

Static and dynamic simulations for automotive interiors components using ABAQUS

Mauro Olivero, Vincenzo Puleo, Massimo Barbi,
Fabrizio Urbinati, Benedetta Peyron

Fiat Research Centre

Giancarlo Luciani, Alberto Novarese

Fiat Group Automobiles S.p.A.

Abstract: *Over the last years the car manufacturers have been working intensively to improve the quality of the interiors since it is extremely important to the customer's perception of the car.*

The numerical simulation can help the development of the interiors components design in order to guarantee better structural performances and to increase the quality of the products and therefore it is important to develop new calculation methodologies.

In this paper, we will present two methodologies developed to improve the quality and efficiency of dashboards.

The first one concerns the dynamic simulation of the passenger air bag cover opening: the key points of the activity are the characterization of a polymeric material with a high elongation at break, the development of a simplified model of the airbag's tear seams and the setup of a process that can be included into an optimization loop.

The second one concerns the simulation of the opening and the closure of the glove box: in this case the key point are the development of a functional model of the locking mechanism and of the interactions between the glove box and the various components that affect the opening and closing minimum load, like the rubber bumpers and the internal light switch.

Both the methodologies have been validated by means of a series of experimental tests and therefore it can be used during the design development process.

Keywords: *Air-bag, Collapse, Connectors, Damage, Dashboard, Fracture & Failure, Glove box, Mechanisms, Polymer.*

1. Introduction

In the last years, current market request to reduce the time and cost development. In order to achieve this result, virtual simulations play an important role since they allow to minimize the number of experimental tests.

The two numerical methodologies that we are going to show can help the development of the interiors components design in order to improve the structural performances and to increase the quality of dashboards.

The simulation of the passenger air bag cover opening is a dynamic simulation made with Abaqus/Explicit: a rigid impactor hits a portion of the dashboard containing the passenger air bag

module. In order to correctly evaluate the opening force of the cover bag it is necessary to implement a simplified model of the airbag's tear seams.

The simulation of the opening and the closure of the glove box is a static simulation made with Abaqus/Standard: in order to correctly evaluate the opening and closing force it is very important to implement a correct model of the locking mechanism and to simulate the interactions between the glove box and all the various components involved during the closure and the opening.

2. Cover bag collapse simulation

The starting point of this work is to understand the complexity related to the designing phase of interiors components and, in particular, the zone of the dashboard under which is placed the passenger airbag system.

During the development of the component, the designer has a lot of task to take into account:

- guarantee the correct opening of the cover under the pyrotechnical load coming from the airbag system;
- avoid tears during the deployment of the air bag (Figure 1);
- avoid sharp corners on the area interested by the head impact;
- avoid unexpected failure in the dashboard due to the airbag inflation (Figure 1).

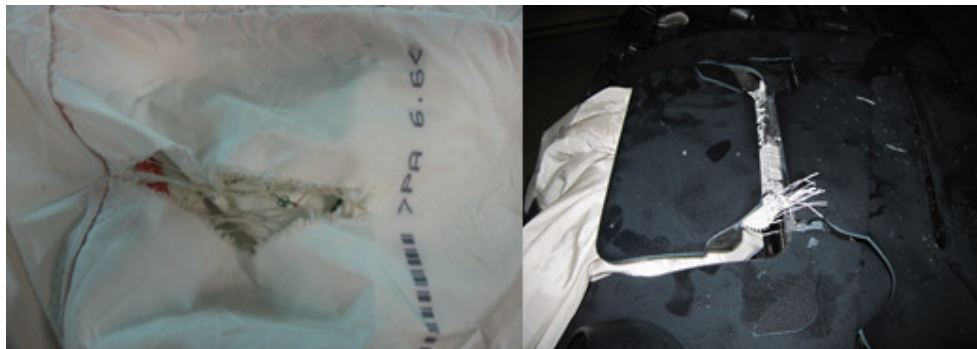


Figure 1. Examples of problems related to incorrect air bag deployment

The setup of this component is very complex, since there is a lot of technical requests that have to be fulfilled and the physical problem is guided by many designing parameters that are related one to each other by very complex relationships.

