational efficiency, establishing an ideal compromise between solution fidelity and processing time required for the analysis. It was considered to establish a strategy to avoid the presence of triangular elements on the edges of the parts and in critical areas, thus preventing the false indications of high plasticity in regions where they would not occur, which could appear as numerical singularities. As a result of this methodological approach, the maximum limit of 5% of triangular elements was maintained throughout the structure of the finite element model.

Constraints were strategically implemented in the region highlighted in Figure 7. These include complete limitations of rotation and translation along all axes, precisely reproducing the conditions established during the physical test, in which the corresponding components were fixed to structural pillars. The configuration of constraints was designed to replicate the real conditions of the physical test, thus ensuring comparative validity between the simulation results and the test data obtained.

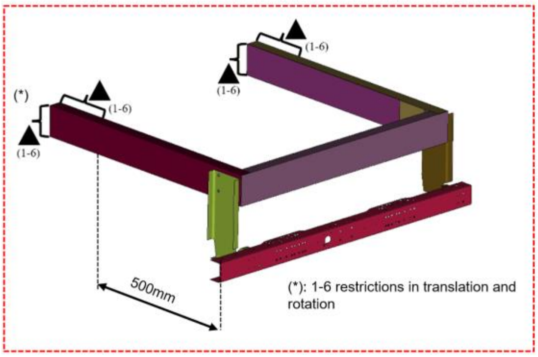


Figure 7. Constraints implemented in the model

The load was applied at three selected points, P1 and P2, highlighted in Figure 4, using an impactor with specific dimensions to assure a high stiffness. For the mesh of this load impactor, an average element criterion of 28 mm was implemented, establishing mesh parameters with a maximum limit of 30 mm and a minimum of 15 mm. A rigid element was created at the center of the impactor's external face, allowing the load to be applied directly to the central node of the impactor.

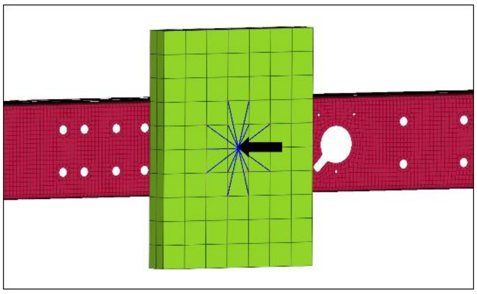


Figure 8. Impactor with the application of force

To avoid any inconvenience in the convergence of the calculation, a contact interface was developed between the impactor and the structural plate.

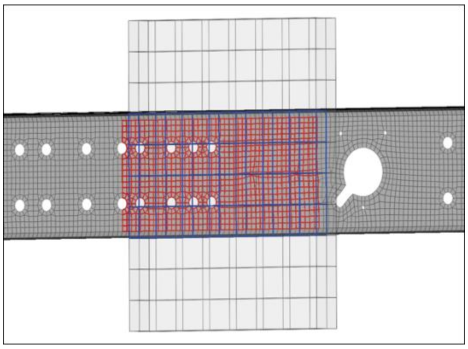


Figure 9. Representation of contact surface between component and impactor

This contact interface was implemented and repositioned on the plate surface, specifically adapting to each of the designated points P1 and P2, for force application. The configuration of this interface ensures efficient load transmission and accurately simulates the physical interaction between components, ensuring results that are representative of the actual test conditions.

The bolts were modeled as illustrated in Image 10, using an efficient simplification strategy. Rigid elements (identified in blue) have been strategically positioned in the holes of the parts intended for fixing with screws. Beam elements (marked in orange) were used to connect these rigid elements, thus creating a functional representation of the assembly.

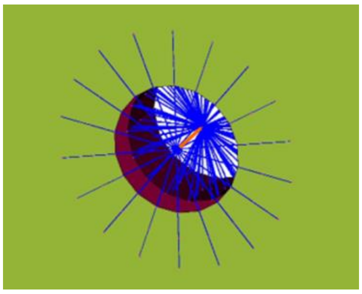


Figure 10. Representation of rigid body and beam element for bolt connection

This bolt assembly simplification model approach was adopted to optimize the computational time, while maintaining the accuracy of results compared to real test conditions.

For a quantitative analysis of the mechanical behavior of the bolt connection, the reaction forces of the represented beam elements were extracted, thus allowing a detailed assessment under the applied load conditions.

The modelling of welded joints used a simplified approach through structured line of elements between the brackets and chassis, as shown in figure 11. The methodology adopted aims to reproduce with greater fidelity the structural behaviour of welded joints, preserving fundamental mechanical characteristics such as local stiffness and stress distribution. This modelling technique provides a more realistic representation of the weld boundary conditions, resulting in a computational mechanical response significantly more correlated with the results obtained in physical validation tests.

