Asociación Argentina
de Mecánica Computacional

Mecánica Computacional Vol XXIX, págs. 1913-1924 (artículo completo)
Eduardo Dvorkin, Marcela Goldschmit, Mario Storti (Eds.)
Buenos Aires, Argentina, 15-18 Noviembre 2010

HARDENING AND FINAL THICKNESS DATA USED IN THE QUASI-STATIC NONLINEAR ANALYSIS OF STAMPING PARTS

Alysson L. Vieira[a], Bruno C. Pockszevnicki[b], Evandro Q. N. Vera[c], Danilo R. de Mesquita[a], Márcio E. Silveira[d]

[a]*ABCz Design, Rua das Paineiras, 2320, Contagem, MG, Brasil, alysson.vieira@abczdesign.com.br; danilo.mesquita@abczdesign.com.br*

[b]*VisionGraphic, Av. José Faria da Rocha, 2136, Contagem,MG, Brasil, brupock@yahoo.com.br;*

[c]*Fiat Automóveis, Rodovia Fernão Dias, Km 429, Betim, MG, Brasil, evandro.queiroz@fiat.com.br;*

[d]*Universidade Federal de São João del Rei – UFSJ, Rodovia MG 443, km7, Ouro Branco, MG, Brasil, msilveira@ufsj.edu.br*

Keywords: Forming Effect, Palm Printing, Quasi-Static Nonlinear Analysis.

Abstract. During the development stages of automotive products, the use of numerical simulation tools to evaluate the structural performance of a particular component is essential, since it avoids spending on trial and error, reducing design time while optimizing the shape and function of the component. Most of these numerical simulation tools are based on finite element techniques, with appropriate formulation to solve linear static, dynamic linear, quasi-static non-linear and nonlinear dynamic (impact). Among the tests quasi-static non-linear can cite the simulations of indentation and contact known in the automotive sector such as "palm-printing", "oil-canning" and "elbow-dimpling". These analysis simulate a situation of contact between a given indenter (with different formats) and the external surface of a vehicle component, involving material nonlinearities (plasticity), boundary nonlinearities (contact) and possibly geometric nonlinearities (large displacements and buckling). These external components may be parts of the body as well as fenders, doors and hood. In these simulations, typically are used finite element methods where the sheet metal is modeled with Kirchhoff plate element, with implicit integration for the load and robust formulations enough to consider the presents nonlinearities. Among the main geometric and material input data of the problem, it takes the thickness of the component and the stress-strain curve of material. These data are obtained through tests on specimens of the flat plate (data from the rolling process). External components of an automotive body are manufactured from a stamping of sheet metal plane resulting in a final product with variable thickness due to different levels of stretch and a heterogeneous distribution of residual plastic strain. Generally, these informations are not considered in numerical simulations of the product and may cause considerable errors in the analysis of stamped parts involving nonlinearities. This study aimed to simulate an event called palm-printing in an automobile fender, with and without the consideration of the final data of the numerical simulation of the stamping process (final thickness and residual plastic strain per element) and the results compared with those obtained experimentally. Results showed that the consideration of stretching and hardening from the stamping process can improve the correlation of final results in analysis involving small material and geometric nonlinearities.

1 INTRODUCTION

Increased pressure to reduce manufacturing lead times and program cost in automotive industry has led to rapid developments in analysis software areas. The structural analysis by FEM (Finite Element Method) is commonly used to virtually predict the product performance in terms of crash, NVH and durability in early concept development phase. Experience of full vehicle crash simulations showed that FEM predict higher intrusions than those observed in physical tests, reducing costs and time with trial and error. Analysis of problem showed that reasonable material property input is one of most important factor that controls accurate prediction of analysis performances of structures. However in FEM simulations involving material and geometric nonlinearities, it is very difficult to accurately represent material data for two reasons - first, it varies over a wide range due to limitations in steel manufacturing process and second, it changes during component manufacturing process, ie the thickness changes and work hardening arising during the forming process are generally ignored further analysis.

Material properties of sheet metal components used in vehicle body significantly changes during forming process. Consideration of these material property changes in crash simulation is vital for achieving good simulation accuracy. Steel metal components are manufactured using combination of bending, forming and drawing. Each of these processes cause different levels of permanent (plastic) strains on the component. As a result of the forming strain, the yield stress of the material of component is higher than the original blank, Figure 1, while the thickness tends to decrease as the component is drawing.

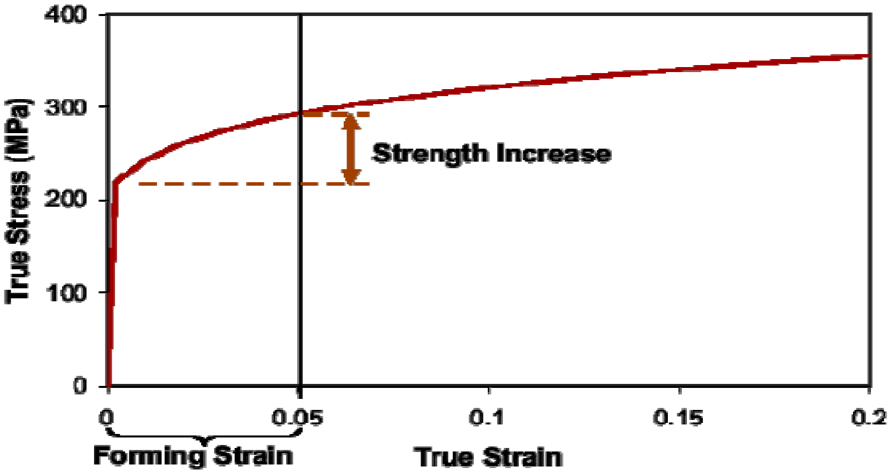

Figure 1: Example of hardening after forming (Gadekar et al, 2006).

