Design and drop test simulation of a helicopter skid landing gear with Abaqus/CAE

Xavier Élie-dit-Cosaque[1], Augustin Gakwaya[1], Julie Lévesque[1]

1 Département de génie mécanique, Université Laval, Québec, Canada

Abstract: The aim of this paper is to study the impact of the manufacturing process of a skid landing gear on its performance during drop test simulation. A complete methodology is proposed to implement it on Abaqus. When a new landing gear design is to be designed, it has to be validated by experimental drop tests, and has to demonstrate the same overall impact resistance and energy absorption capability as well as being of similar or reduced weight. As this approach is rather expensive, an analytical approach validated experimentally has been used and yielded a complete methodology for drop tests simulation using dynamic explicit finite element codes. For more accurate results, conventional shell elements have been used because of small tube thickness. However if contact effect and through-thickness behavior are crucial to the analysis, this is not always the best approach. Hence we went a step further by simulating drop test using reduced integration stabilized shell elements. Their performance is assessed by considering the complete material and loading history of the tube, resulting from the multistage cross beam bending process. The drop test simulations are used to assess the structural integrity of the product under severe operating conditions. Results obtained using Abaqus explicit software show that the fabrication process leads to a non negligible amount of plastic deformation.

Keywords: Aircraft, Plasticity, Impact, Shell Structures, Tube Bending.

1. Introduction

A drop test simulation is very expensive and difficult to implement numerically if the complete material history involved in all designs and manufacturing phase is to be accounted for. For that reason, some researchers have decided to use the rigid/flexible multibody dynamic code such as ADAMS to simulate the drop test of a skid landing gear (Bianchi, 2001). An assembly of rigid elements was used in the simulations which were compared to the elastic finite element analysis. They showed that the flexible body model presents good results but the behavior law of the landing gear has to stay within the elastic domain. Other researchers decided to investigate on reducing the computational time using a hollow rectangular beam element with Ls-Dyna. This used element appeared to be time effective and relatively accurate compared to experimental results. However, it does not provide enough integration point through the thickness and hence not enough precision (Cheng-Ho Tho, 2003). However, they have developed a complete methodology to simulate drop tests and to compare them with experimental results. But these works did not take into account the impact of manufacturing process on the drop test results. For example, some other researchers studied analytically the impact of bending process on subsequent in-service conditions of the part. In fact the bending of a tube consumes a certain amount of its ductility, thus reducing its formability in subsequent operations (Xi Wang, 2002). A more accurate study of these effects with a finite element model for drop test simulation is still to be done. In the bending process, like in all forming processes by plastic deformation of the parts, the springback problem has to be taken into account in order to obtain the desired shape and part properties after the forming process (Zhan, 2006).

2. Element overview

2.1 Element choice

The part studied here was a simplified skid landing gear cross beam. This tubular part has relatively large dimensions compared to its thickness as seen in figure 1.

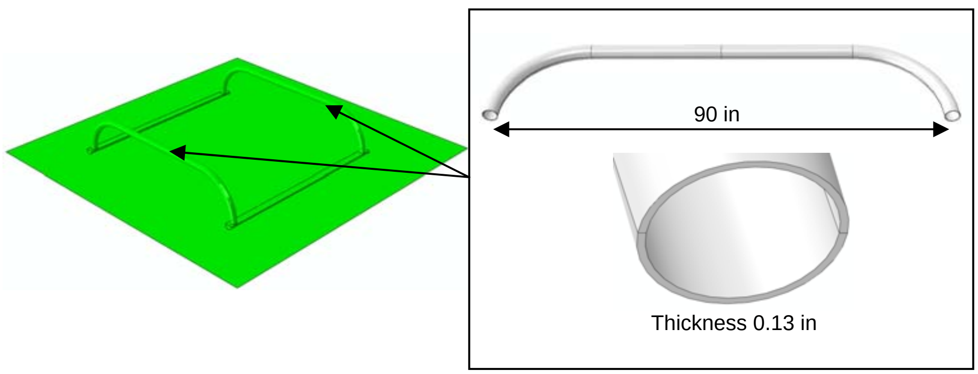

Figure 1. Simplified model

It has been decided to use only one element through the thickness in order to avoid a mesh with a large number of elements and thus to reduce the computational time. A convergence study was previously made to find an acceptable C3D8R element mesh size by analyzing the center of gravity vertical displacement (see figure 2). The efficiencies of Abaqus solid element C3D8R and of Abaqus continuum shell element SC8R were then tested using the specification of one element through the thickness for the simple model (see figure 1) when compared to the one described in section 4.1 using a different aluminum alloy and a different initial vertical speed. Since no experimental data were available, the results (see figure 3) were compared with those obtained with the Ls-Dyna code using similar element formulation.