Up today, the development of the component is made by a huge number of experimental tests, made to calibrate the designing parameters, like the distance between the holes, the residual thickness and the shape of the opening line of the cover. This generates a big amount of work made in conjunction by the car maker and the supplier: this is why CRF and Fiat decided to develop a methodology to virtually simulate this kind of phenomena: once the methodology will be ready to be used into the designing phase of the cover bag, this could reduce the amount of experimental test to calibrate the component and reduce the development cost of the component itself.

2.1 Numerical model of the cover bag material

Polymeric materials show a very complex mechanical behavior that could be resumed in few point:

- they show a mechanical behavior which is strongly dependent from the temperature and from the strain rate;
- the elastic modulus is a function of strain rate and plastic strain;
- polymers have different hardening behavior in tension, compression and shear;
- the volume is not constant during the plastic deformation, since during the plastic deformation, polymers show a phenomena called crazing, which is basically the nucleation of voids inside the material that generate an increase of the volume during the deformation.

In order to calibrate our numerical model of the material, we started from experimental tensile test made at different value of strain rate and at different values of temperatures.

Another important issue that we included in our numerical simulation is the possibility to take into account of yielding criteria related with the pressure and the volumetric variation during the plastic deformation. We developed a numerical model of material which has the following characteristics:

- the yielding shows a linear variation with respect to the pressure;
- the plastic deformation is not isochoric;

That could be done by an optical measurement made during the tensile test: starting from the measurement of the transversal and the longitudinal strains on the specimen during the tensile load and by some considerations about the trough the thickness strain, we obtained the experimental curve of the volume variation over the strain.

Starting from this curve, we implemented our numerical model of the material in order to fit this behavior and we could fit quite well the experimental results: in Figure 2 we can see the comparison between experimental (orange) and the numerical test (blue).

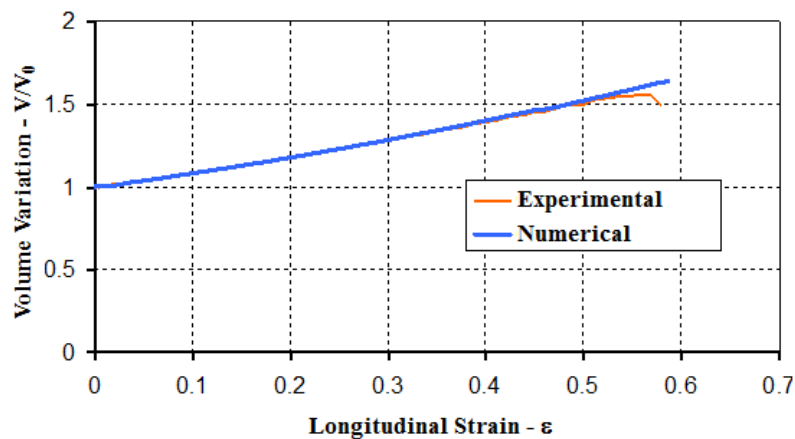


Figure 2. Volume variation (correlation between numerical and experimental results)

To complete the description of the numerical modeling of polymeric material we have to show the correlation of the stress – strain curve in the tensile test between numeric model and experimental test: in Figure 3 we can see that the behavior of the numerical model (red line) is very similar to the experimental test (blue line).

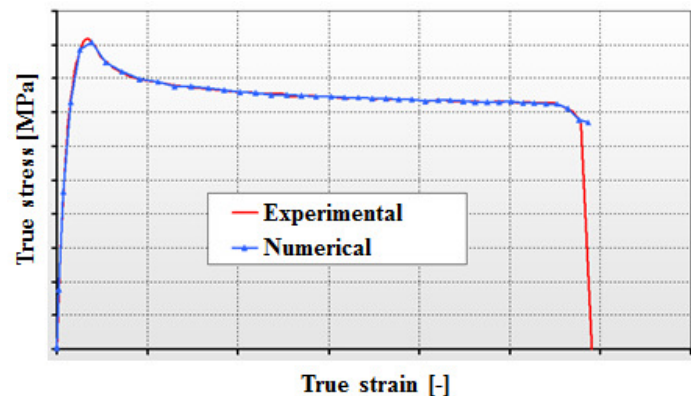


Figure 3. Tensile test (correlation between numerical and experimental results)

2.2 Characterization and modeling of airbag's tear seams

In Figure 4 there are some picture showing the process of creation of the micro holes in the cover bag: a laser is guided by a robot over rear part of the cover bag to create micro holes: this operation makes the section weaker to force the opening of the cover bag along the tear lines during the deployment. On the front part of the cover, a residual wall thickness is kept to hide the tear line to the eyes of the passenger; the parameter design are the diameter of the hole, the residual wall thickness (the hole penetration) and the distance between the holes.