Some papers in the area of coupling the crash analysis with the forming analysis has already been reported in the literature. The main two effects which are transferred from the forming simulation to the crash simulation are the thickness distribution and the effective plastic strain (resulting in isotropic hardening). Gadekar et al (2006) presented a study carried out at Tata Motors to illustrate the effect on crash simulation accuracy and the use of one step forming simulation tool to account for forming effects in crash simulation, especially during early concept development stage of vehicle structure. In order to account the forming effects in crash simulations, the end state of forming in terms of plastic strains distribution and thickness variation over component surface (calculated using one step forming methods) is used as an input for crash simulation. It was observed that inclusion of forming effect improves the crash simulation accuracy.

Lanzerath et at. (2001) shows the importance of including work hardening and thickness reduction effects due to the stamping process in crash FE analyses of AHSS (Advanced High Strength Steels) such as, DP (Dual Phase), TRIP (TRansformation Induced Plasticity) and TWIP (TWinning Induced Plasticity).

In simulation of impact of a crash box, Dutton et al. (2001) showed that the residual effect stamping (hardening and thickness) did not exert significant changes in the mode of deformation. However, the collapse load increased by 18% as deceleration decreased approximately 3.7g.

Dagson (2001), Kim et al. (2003) and Hoek (2006) showed that the influence of the residual stresses can be neglected, and the influence of the effective plastic strain is more prominent that decreased of thickness.

Hydroforming leaves significant thickness changes and work hardening and therefore can have a major effect on crash results. Stamped parts are thought to show less sensitivity, because areas that are work hardened would also, in general, be thinned by the stamping process; the two effects might approximately cancel each other (Hoek, 2006).

Most papers reporting on the influence of the stamping process are related to the simulation of high speed impact (crash test), involving large strain and large displacements, beyond to need considering the influence of strain rate on the constitutive model of material, data not always available. The aim of this study was to evaluate the influence of the sheet metal forming (effective plastic strain and thickness) on quasi-static analysis involving small plasticity. For this, it was simulated an analysis of "palm printing" in an automobile fender. This analysis consists to impose a forced displacement (low speed) by an indenter to the surface in order to evaluate possible damage. The numerical results were compared with those obtained experimentally.

2 EXPERIMENTAL TEST

The fender was divided into areas where the palm printing test were performed, as shown in Figure 2.

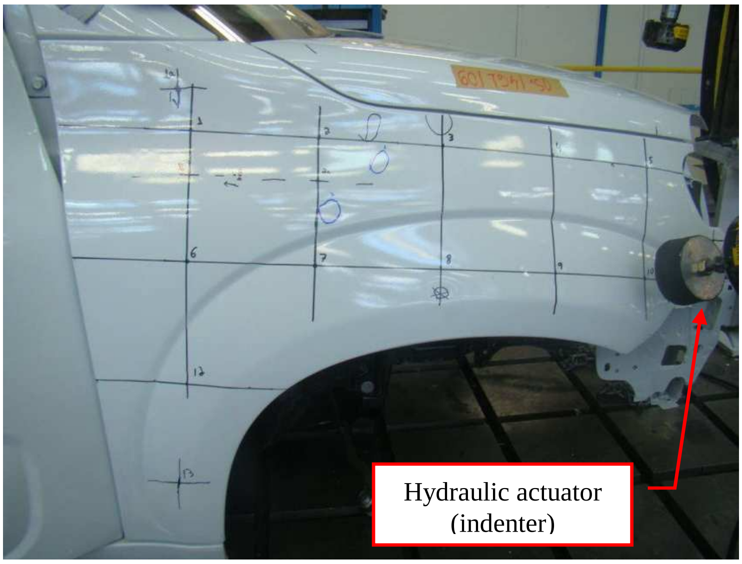

Figure 2: Palm printing test: automobile fender and hydraulic actuator.

This test consists of imposing a displacement on the indenter (Palm) by a hydraulic actuator until the reaction force reaches 400N. These data are monitored and evaluated whether there was excessive plastic deformation (visible) or buckling. On this work, the point-7 was chosen to be correlated with the numerical simulation. The experimental test results of point-7 can be seen in the graph of Figure 3.