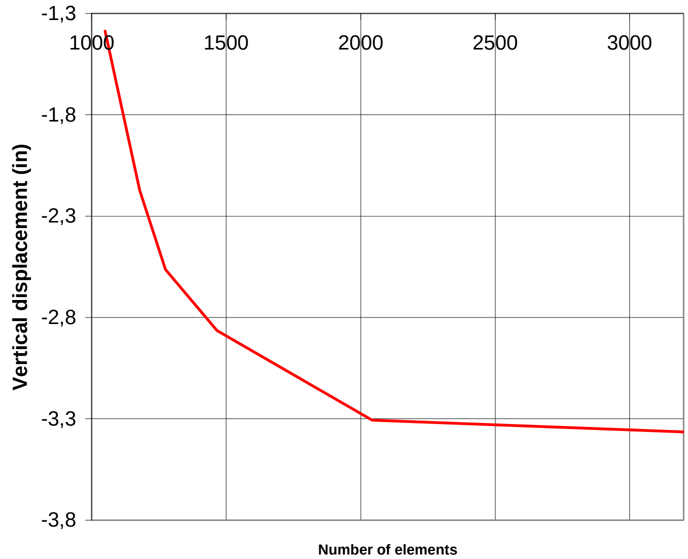

Figure 2. Center of gravity vertical displacement related to mesh elements number

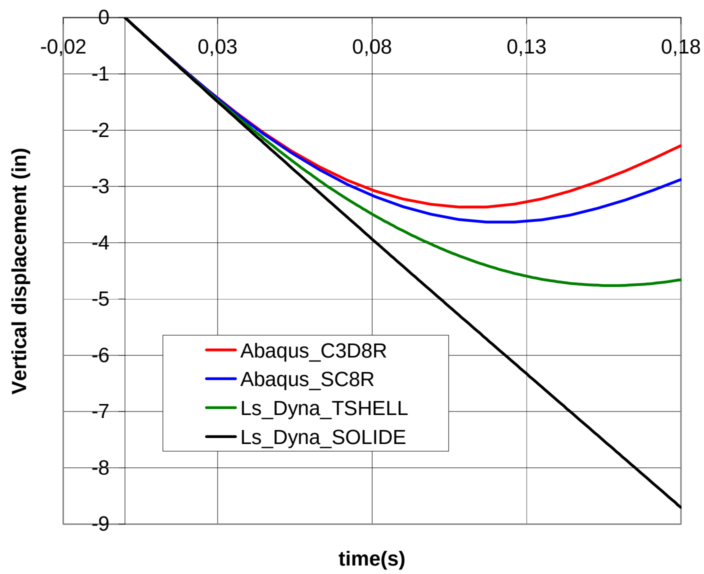

Figure 3. Center of gravity vertical displacement for the simplified model

From figure 3, it can be deduced that the Abaqus SC8R element give results close to those obtained with the Ls-Dyna Tshell element using the same converged mesh. Besides, the Abaqus C3D8R element shows a stiffer response as compared to the shell elements. However the Ls-Dyna solid element response is not correct and produce completely distorted regions. Due to close agreement between the two shell element responses, the Abaqus SC8R element was used in the subsequent bending and drop test simulations.

2.2 Element description (SC8R)

The 8-node linear continuum shell element SC8R, with reduced integration and hourglass control (see figure 4), and with a 3-point Gauss integration through the thickness to obtain good results accuracy was used in this study. The element thickness is determined by its connectivity. That is why this connectivity has to be verified after meshing. This element is available in Abaqus/ Explicit for improved computational efficiency.

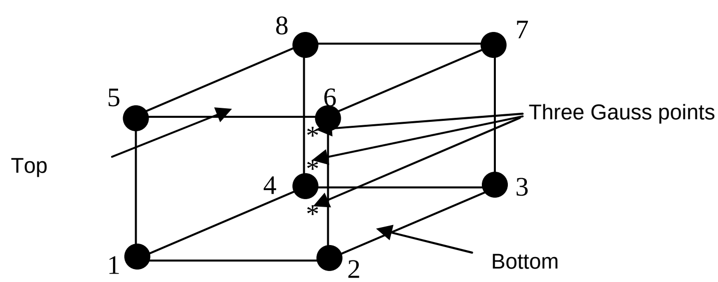

Figure 4. SC8R element

3. Finite element modeling for bending

3.1 Bending and springback description

The bending process that transforms a straight tube into a bent cross beam comprises the following steps:

- First the left side of the straight tube is bent using an explicit strategy;
- This followed by a springback simulation using an implicit strategy;
- Then bending of the right side of the pre bent tube is bent again using an explicit strategy
- And a final springback simulation is done using an implicit strategy.

The Abaqus model takes into account the relevant features of the real bending machine (figure 5).

Figure 5. Bending machine

The complete load history was recorded during the process, so the induced plastic deformation and the corresponding change in the mechanical properties (yield stress) are considered for the drop tests. The main components modeled in this bending process were a straight tube, a ram die and two wing dies. The tube cross section has a constant thickness of 0.13 in. The tube was meshed with SC8R elements with only one element through the thickness. An elasto-plastic constitutive law of an aerospace aluminum alloy was used for the tube. The ram die was considered as a rigid body. The wing dies were considered as deformable parts with a high modulus of elasticity. In term of mesh, the ram die was modeled with R3D4 discrete rigid elements and the wing dies were modeled with C3D8R elements. A force applied on the wing dies allowed a permanent contact with the tube when the ram die was displaced forward. Surface to surface contact was used by defining a ***contact pair** between interacting bodies, with a friction coefficient of 0.2 for the tangential behavior. When the tube is unloaded from the dies, springback calculation is launched during a calculation time of one second. A node of the tube was fixed far from the plastification zone to prevent rigid body modes occurrences.