Figure 4. Micro holes created by a laser machine

From a numerical point of view the problem is how to simulate in a simple way these micro holes in a FEM analysis, since the dimensions of the holes are so small that the discretization of the finite element should be so accurate to become computationally unmanageable.

The strategy is to simulate a small portion of the cover with a detailed model of the holes using the material model calibrated in the previous step: this kind of simulation is quite expensive in term of computational time and has to be reduce to a small part of the component but, at the end you can get information about the failure of the section in which are placed the micro holes. The same simulation is repeated with a simplified model with shell elements (which are more suitable for large scale model because they are easier to model and less expensive in terms of computational time): obviously, the zone of the hole has been modeled with a material that has to be calibrated, starting from the base material, to obtain the same behavior of the detailed model (Figure 5).

Since we have to find a way to scale the material of the simplified model in order to match the behavior of the detailed one, the main idea is to scale it according to the geometric parameters of the two sections.

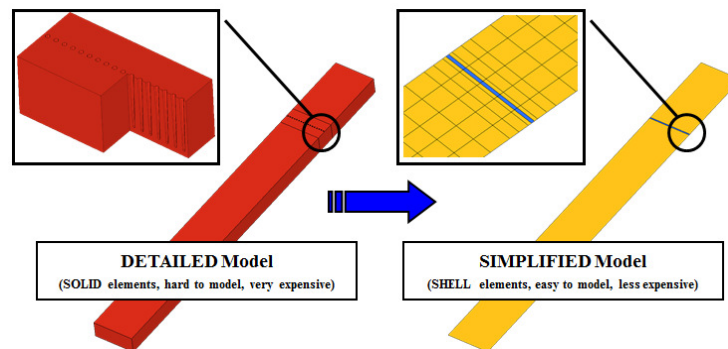


Figure 5. Calibration of a simplified shell model of the micro holes

In order to validate the simplified model, we compared the behavior of the numerical models under bending load (Figure 6): the maximum force is quite the same for the two model and the displacement at which the failure occurs is pretty the same.

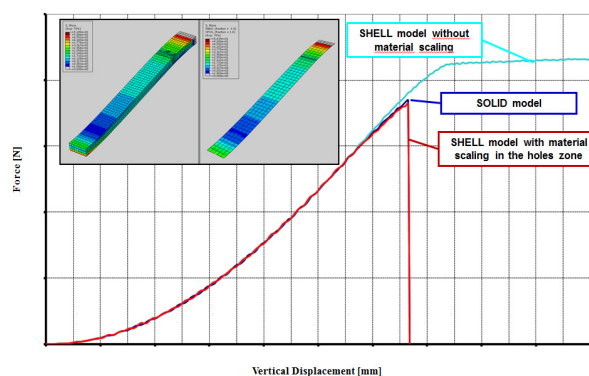


Figure 6. Comparison between detailed and simplified model

2.3 Simulation of cover bag opening

The simulation of the cover bag opening is a dynamic analysis made with Abaqus/Explicit: the model can be divided into three principal parts (Figure 7): there is a fixed part on which it will be fixed the portion of dashboard including the cover bag and a moving impactor that will hit the cover bag, causing the opening of the device.