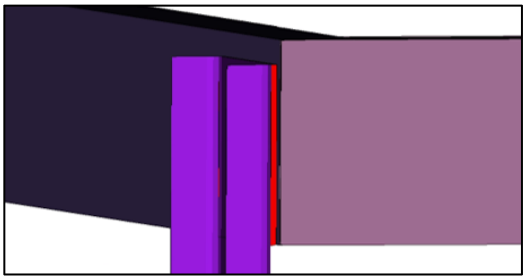


Figure 11. Representation of the welded connection

Precise material characterization is essential to correlate physical with virtual tests. After performing the laboratory tests, the real elastoplastic curves were obtained for each material type integrated in the component.

TRUE VALUES

Sample	Yield Strength (MPa)	Ultimate Strength (MPa)	Homogeneous Elongation (%)	Elongation at break (%)	Thickness
RPPCE_1	571	748	16,74%	25,27%	8,07
RPPCE_2	564	736	16,61%	24,91%	8,14
RPPCB_3	566	744	16,70%	25,09%	8,11
Average	567	743	16,69%	25,09%	8,11
Standard deviation	4	6	0,07%	0,18%	0,04

MATERAL DATA FOR ABAQUS AVERAGE VALUES	
Stress Mpa	Strain
567	0
569	0,010
577	0,020
596	0,030
621	0,040
636	0,050
654	0,060
670	0,070
681	0,080
693	0,090
702	0,100
710	0,110
717	0,120
725	0,130
731	0,140
737	0,150
741	0,160
743	0,170

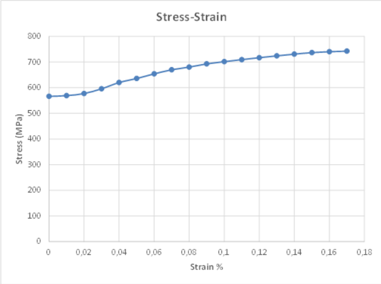


Figure 12. Material curve from tensile test

4.2-Simulation processing

For the execution of calculations and simulation processing, the *ABAQUS* software, an advanced platform specialized in finite element analyses, were used. The computational configuration was optimized with the allocation of 8 CPUs in parallel processing, maximizing solver efficiency and result accuracy.

The model was parameterized with time increments of 0.1 second for response data recording and storage, providing adequate resolution to capture transient phenomena in detail and ensure balance between analysis precision and the volume of generated data.

4.3- Post processing

To ensure a rigorous and systematic evaluation, the engineer configures the post-processing parameters in the simulation software to extract the relevant data required for compliance verification with Regulation UN R58. This step is fundamental for a comprehensive analysis of the component under the requested test conditions.

- Displacement Analysis

The determination of displacements constitutes an essential evaluation parameter. According to the criteria established by the Regulation UN R58.03.

Computational simulation allows quantification of displacements at all points of the component under test, identifying critical regions where excursions beyond established limits may occur.

- Plastic Deformation Assessment

The analysis of plastic strain is implemented to identify potential structural failure zones. The acceptance criterion establishes that the maximum plastic deformation in any region of the component must remain below the ultimate strength limit of the material.

This verification looks for:

-Identifying potential stress concentration zones

-Evaluating structural integrity under load application

- Analyses of forces in the bolted elements

For bolted assemblies, the extraction of sectional forces in the bolts is programmed to quantify:

-Axial force components (tension)

-Transversal force components (shear)

These data will serve as inputs for a subsequent specific analytical evaluation, in which each bolted element will be individually verified according to applicable bolted connection design VDI 2230 standard.

This structured post-processing methodology ensures obtaining all necessary parameters for a complete and reliable evaluation of the component regarding the requirements of regulation UN R58.03.

5. VIRTUAL TOOL CALIBRATION (COMPARISON)

To correlate the results obtained in the simulation with physical tests, several parameters had to be controlled during testing, such as the model construction, component fixation, force application points, and displacements. With these controlled parameters, the physical test results were obtained, and simulations were performed using the same parameters.