3.2 Bending results

At the beginning of each bending simulation, the parts were positioned as seen in figure 6.

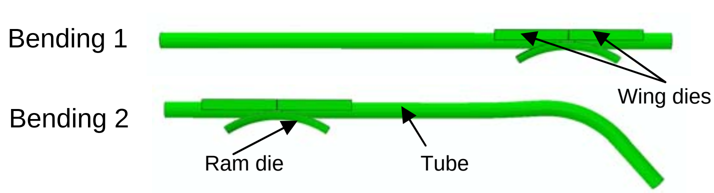

Figure 6. Parts positioning for bending simulations at time t=0s

In order to better see the obtained equivalent Von Mises stress and equivalent plastic strain fields, the tooling devises are removed from the display of figure 6 and the results are shown in figure 7 and in figure 8 respectively.

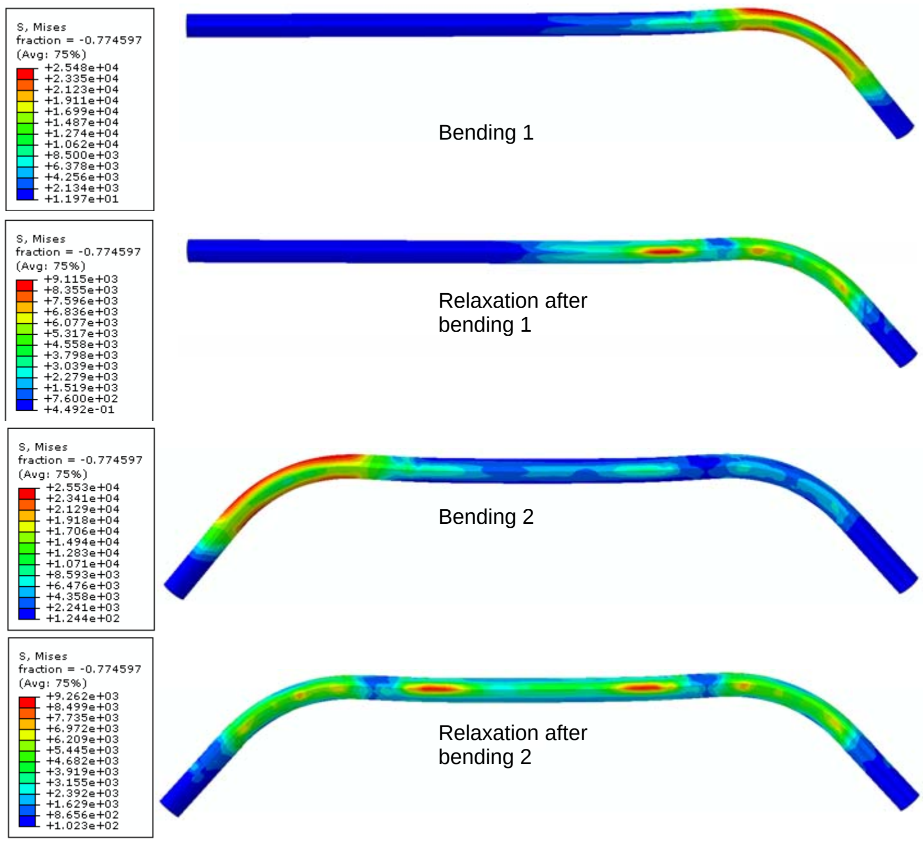

Figure 7. Bending results, equivalent Von Mises stress

The maximum residual stresses and plastic strains occur in the bent portion of the final cross beam. During this process, approximately plastic strain of 8% was achieved. As the allowed maximum plastic strain for the material is about 20%, a residual margin of 12% hence still exists for the drop test simulation with this particular material.

After bending, the equivalent plastic strain distribution following a certain path along the cross beam longitudinal direction is presented in figure 9. Theoretically, the results should be symmetric if the two bends were applied simultaneously; however this not the case since the loading process is not symmetric due to a sequential bend forming. However only weak asymmetries are observed.