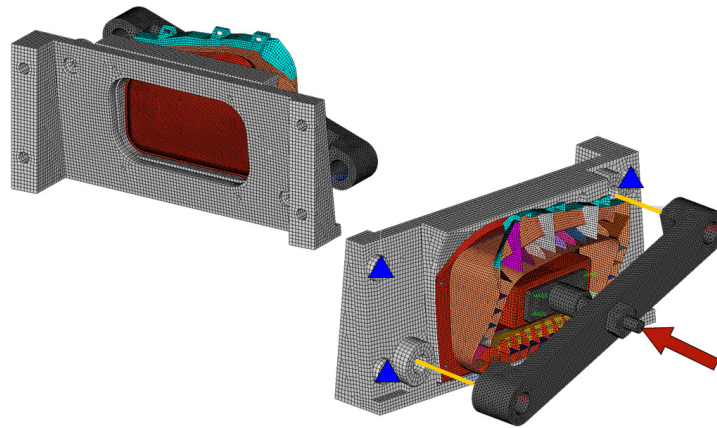


Figure 7. FEM model for simulation of cover bag opening

An important part of the model is the detail of the cover bag (Figure 8): in order to model the behavior of the tear lines on the cover, we implemented different material behaviors, according to the methodology explained in §2.2, for each part of the cover: the red part of the cover is associated with the base material, while the different colored lines represent the opening lines on which the section is reduced by the presence of the micro holes: each different colored line could be associated with a different scaled material, starting all from the same base material.

The differentiation between each portion of the tear line can be related to the different geometrical parameter that define the line, for example, each line can be different for the distance between the holes, for the residual wall thickness etc...

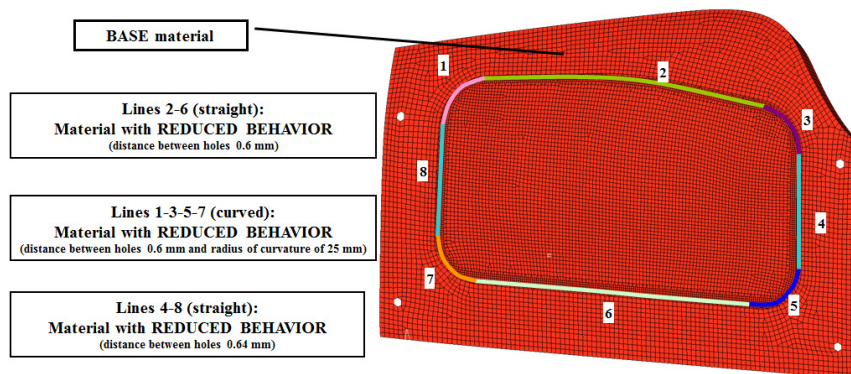


Figure 8. Detail of the material discretization on the cover bag

2.4 Results

Finally, we can show the comparison with the experimental test in order to validate the numerical methodology: the actuator of the real testing machine is a pneumatic cylinder; at the head of the cylinder is placed a force transducer and, after that we have the impactor that will hit the cover bag.

In Figure 9 we can see the comparison between experimental and numerical results: the blue curve represents the experimental result, while in red we have the numerical result.

The correlation is quite good: on the X axis we have the displacement of the impactor and we can see that the displacement on which the failure occurs in the numeric model causing the opening of the cover bag is really similar to the experimental result.

On the Y axis we have the opening force: here we can see that the numerical model shows a little overestimation of the opening force (about 10%), but we can say at the end that the model is predictive of the phenomena and could be used to help the designers in the development phase of a new dashboard.