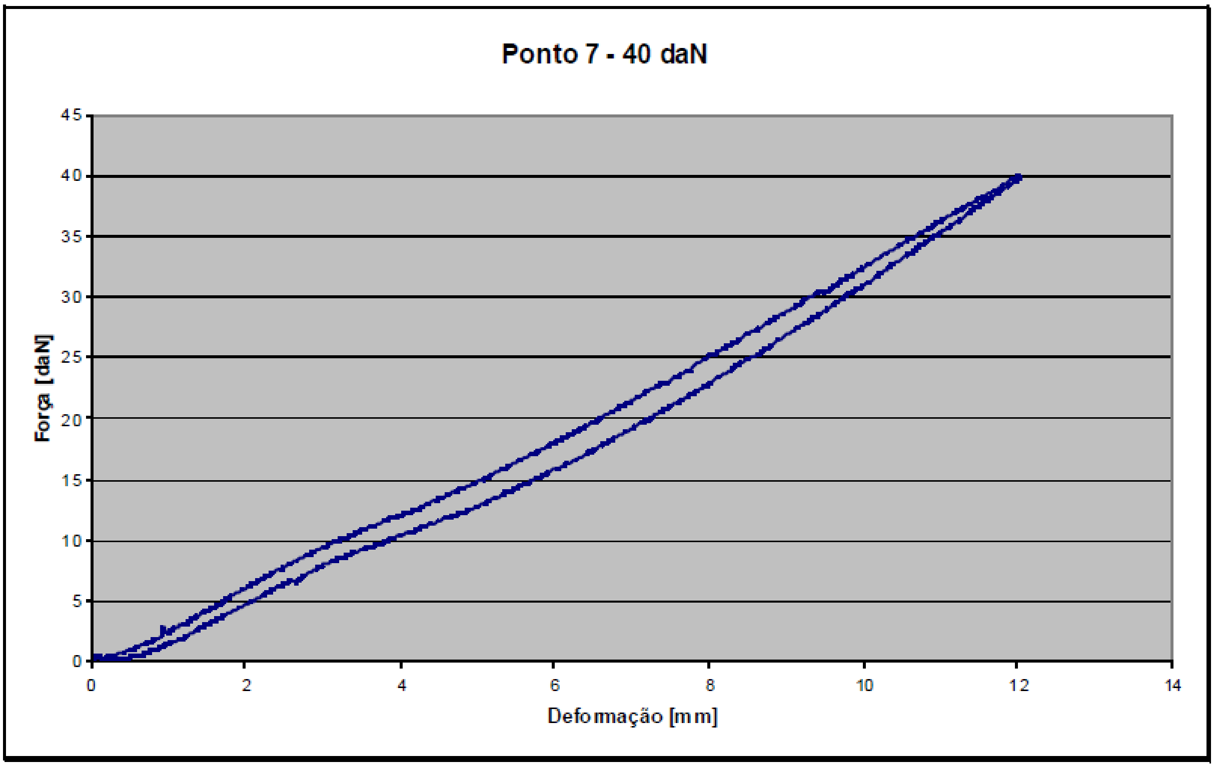

Figure 3- Force x Displacement graph of point-7.

3 NUMERICAL SIMULATION

Finite element method began to be introduced in the fifties as an analytical tool to aid engineering projects. In the sixties, with the advancement of the aerospace sector, the demand for more efficient and reliable methods increased dramatically at universities and industries. Nonlinear analysis using finite element method also started in this period. Among the first articles about nonlinear analysis, one can cite: Argyries (1965) and Marcal et al. (1967), cited in Belytschko et al. (2000). Pedro Marcal, who was a professor at Brown University, created in 1969 a company that released the MARC, first software on the market with nonlinear finite element analysis, which its implicit formulation is still used nowadays. A milestone of the progress in the explicit finite element formulation was the work of John Hallquist at Lawrence Livermore Laboratory. John began his work in 1975 and the first version of Dyna explicit code was released in 1976. Dyna code evolved over time and formed the basis for several commercial programs such as Radioss, LS-Dyna, PamCrash and Dytran. The stamping simulation was performed by software with explicit formulation (LS-Dyna) and quasi-static analysis of palm-printing done in an implicit software (Abaqus Standard).

3.1 Sheet Metal Forming

The stamping simulation was performed in the LS-Dyna, where the equilibrium equation of a dynamic analysis discretized in finite elements can be given as follows:

$$[M]\{\ddot{U}\}+[C]\{\dot{U}\}+[K]\{U\}=\{F(t)\} \tag{1}$$

where [*M*] is the mass matrix, [*C*] is the damping matrix, [*K*] is the stiffness matrix and {*F*(*t*)} is the vector of external forces. The vectors of acceleration, velocity and displacement are given by $\{\ddot{U}\}$, $\{\dot{U}\}$and $\{U\}$ for $t = t_0$. LS-Dyna uses in its explicit formulation the finite difference method which uses the equation (1) evaluated at a time *t*, where the vectors of acceleration and velocity are given as:

$$\begin{aligned} {}^{t}\{\ddot{U}\} &= \frac{1}{\Delta t^2}\left({}^{t-\Delta t}\{U\}-2\,{}^{t}\{U\}+{}^{t+\Delta t}\{U\}\right) \\ {}^{t}\{\dot{U}\} &= \frac{1}{2\Delta t}\left(-{}^{t-\Delta t}\{U\}+{}^{t+\Delta t}\{U\}\right) \end{aligned} \tag{2}$$

The solution for ${}^{t+\Delta t}\{U\}$ can be written as:

$$\begin{aligned} \left(\frac{1}{\Delta t^2}[M]+\frac{1}{2\Delta t}[C]\right){}^{t+\Delta t}\{U\} = {}^{t}\{F\}-\left([K]-\frac{2}{\Delta t^2}[M]\right){}^{t}\{U\}- \\ \left(\frac{1}{\Delta t^2}[M]-\frac{1}{2\Delta t}[C]\right){}^{t-\Delta t}\{U\} \end{aligned} \tag{3}$$

It can be observed that is not necessary the factorization of matrices on each time-step, since [*C*] and [*M*] are approximate to diagonal matrices (lumped mass). This is the main factor of the robustness and low computational cost of explicit methods. The stability of the problem depends on the size of the time-step, which is determined by Courant criterion that is directly related to the element size and the speed of sound in the environment.