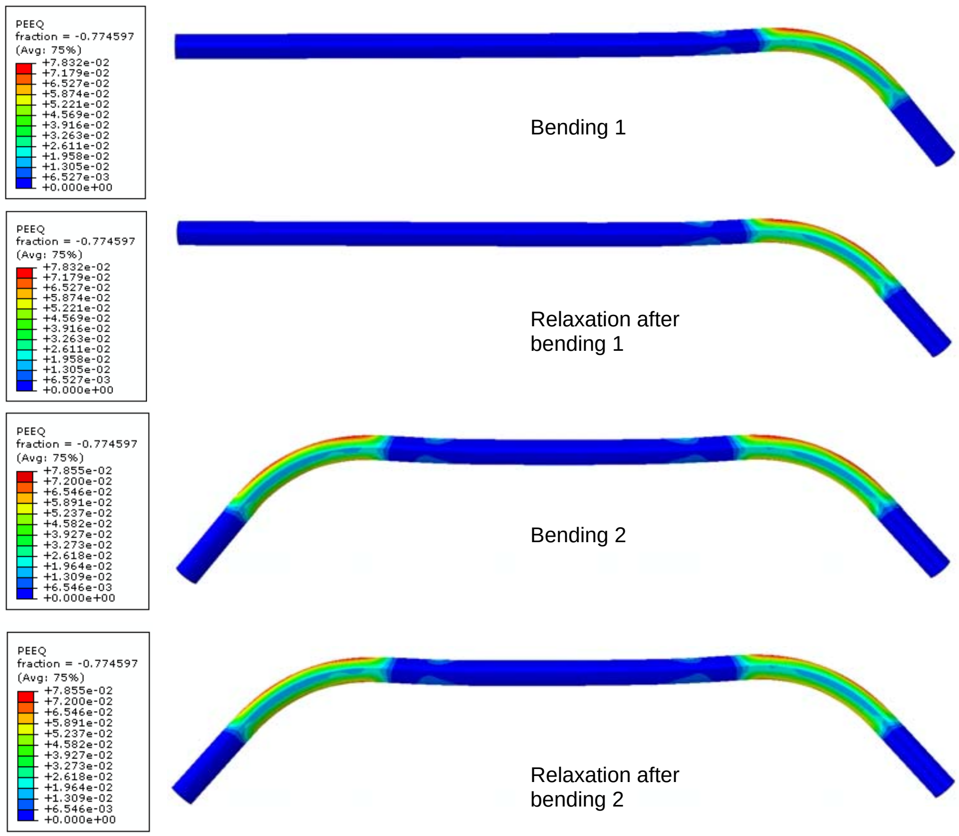

Figure 8. Bending results, equivalent plastic strain

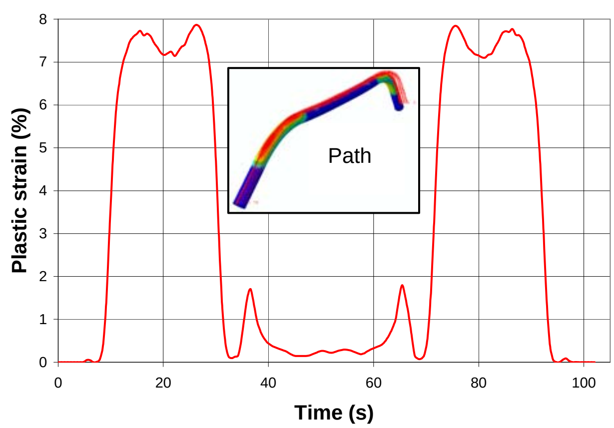

Figure 9. Equivalent plastic strain along the cross beam after bending

4. Finite element modeling for dropt test simulation

4.1 Drop test description

The drop test simulation was performed using an explicit strategy. It was done on a generic skid landing gear that reproduces the phenomenon observed on a real one. The main modeled components for this landing gear drop test simulation were two cross beams, two skids, four fuselage attachments, four skid attachments and the ground (see figure 10 and Cheng-Ho Tho, 2003).