Study of first correlation P1:

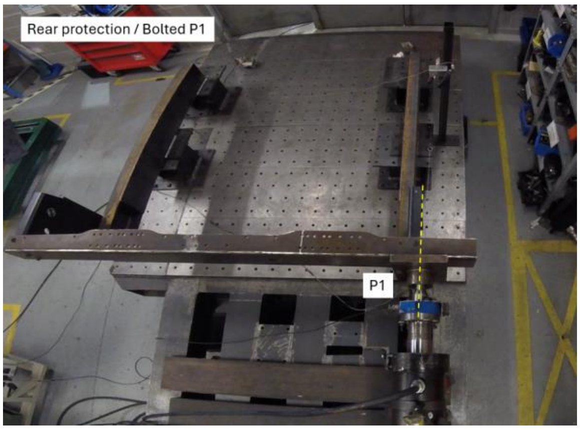


Figure 13. Component sample fixed in bench for P1 test

Analysing the physical tests of P1, performed on the bolted model, what could be observed was that when applying the force of 80kN in the direction of the plate, two bolts ruptured due to shear, as can be seen in the figures 14 and 15.

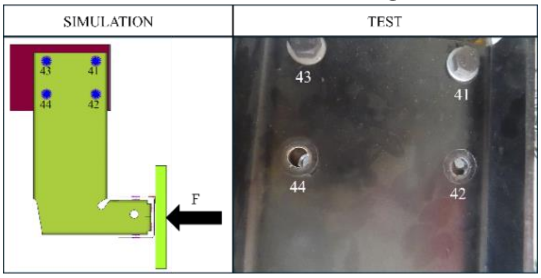


Figure 14. Position of the broken bolts

Figure 15. Broken bolts

As the simulation was simplified by using beams and rigids to represent the bolts, there was no way of seeing the broken *bolts. So, the reaction forces of these beams were taken and, in Figure 16, where the graph shows the reaction forces of* the four bolts on the side where the force was applied, the same two bolts that broke in the physical test had higher reaction forces than the other bolts.

In the graph of figure 16, the reaction force curves of the bolts are represented and the red lines represent the rupture, yield, and slip limits of the bolts in this model. The curves representing bolts 42 and 44 are above the rupture limit of the bolt, showing that these two bolts would indeed break. On the other hand, bolt 43 is slightly below the bolt's rupture limit and finally, bolt 41 is at the yield limit.

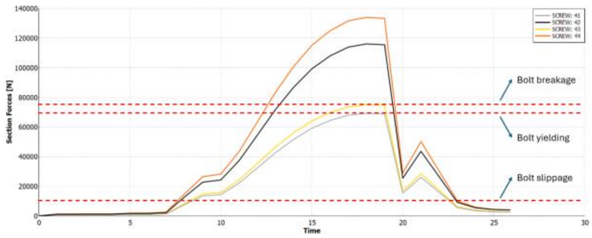


Figure 16. Graph of bolt reaction forces for P1

Study of second correlation P2:

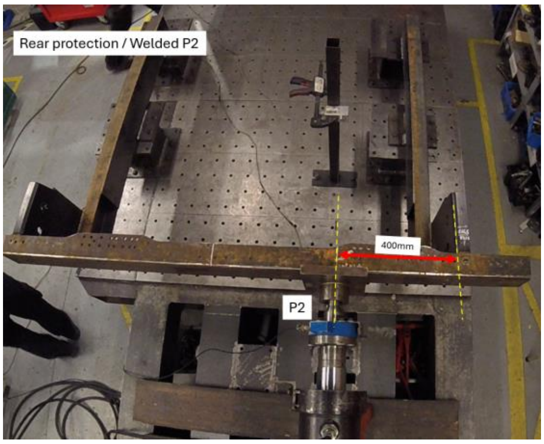
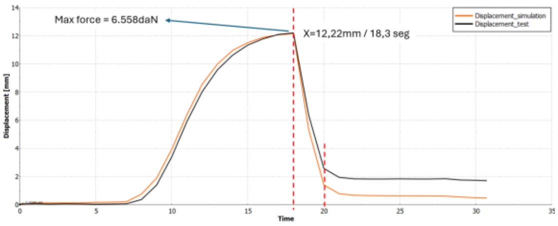


Figure 17. Component sample fixed for P2 test

Analysing the physical test of P2, carried out on the welded model, obtained the graph shown in figure 18 of displacement by time, extracted from the simulation, as well as the curve corresponding to the test results. The curves are practically the same, with some differences in some moments, with the same time value at the highest force application and the maximum displacement results with a small difference of 0.07 mm, where the simulation gave a value of 12.15 mm and the physical test 12.22 mm maximum displacement.



Force (N)	Time (seg)	Max displacement in test (mm)	Max displacement in simulation (mm)
6.558	18,3	12,22	12,15

Figure 18. Graph displacement vs Time for P2

In the plot comparison, there is a good correlation in maximum values between test and simulation. The slopes along the testing time are also identical. After the maximum value is reached the final part of the plot shows bigger differences in displacements with the test curve over the simulation curve. This is due to the dynamic effects of the hydraulic cylinder during the test.
Figure 19 shows the status of the component with elastic recovery after the physical test.