In the blank, it was used plate elements with bi-linear interpolation and 6 degrees of freedom per node. In this case, Mindlin model was adopted, where the cross section remains plane after deformation but not necessarily orthogonal. This element has only one point of integration in the plane, which is providential to avoid locking problems, but can generate questionable results due to hourglass (spurious zero energy modes). To circumvent this problem, it was necessary to add “anti-hourglass” forces and moments, available in the LS-Dyna. Along the thickness 5 points of integration were used. The material was modeled as elastic-plastic with isotropic hardening and the yield criterion of Hill48 (Owen and Hilton, 1986). The LS-Dyna has appropriate formulation to deal with problems involving geometric, material and boundary nonlinearity, which is the case of the sheet metal forming simulation.

The Figure 4 show the model of fender forming, performed in the LS-Dyna software. The material of the fender is the steel BH220 with a nominal thickness of 0.65 mm. The Figure 5 show de graph of true stress x true plastic stress of BH220, and other data are given in Table 1.

	E (GPa)	ν	ρ (Ton/mm^3)	$\bar{R}$
BH220	207	0.28	7.85E-09	1.338

Table 1: Steel BH220 material data.

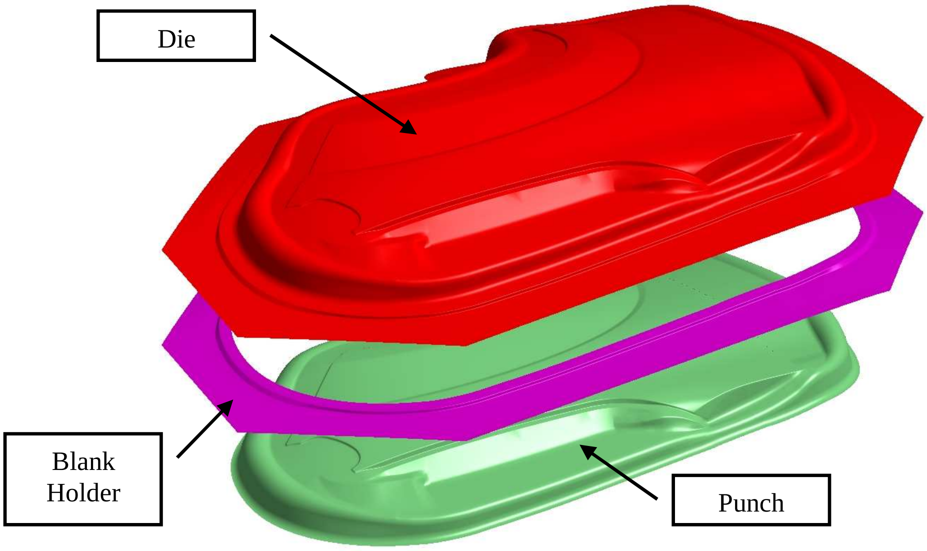

Figure 4 - Model of fender simulation forming.

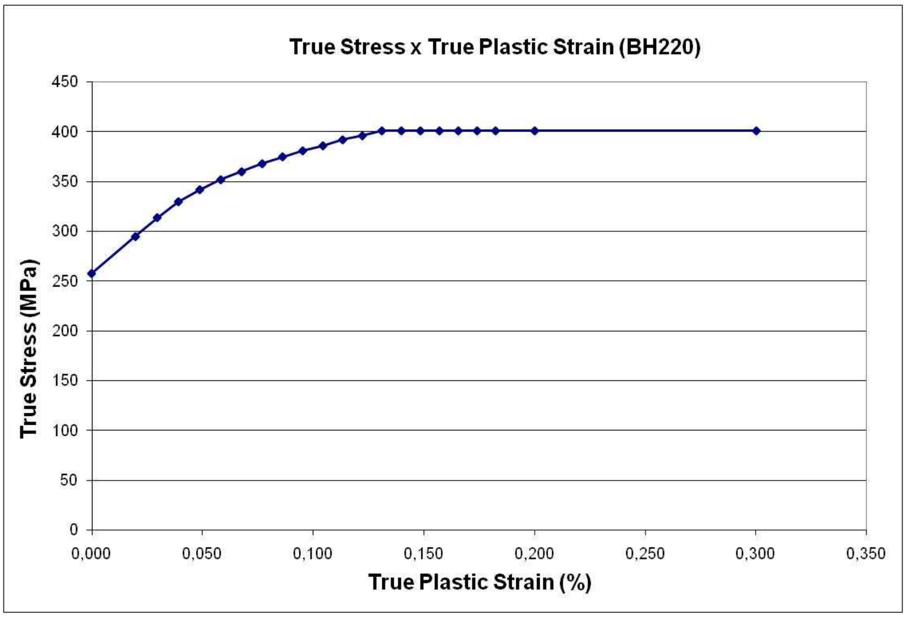

Figure 5 - BH220 - True stress x true plastic strain.

3.2 Results Mapping

The results of the forming simulation must be mapped to the model of quasi-static analysis, because they use different meshes (and different software). Thus, it was used a feature of LS-Dyna that allowed mapping the effective plastic strains and the thickness from the forming process to the model of quasi-static analysis of palm-printing. In the Figure 6a it can be view the results of effective plastic strain in the original software (LS-Dyna) and the result of mapping on the mesh of quasi-static analysis (Abaqus).