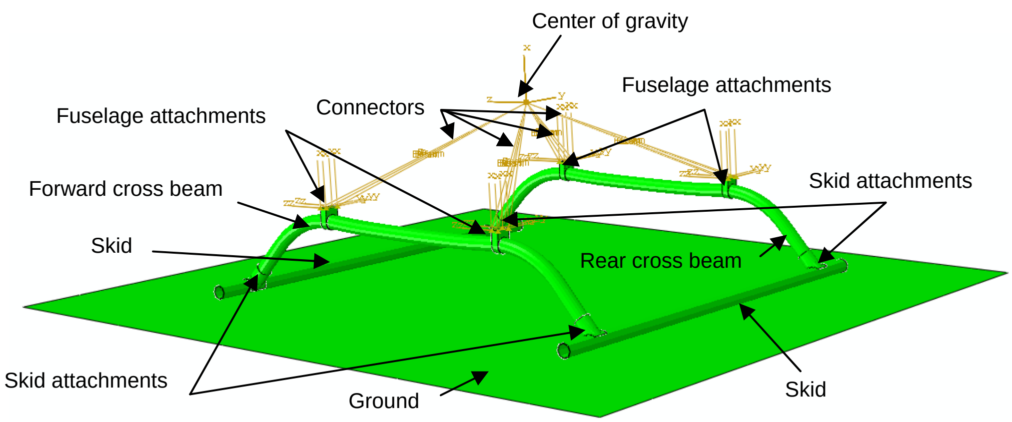

Figure 10. Drop test model

The forward and the rear cross beams were geometrically identical. They were provided from the previous tube bending and springback simulations, so their cross section was theoretically constant except where defaults appeared after bending. Normally a heat treatment is performed on the bent tube in order to obtain final material properties required in for final landing gear assembly before drop test. In our case, the material was not heat treated between the end of bending step and the drop test. Therefore the skid landing gear is more flexible than the real one and will not able to absorb the same amount of energy. Since the aim of this paper is not to compare numerical results to experiment ones but rather to demonstrate a simulation methodology, this discrepancy is not important at this step of development. However, the issue of studying the influence of material history will be assessed. Two types of analysis of the forward cross beam were performed, one taking into account the material history by using Abaqus ***predefined field** option and the other without the ***predefined field** option. The cross beams were modeled with SC8R elements and with a circular cross section and elastic-plastic material model was used for the employed aluminum alloy. The fuselage, the left and right skids and the skid attachments were modeled with C3D8 elements and considered as deformable bodies with a high modulus of elasticity. The fuselage attachments are add-ons that created contacts with the cross beams and allowed to record reaction forces between the fuselage and the skid landing gear. The skid attachments were used to link the forward and the rear cross beams to the skids. As no data are required from them, the keyword ***tie** was used to link the skid attachments to the other parts in contact. The ground was considered as a rigid horizontal surface with S4R shell elements. All degrees of freedom of the ground were set to zero. Surface to surface contact was used by defining a ***contact pair** between interacting bodies, with penalty contact algorithm for the mechanical constraint formulation. The friction model used decreases exponentially from the static friction coefficient to the dynamic friction coefficient and is written (see figure 11)

$$\mu = \mu_k + (\mu_S - \mu_k)e^{-d_c \dot{\gamma}_{eq}}$$

Where μ_k is the kinetic friction coefficient, μ_s is the static friction coefficient, d_c is a user-defined decay coefficient and $\dot{\gamma}_{eq}$ is the slip rate (Abaqus/CAE user's manual, 2007).

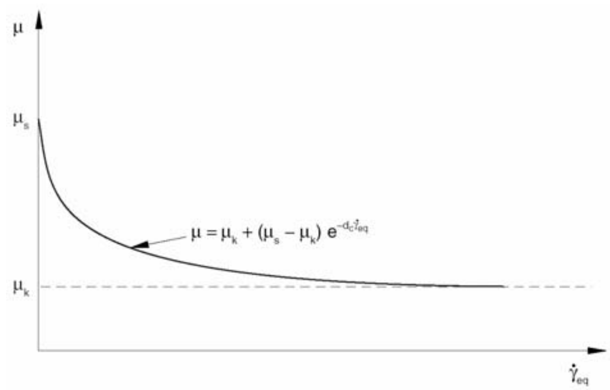

Figure 11. Exponential decay friction model (Abaqus/CAE user's manual, 2007).

A coefficient of friction equal to 0.5 was used for the static friction while a value of 0.35 was used for the dynamic friction and an exponential decay coefficient of 0.05 was considered.

In order to simulate the effect of fuselage, the center of gravity of the skid landing gear had a mass and inertia. Moreover, this center of gravity was linked to the fuselage attachment with connectors so that it could transmit its motion to the skid landing gear. Moreover an upward force of 5 000 lbf representing the lift due to the engine was applied at the center of gravity as well as a downward force of 5 000 lbf representing the weight of the helicopter. Then an initial sink speed of 50 in/s was applied to the whole model including the center of gravity. Lift and weight were supposed to entirely compensate each other so that the initial speed remained constant before impact.

4.2 Drop test results

The vertical reactions on the fuselage attachments are presented here using a **Butterworth filter** with a **cutoff frequency** of 10Hz to limit noise in the results. Figure 12 shows the reaction forces on the forward cross beam. Left and right results were not exactly the same due to the mesh asymmetry. Moreover, this asymmetry is accentuated for the model which considered the forward cross beam load history due to the existence of asymmetric residual stress strain previously mentioned. But this difference was mainly due to the applied inertia on the helicopter center of gravity. Besides, the maximum vertical reaction forces were higher when material history was considered.