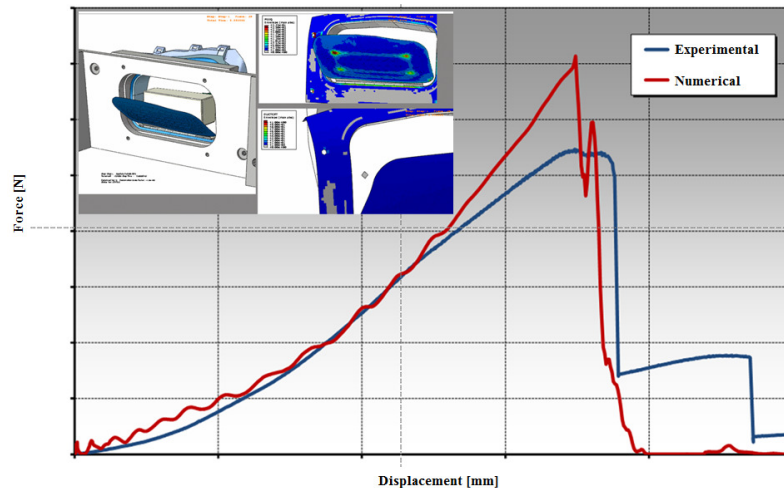


Figure 9. Comparison between experimental and numerical results

Certainly, the model could also be improved and we are working on it in order to reduce the distance between experimental and numerical results: for example we are working on the possibility of take into account for the different behavior in compression and shear and we are also thinking about the possibility of taking into account of the molding process: in fact, we usually consider this kind of polymers like a isotropic materials but the molding process could create big differences in the mechanical behavior of a polymeric component, depending on the flow direction during molding process.

3. Glove box opening and closure simulation

The second part of this work concerns the design of glove boxes. During the development of this component, the designer has to take into account for a lot of different tasks:

- guarantee the correct opening and closure of the device;
- limit the opening and closing forces under a certain threshold;
- avoid unexpected gaps that can reduce the quality perception of the dashboard in the zone of the glove box;
- guarantee all the structural performances in terms of stiffness, strength and safety behavior.

Sometimes it happens that, during the designing phase, there could be some problems related to the closure of the glove box, since in some cases the closure cannot be reached correctly. The designing parameters involved in this particular are different: the stiffness of the entire module is important, but also the behavior of the springs and the configuration of the locking device play an important role.

Up today, the experimental tests are very important to help the designer to develop the component, but the numerical simulation could reduce the amount of experimental tests during the designing phase, therefore CRF and Fiat decided to develop a methodology to simulate the opening and closure of glove boxes.

3.1 Finite element model of the glove box

The simulation of the closure and opening of the glove box is a static analysis made with Abaqus/Standard: starting from a complete model of the dashboard, we created a reduced model including the glove box module, the rubber bumpers and the light switch. All the possible contacts between the parts were considered.

In addition, we also included the model of the locking device: depending on the car segment, it can be a single (Figure 10) or a double latch closure device (Figure 11).

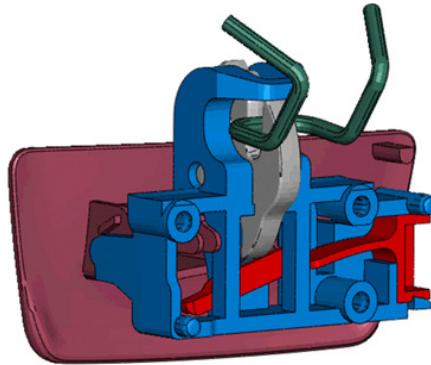


Figure 10. FEM model of a single latch closure device

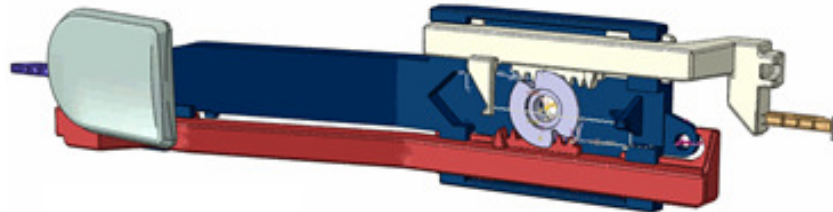


Figure 11. FEM model of a double latch closure device

The modeling of the locking device is very important to achieve an accurate evaluation of the closing and opening forces of the glove box: both the two types of device are a mixture of deformable, rigid and connector element: all the parts involved in contact were modeled like deformable parts; hinges, springs, bumpers and mechanisms (like the double rack and pinion in Figure 11) were modeled with connector elements (CONN3D2); all the remaining parts were defined as rigid.

3.2 Closure simulation

Starting from the glove box model described in §3.1, to complete the closure simulation we first rotated the door with a rigid rotation around the hinges removing the contact on the locking device.

Once the minimum opening condition were reached, we started pushing on the door with a pushing device until the complete closure of the glove box is reached (Figure 12): during the simulation, the reaction force on the pushing device is recorded and therefore it is possible to see the different contribution of the part in contact (latch, rubber bumpers, light switch) to the overall closing force.