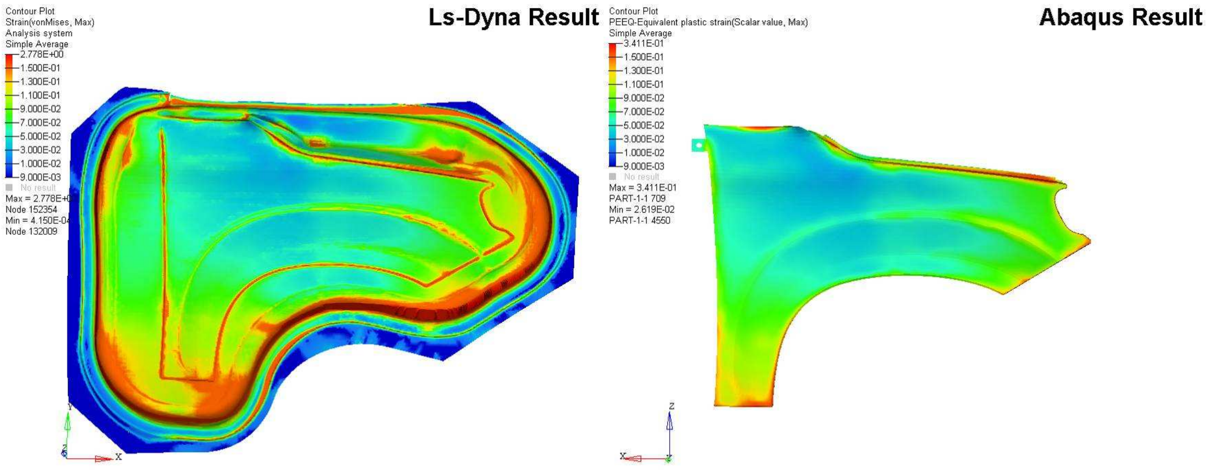

Figure 6a - Distribution of effective plastic strain in the original mesh (LS-Dyna) and the result of mapping on the new mesh.

In the Figure 6b it can be view the results of final thickness in the original software (LS-Dyna) and the result of mapping on the mesh of quasi-static analysis (Abaqus).

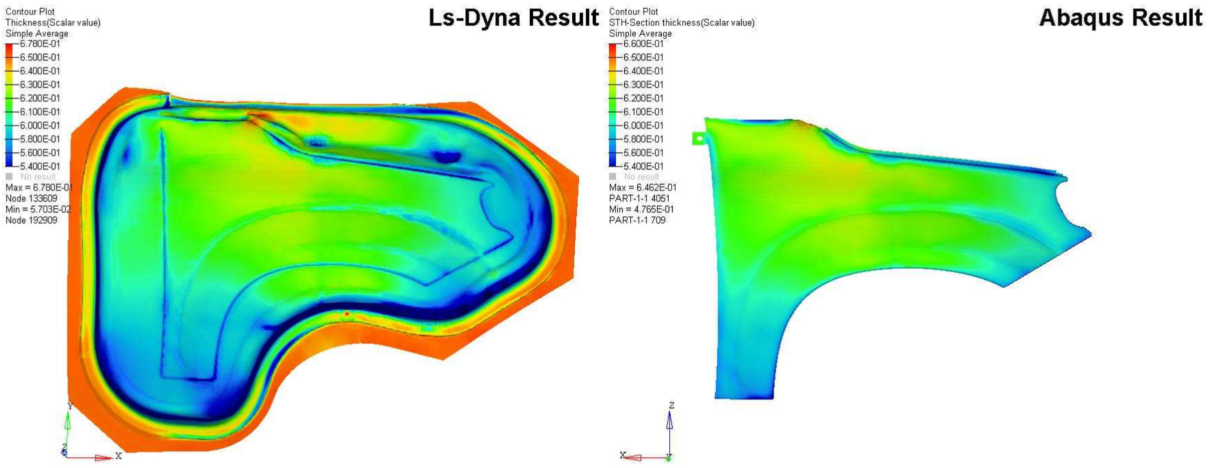

Figure 6b - Distribution of final thickness in the original mesh (LS-Dyna) and the result of mapping on the new mesh.

3.3 Palm Printing Simulation

The palm printing test was simulated as being a quasi-estatic analysis. It was used the ABAQUS/Standard that generally uses Newton's method as a numerical technique for solving the nonlinear equilibrium equations. In nonlinear problems the objective is to obtain a convergent solution at a minimum cost. The nonlinear procedures in ABAQUS/Standard offer two approaches to this. Direct user control of increment size is one choice, whereby the user specifies the incrementation scheme. Automatic control is the alternate approach: the user defines the step and specifies certain tolerances or error measures. Abaqus used the Updated Lagrangian with the classical "additive rate of deformation decomposition" of plasticity theory (Belytschko, 2000) for the most of analysis nonlinear. The material was modeled as elastic-plastic with isotropic hardening and the yield criterion of Von Mises. The sheel element used (S4R with reduced integration) provide robust and accurate solutions in all loading conditions for thin and thick shell problems. Thickness change as a function of in-plane deformation is allowed in their formulation. They do not suffer from transverse shear locking, nor do they have any unconstrained hourglass modes. The effective plastic strain and the thickness from the output forming simulation were added as input data for this analysis.