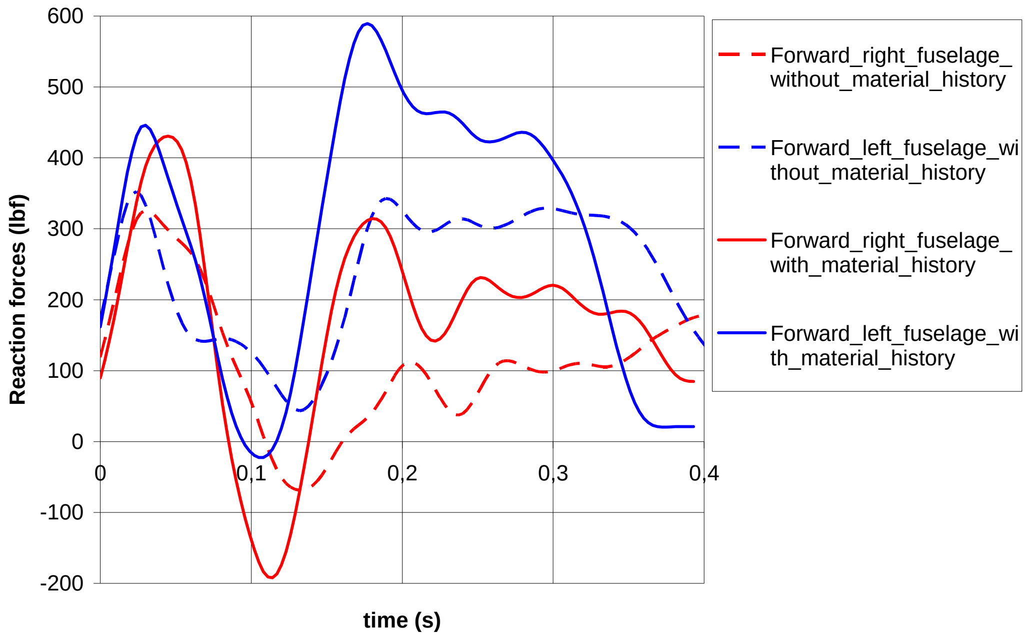

Figure 12. Vertical reactions, forward attachments

Figure 13 shows the reaction forces on the rear cross beam. The same observations can be made as for the reaction forces on the forward cross beam, except the fact that the related maximum vertical reaction forces are higher when material history is not considered. Moreover, these reactions forces seem to be more similar at the rear of the helicopter for the two models than at the front.

These last observations are emphasized on figure 14 where the resultant of the left and right reaction forces is applied to each cross beams. It is shown that the rear part of the helicopter has to resist to approximately the same vertical force whether the forward cross beam material history is considered or not, while this is not the case for the front part. The front region has to resist to a higher amount of vertical reaction forces when the forward cross beam material history was taken into account.