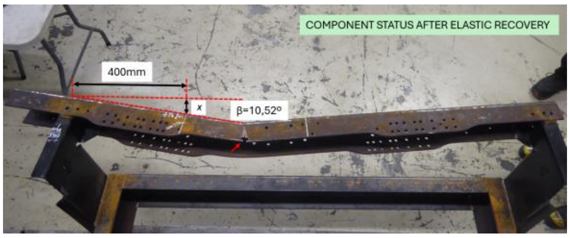


Figure 19. Component status after elastic recovery

In a position 400m long, the final displacement on the x-axis was 74.28mm, whereas in the simulation it was 74.42mm, with a small difference of 0.14mm in final displacement.

"X" value after elastic recovery	
Final distance after test (mm)	Final distance after simulation (mm)
74,28	74,42

Figure 20. Results component status after elastic recovery

Another aspect to note in figure 19 is that the plate bends at an angle of 10.52°, where a hole in the end of the plate breaks off after bending (figure 21).

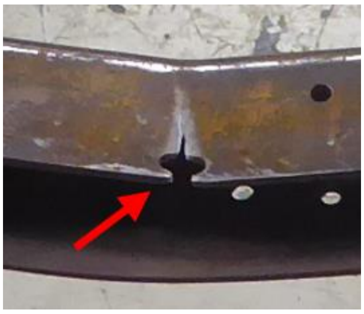


Figure 21. Broken hole after bending

6. MODEL QUALITY CONTROL

The control of the model uses the next quality check control for the input file, pre-processing, and for the output file, post-processing.

The pre-processing requirements check the next points:

- Element criteria selected for different components.
- Mesh quality control.
- Setting for intersections and penetrations between parts.
- Contact and frictions.
- Connection settings used for assembly parts.
- Setting of load and positions.
- Appropriate material models.
- Setting to define the output messages.
- Definition of frequency time for animation output (i.e. deformation, plastic strain maps, etc).

The post- processing requirements check the next points:

- Plausibility of requested signals.
- Plausibility of contact interactions: Visual animation checks with the verification of trajectories of stablished landmarks.

7. MODEL TRACEABILITY

Implementing robust data security of the model files used is crucial to address concerns about unauthorized modifications or access to the simulation model. This involves controlling access, secure storage, backup systems, and monitoring for any unauthorized changes. By safeguarding the integrity and security of the simulation model data, the risk of manipulation or tampering is minimized, providing reassurance to technical services.

The generation of a unique serial number to each approved simulation model enhances traceability and control. This input and output model control uses cryptographic algorithms such hash functions or digital signatures that ensures the integrity and authenticity. The technical services can use a verification tool to verify that the models were not manipulated compared to other simulation runs.

8. CONCLUSIONS

The development of safety components as rear underrun protection, front underrun protection and lateral protection have gained a big improvement with the use of virtual methods.

Correlation activities validate that finite element models accurately represent realistic conditions, establishing the proof justification for the creation of a fixed procedure to follow.

Virtual testing requires preliminary validation to ensure simulation tools produce realistic, conservative, and stable results before use in vehicle approval processes. This demands new procedures and technical skills from homologation engineers.

Document control and traceability protocols must be followed by technical services to prevent model manipulation after verification.

The adoption of virtual testing as an alternative to physical tests challenges homologation engineers to develop new technical competencies for evaluating simulation results, ultimately enabling proper technical validation of virtual tests.

9. REFERENCES

[1] Regulation (EU) 2018/858 of the European Parliament and of the Council of 30 May 2018 on the approval and market surveillance of motor vehicles and their trailers, and of systems, components and separate technical units intended for such vehicles, amending Regulations (EC) No 715/2007 and (EC) No 595/2009.

[2] Regulation No R58 of the Economic Commission for Europe of the United Nations (UN/ECE) Uniform provisions concerning the approval of vehicles rear underrun protection.

10. ABOUT THE AUTHORS

Carine Alves Teixeira, the author and presenter of this document, is a mechanical engineer with experience in CAE. She currently works as a durability engineer at the body performance department and has the role of coordinator of virtual testing activities for homologation at IDIADA Automotive Technology in Brazil.
Contact : carine.teixeira@idiada.com
LinkedIn: https://linkedin.com/in/carineteixeira

Manuel Laso Rodríguez, co-author of this document is a mechanical engineer specialized in the automotive industry, with experience in vehicle structure and functional systems. Currently, Manuel serves as a project manager for commercial vehicles at the Body Performance department of IDIADA Automotive Technology in Spain.
Contact : manuel.laso@idiada.com
LinkedIn: https://es.linkedin.com/in/mlaso