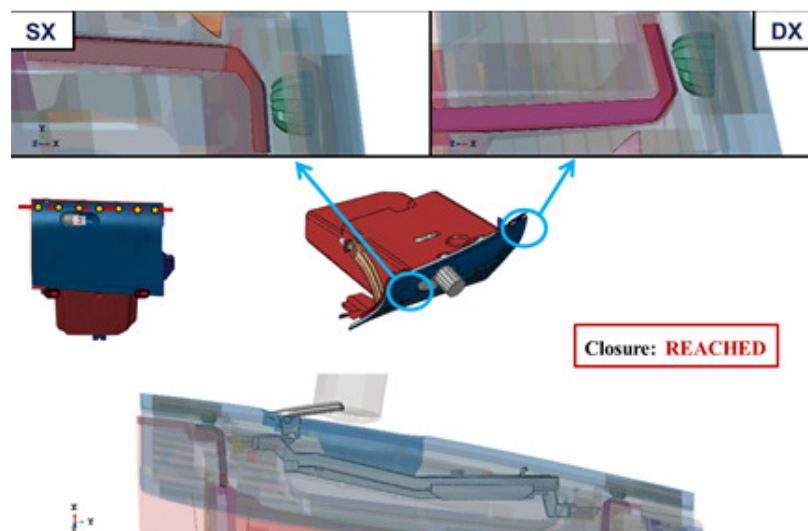


Figure 12. Closure simulation on double latch glove box

The simulation were repeated putting the pushing device in difference position, in order to evaluate if the closure could be reached in every position.

3.3 Opening simulation

Starting from the closure condition of the glove box described in §3.2, we implemented the model for the opening simulation by connecting a connector element to the handle and then by using a ***CONNECTOR MOTION** to pull the handle until the opening is reached (Figure 13).

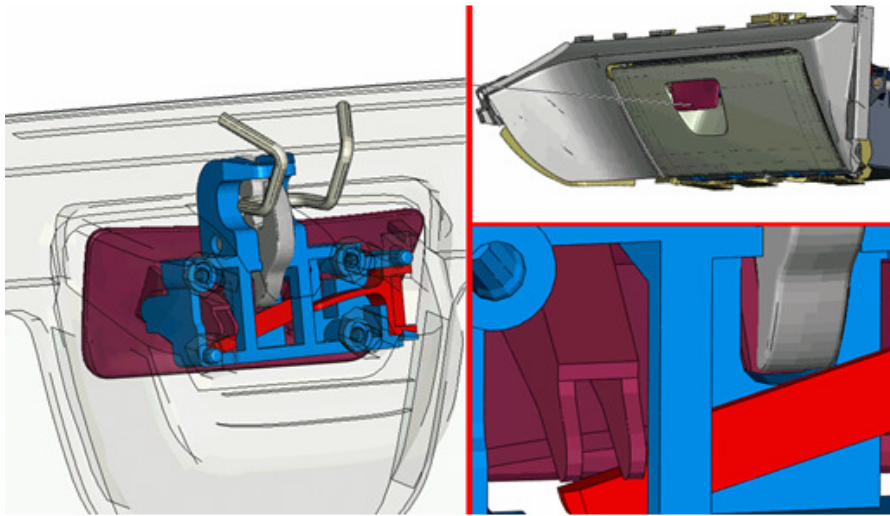


Figure 13. Opening simulation on single latch glove box

During the simulation, the internal forces of the connector element are stored and therefore it is possible to see the different contribution of the parts (latch, rubber bumpers, light switch) to the overall opening force.

3.4 Results

In order to validate the methodology, we have to compare the numerical results with the experimental test.

For the closure, the experimental test was made by putting the dashboard on a fixed support and trying to close the glove box by pushing on seven different positions (Figure 14): for each position, the relative closing force was recorded.



Figure 14. Glove box closure experimental test

The same type of analysis was repeated by the numerical simulation and the comparison between the results is shown in Figure 15: the correlation is very good from point 1 to 6. On point 7, it was not possible to reach the closure of the glove box during the experimental test, while in the numerical test we couldn't reach the convergence due the excessive deformation of the right rubber bumper.