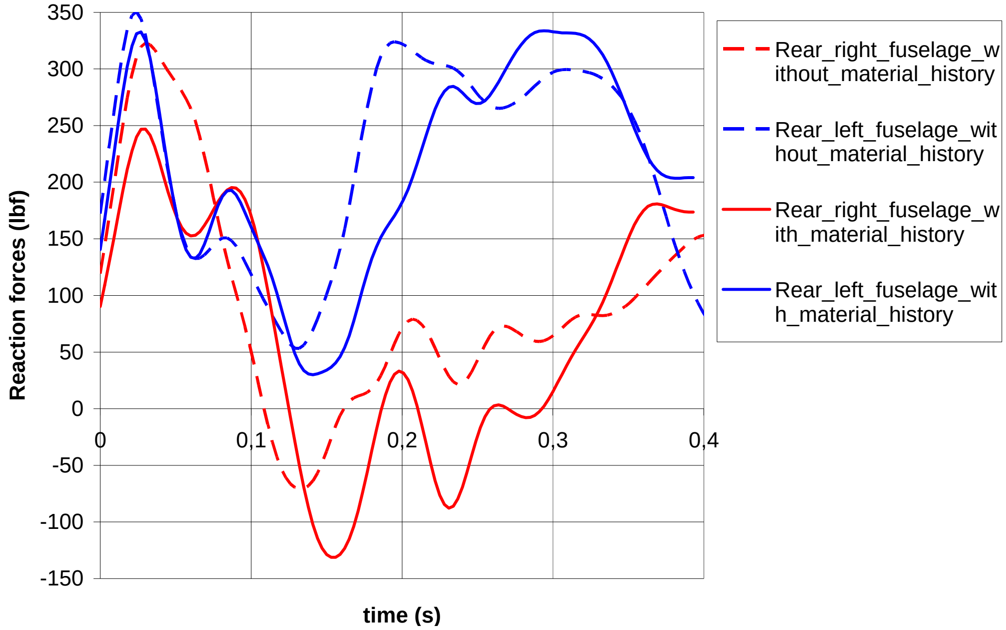

Figure 13. Vertical reactions, rear attachments

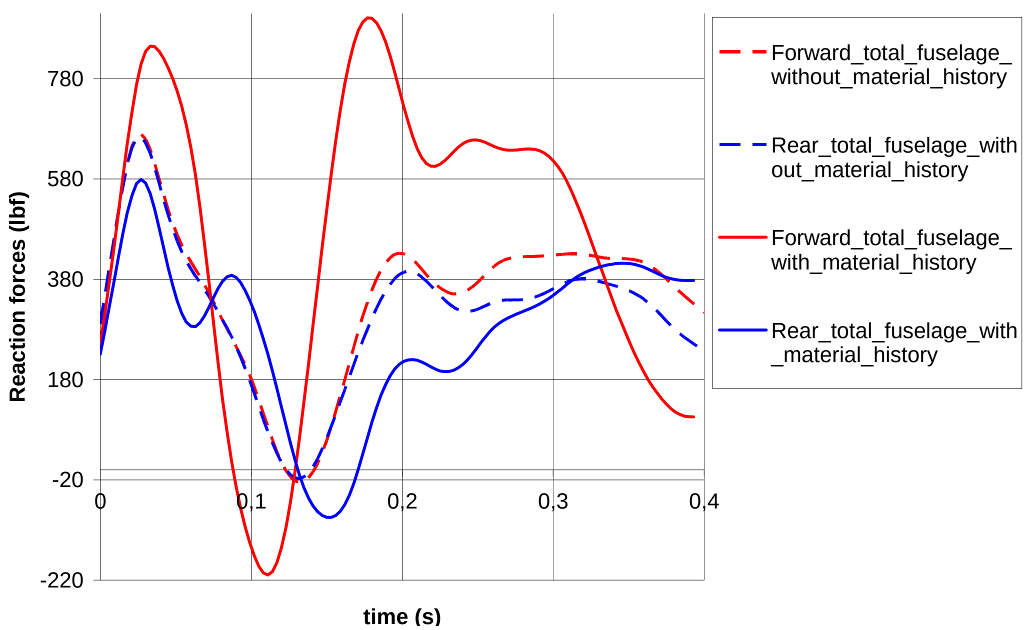

Figure 14. Vertical reactions, forward and rear attachments

The vertical deflections of the two models are presented in figure 15 in three different points: one node on the forward, another on the rear cross beam central part and the last on the center of gravity. When material history is considered, the absolute value of the maximum vertical deflection of the forward cross beam is lower and the forward and rear cross beams rebounded earlier. The skid landing gear of this model is then stiffer than when no initial deformation history is considered.

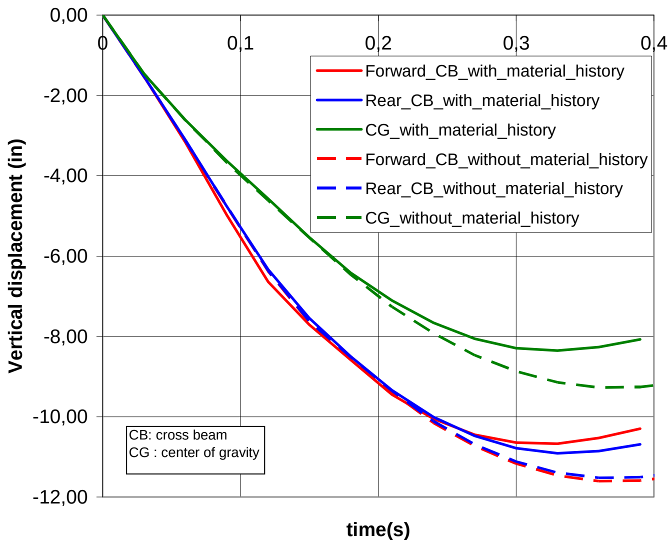

Figure 15. Vertical deflections

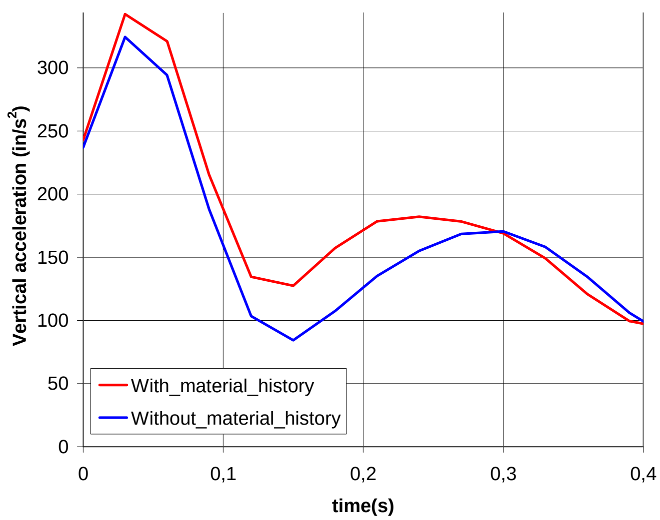

Figure 16. Vertical center of gravity acceleration

Figure 16 presents the vertical acceleration of the center of gravity. A **Butterworth filter** has been used with a **cutoff Frequency** of 5Hz to limit noise in the results. The upward acceleration was higher when material history is considered which means that this model slowed down faster than the other one. This is why the center of gravity rebound seen in figure 15 occurs earlier in this last case.

The kinetic energy presented on figure 17 indicates that they are two rebounds in the models corresponding to the skids rebounds on the ground. Moreover the zero kinetic energy observed corresponds to the instant of maximum deflection. It is interesting to notice that the rebounds at maximum deflection time occurs earlier for the model in which deformation history is accounted for, thus emphasizing that this model is stiffer than the other without material history.