The model for fender quasi-static analysis of palm printing, discretized into finite elements can be seen in Figure 7. It can be observed that the fender was modeled already mounted on the vehicle, in order to include all the boundary conditions necessary to bring the virtual simulation to the experimental test.

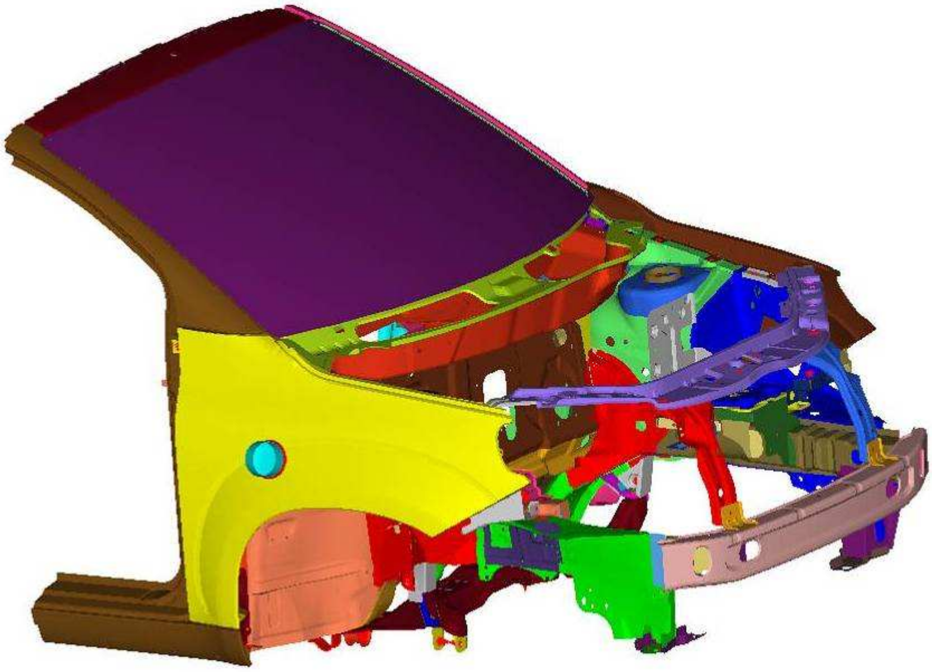

Figure 7 - FEM palm-printing analysis model.

O results of Von Mises Stress for the fender with the nominal data (thickness from blank and no plastic strain) can be seen in the Figure 8a. In the Figures 8b, 8c and 8d are showed the results of Von Mises Stress for the fender, considering the only effective plastic strain, only thickness and both respectively.