Also if the comparison on point 7 available, the numerical closing force at the end of the simulation (if convergence could be reached) should be very high and that could generate a warning on the real possibility of closing the glove box by pushing on this point.

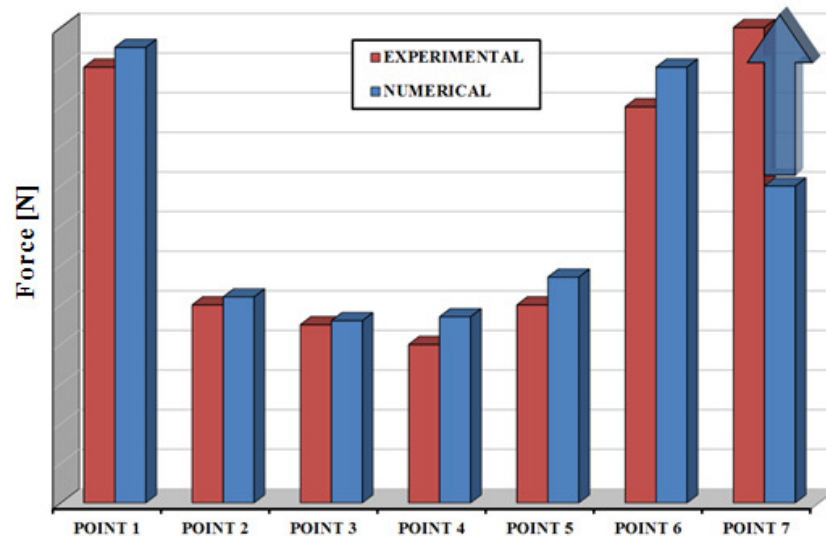


Figure 15. Closure (comparison between numerical and experimental results)

For the opening, the experimental test was made by connecting a wire to the handle and then by pulling the wire until the opening is reached.

In Figure 16 we have a plot of the reaction force on the handle during the displacement of the pulling point: the numerical opening force is quite similar to the result of the experimental test. The numerical opening force underestimates the experimental value but the reason can be related to the fact that on the opening device there is a stopper that limits the rotation of the handle; this device starts to operate immediately after the geometrical opening condition and therefore even a little additional rotation can generate a big increase on the recorded opening force.

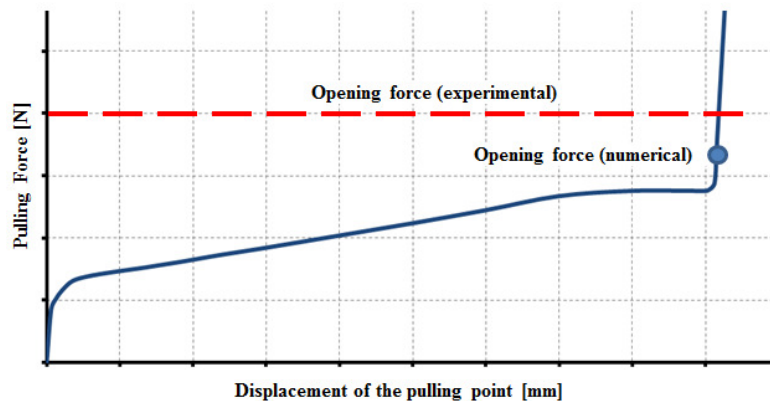


Figure 16. Opening (comparison between numerical and experimental results)

4. Conclusions

In recent years, a big effort has been made to increase the capability of simulating interiors components. Fundamentals parts of the methods developed are the numerical characterization of polymeric materials, the development of criteria for failure of element during dynamic simulations and the capability to model complex mechanisms to simulate locking devices.

Comparisons between experimental tests and numerical results have been done and the correlation is good: the new methodologies can be used into the designing phase of components and this could reduce the amount of experimental and therefore the development cost of the components.

5. References

1. Abaqus User's manual 2011
2. Spingler G., Drazetic P., Markiewicz E., "Dynamic Characterization of Polymers to Improve Numerical Simulations for Passive Safety", International Journal of Crashworthiness Vol. X n°1, 2005
3. Trantina G., Nimmer R., "Structural Analysis of Thermoplastic Components", MacGraw-Hill International Editions, 1994