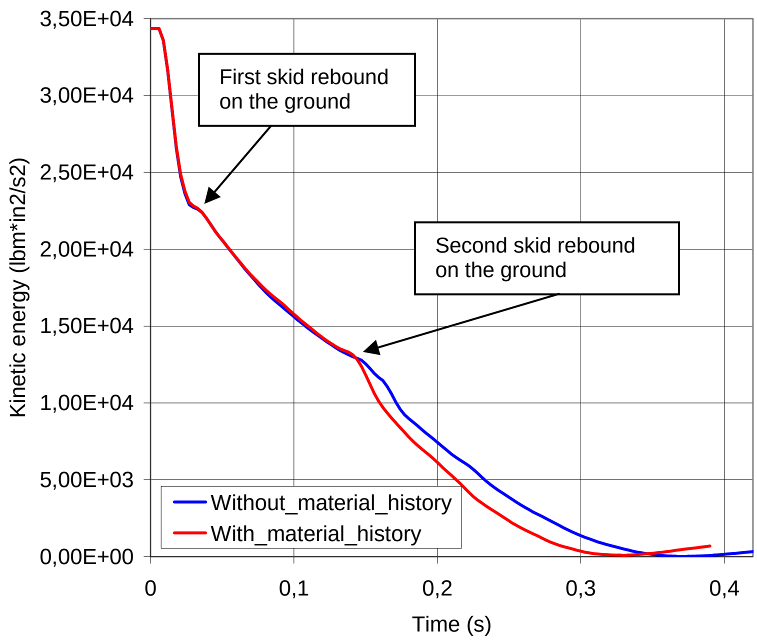

Figure 17. Kinetic energy

The equivalent plastic strain variations following a path along the longitudinal direction of the forward cross beam after bending and after each drop tests are compared in figure 18. This figure shows that the plastic deformation level after drop test without material history was negligible. Moreover the plastified zones spread all over along the tube absorbing at the same time a part of the initial kinetic energy.

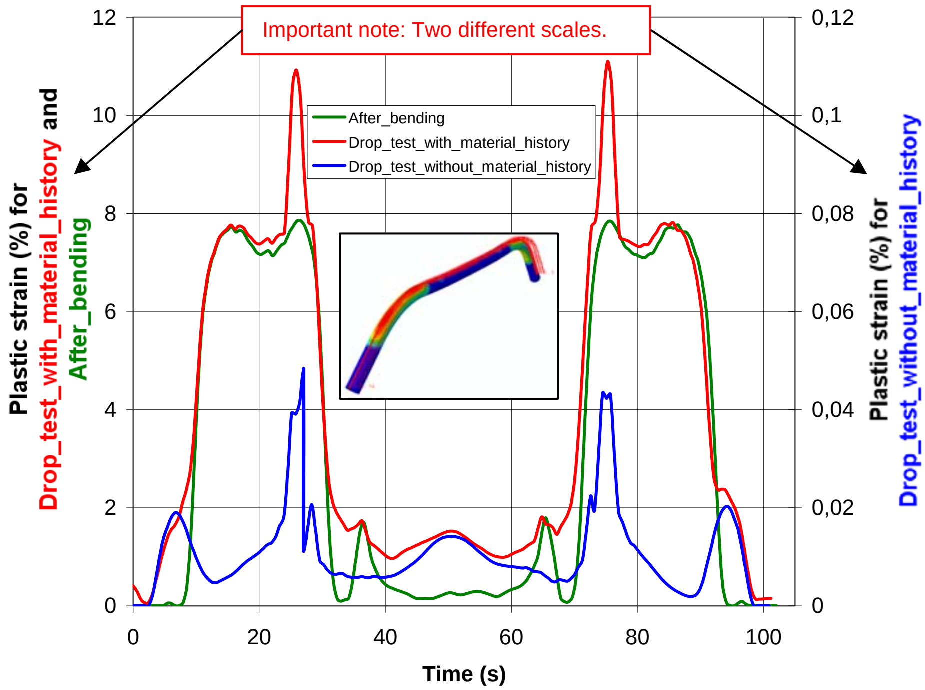

Figure 18. Equivalent plastic strain along the cross beam after drop test

5. Conclusions

Within the Abaqus/CAE environment, a complete methodology has been developed and successfully applied in this paper to simulate a skid landing gear drop test taking into account the manufacturing material history of a cross beam. It was shown that the bending process has a non negligible impact on the cross beam material properties and hence on the whole performance of the complete landing gear.

However, further investigations have to be made since other landing configurations such as landing with drag, rolled attitude landing and level landing with sideward obstruction (Cheng-Ho Tho, 2003) could more emphasize the effect of plastic deformation in the landing gear drop test performance.

Moreover in experimental configuration, the material can be heat treated between the end of bending step and the drop tests. This would provide stronger material stiffness with higher energy absorption capability thus allowing the use of higher drop test velocities in the simulation. Further investigations thus need also to be made on the numerical performance of these effects.

6. References

1. Abaqus user's manual, ver 6.8, Dassault Systèmes, Simulia, 2007.
2. B. Carlo, G. Cozzolino, F. Rossiello, E. Sordelli, M. Usuelli, Fatigue analysis on a helicopter skid landing gear, 16th European MDI user conference, 2001.
3. T. Cheng-Ho, C. E. Sparks, A. K. Sareen, M. R. Smith, Courtney Johnson, Efficient helicopter skid landing gear dynamic drop simulation using Ls-Dyna, 2003.
4. Ls-Dyna Theory Manual, Livermore Software Technology Corporation, 2006.
5. Wang Xi, Jian Cao, 2002, Wrinkling Limit in Tube Bending, Journal of engineering materials and technology.
6. Z. M. Zhan, H. Yang, L. Huang, R. Gu, Springback analysis of numerical control bending of thin-walled tube using numerical-analytic method, Journal of Materials Processing Technology 177 (2006), 197–201.

7. Acknowledgment

The authors are willing to thanks the following institutions for their supports: the department of mechanical engineering of Laval University, the Consortium for Research and Innovation in Aerospace in Quebec (CRIAQ), the Martinique Regional Council associated with the European Social fund and the Aluminum Research Center (REGAL).