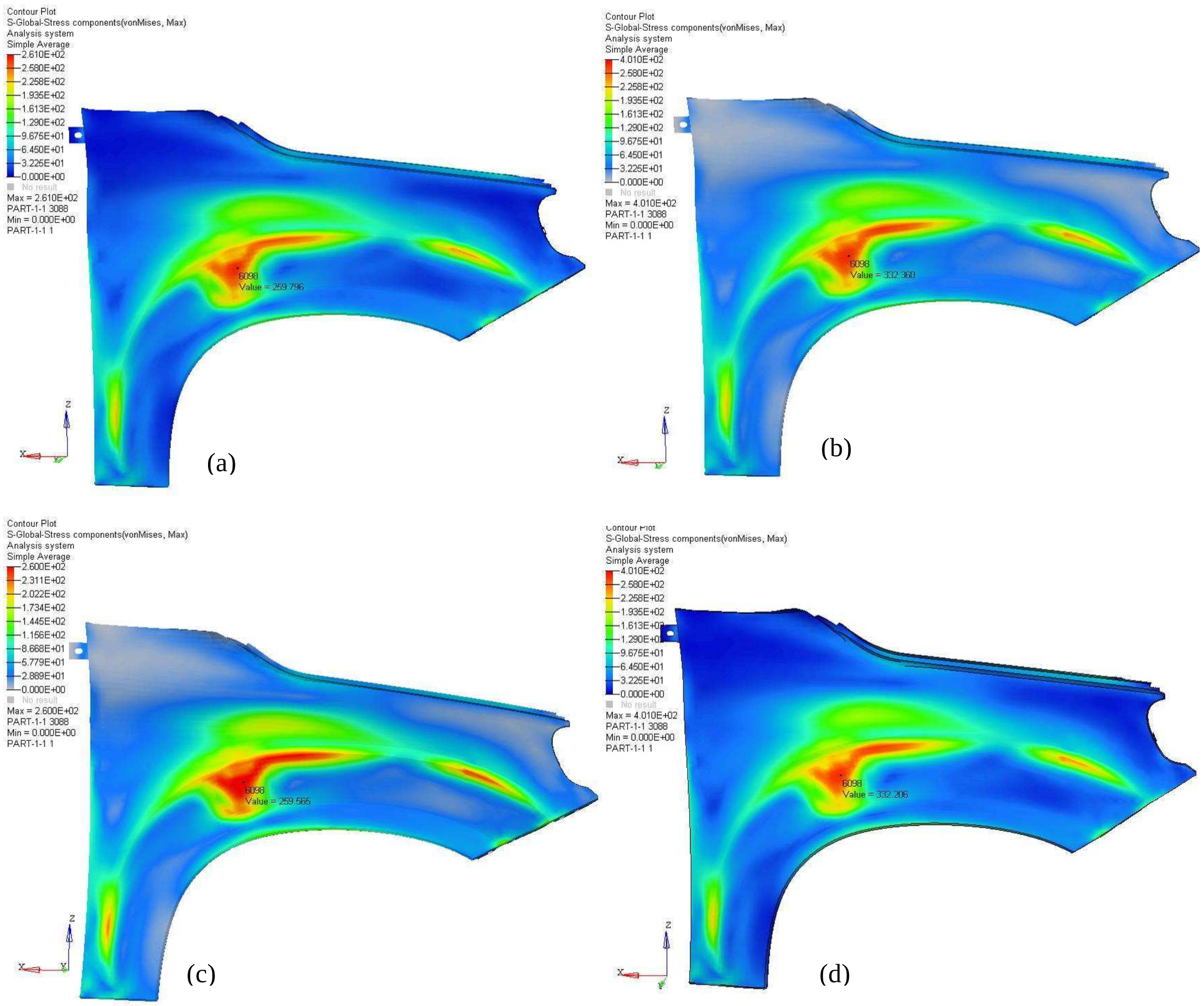

Figure 8 - Von mises stress in contact local of indenter for the fender wih: a) nominal data; b) only effective plastic strain from stamping simulation; c) only thickness from stamping simulation; d) both effective plastic strain and thickness.

Figure 9 show the evolution of the stress over the displacement of the indenter for these four cases. It can be observed a change in stress results considering only the effective plastic strain and the results the original blank. This is due to the hardening in this region of analysis, about 2 %. This small hardening change significantly the "stress x strain" curve for the results considering only the plastic deformation, while in the stress due to reduction in thickness were observed few changes (remembering it is imposed displacement simulation). The graph in Figure 10 show how the new curve for steel BH220 would be with hardening effect (about 2%).

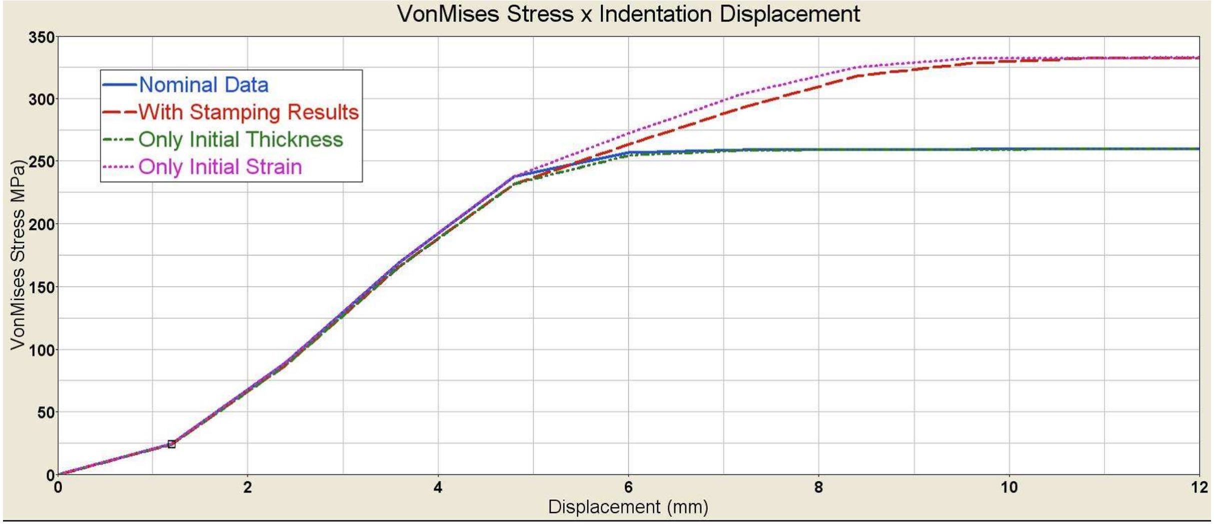

Figure 9 - Evaluation of the von Mises stress along the displacement of the indenter.

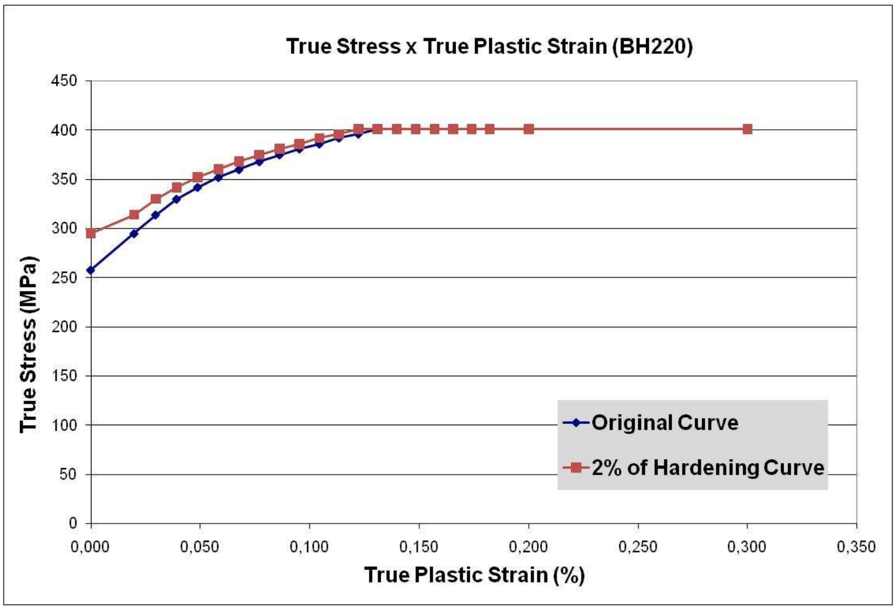

Figure 10 - Original and hardening curve of steel BH220.

In the Table 2, it can observe that the force behavior along the indenter displacement is more prominent in the simulation considering only thickness that in the simulation considering only hardening, for the same displacement (12mm). This fact goes against the results reported in the literature for cases of crash test simulation, which has large strain, resulting in the common occurrence of "plastic hinges". For cases of small plastic strain and load increment quasi-static, such as the analysis of palm-printing, the inclusion of the thickness resulting of process forming provides an important role in the correlation of results.

The Figure 11 show punch force along displacement, comparing the experimental test results with those obtained by numerical simulation (with and without considering the forming data). It can be seen that the inclusion of forming data improves the numerical results, bringing them closer to the experimental results.

Analysis	**Punch Force (N)**
Nominal	443,4
Only thickness	390,8
Only hardening	445,0
Both	391,9
Experimental	400,0

Table 2: Force for a displacement of indenter of 12mm.

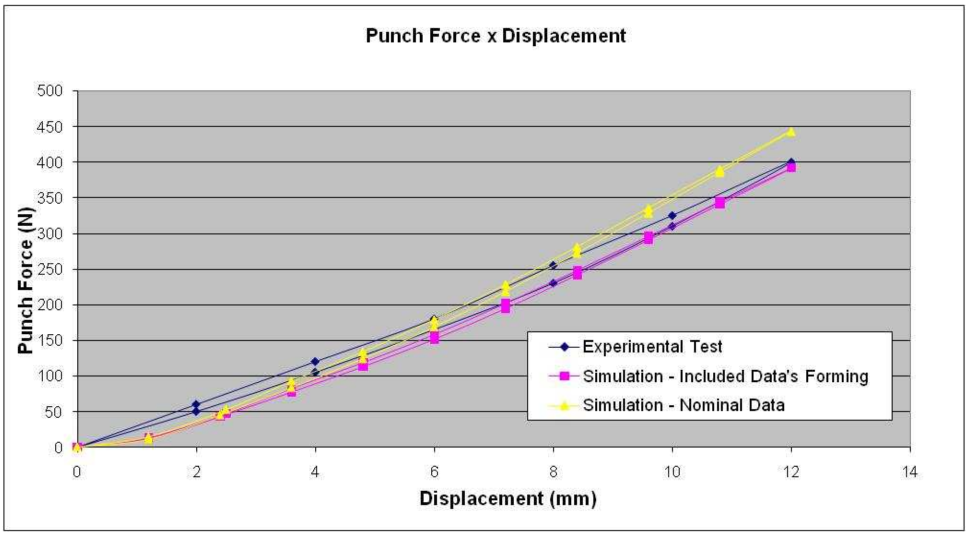

Figure 11 - Graph of punch force along displacement of indenter.

4 CONCLUSION

Results showed that the consideration of thickness and hardening from the stamping process can improve the correlation of final results in quasi-static nonlinear analysis. However, the results showed a more prominent influence of the thickness than the hardening, especially in the correlation of the reaction force of the indenter, contrary to reports in the literature in cases of crash test simulation. A likely cause is that at large deformations, such as crash test, the hardening became a dominant factor. Other points on the fender were being analyzed and results similar to this were found. In another factor that may have influenced, is the fact of the steel BH220 has a hardening rate small if compared to other structural steels (HSS and AHSS), very used in crash tests. This work showed the need for research on this subject, including other important parameters in the analysis, such as the influence of the coefficient of anisotropy and the effect of BH (*Bake Hardening*) of these steel, which can increase the yield stress up to 30MPa (after painting), increasing further the influence of strain hardening in the analysis.

REFERENCES

Belytschko, T., Liu, W. K., Moran, B., *Nonlinear Finite Elements for Continua and Structures*. John Wiley & Sons, 2000.

Dagson, N., *Influence of the Forming Process on the Crash Response of a Roof Rail Component*. M.Sc. Thesis, Department of Solid Mechanics, Linkping University, 2001.

Dutton, T., Richardson, P., Knight, A., Sturt, R., The influence of residual effects of stamping on crash results. *3rd European LS-DYNA Conference.* Paris, 2001.

Gadekar, G.B., Raj, P., Kshirsagar, S., Anilkumar, C., Inclusion of forming effect in crash simulations using one step forming simulations. *Altair CAE User Conference - Inovation Through Simulation*. August 3-5, Bangalore, 2006.

Hoek, T. V., *The History Influence of Forming on the Predicted Crash Performance of a Truck Bumper*. M.Sc. Thesis. Eindhoven, 2006.

Kim, H., Hong, S., Hong S., Huh, H., The Evaluation of crashworthiness of vehicles with forming effect. *4th European LS-Dyna Users Conference*. 2003

Lanzerath, H., Ghouati, O., Wesemann, J., Schilling, R., Influence of manufacturing processes on the performance of vehicles in frontal crash. *3rd European LS-DYNA Conference.* Paris, 2001.

Owen, D. R. J., Hinton, E., *Finite Elements in Plasticity: Theory and Practice*. Pine-ridge Press Limited, Swansea, 1986.