

INFLUENCE OF FORMING PROCESSES ON THE QUASI-STATIC NONLINEAR ANALYSIS OF AUTOMOTIVE PARTS

Alysson Lucas Vieira, alysson.vieira@abczdesign.com.br

ABCz Design - Rua das Paineiras, 2320, Contagem, Minas Gerais, Brazil - 32310.400.

Márcio Eduardo Silveira, msilveira@ufsj.edu.br

Federal University of São João del Rei – UFSJ - Rodovia MG 443, km7, Ouro Branco, Minas Gerais, Brazil - 36420.000.

Bruno Cesar Pockszevnicki, bruno.cesar@tecpronbrazil.com

TECPRON - Alameda Carlos de Carvalho, 417, s2101, Curitiba, Paraná, Brazil - 80410.201.

Evandro Queiroz Nunes Vera, evandro.queiroz@fiat.com.br

Fiat Automóveis, Rodovia Fernão Dias, Km 429, Betim, Minas Gerais, Brazil - 32530.000.

Abstract. *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. 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 quasi-static non-linear tests, the simulations of indentation and contact known in the automotive sector as “palm-printing”, “oil-canning”, and “elbow-dimpling” can be cited. 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. 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 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.*

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

1. INTRODUCTION

The need to reduce manufacturing lead times and program cost in automotive industry has led to the use of numerical simulation tools at various stages of development and product validation. The structural analysis by FEM (Finite Element Method) is commonly used to virtually predict the product performance in terms of crash, NVH (Noise, Vibration, and Hoarshness) 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 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 include material data from manufacturing 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, i.e., the thickness changes and work hardening arising during the forming process are generally ignored in 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, Fig. 1, while the thickness tends to decrease as the component is drawing.

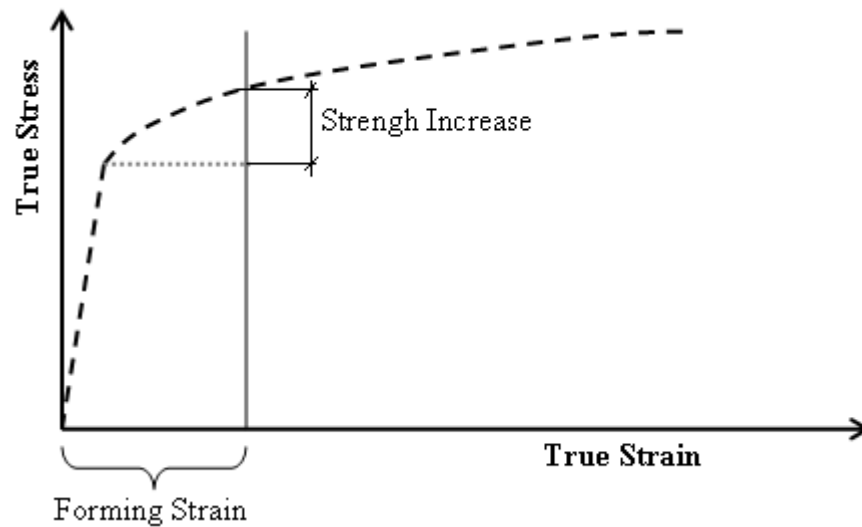


Figure 1: Example of hardening after forming.

Some papers in the area of coupling the crash analysis with the forming analysis has 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) were used as an input for crash simulation. It was observed that inclusion of forming effect improves the crash simulation accuracy.

Lanzerath *et al.* (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 on the simulation of crash of stamping parts.

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 is 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 Fig. 2.

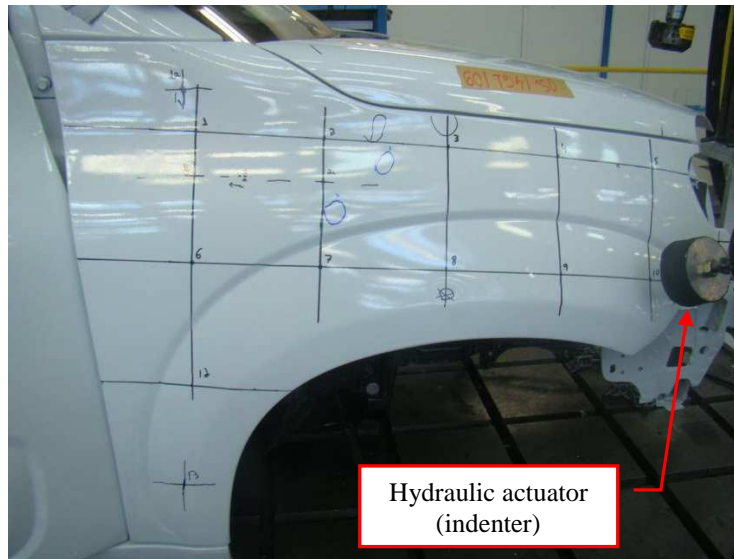


Figure 2: Palm printing test: automobile fender and hydraulic actuator.

This test consists of imposing a displacement on the indenter 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- 11 (located on the right side of the fender) was chosen to be correlated with the numerical simulation. The experimental test results of point-11 can be seen in the graph of Fig. 3. For a force of 400 N, the indenter has moved 13.4 mm. One can see from the graph that, during unloading, the component has undergone permanent plastic strain.

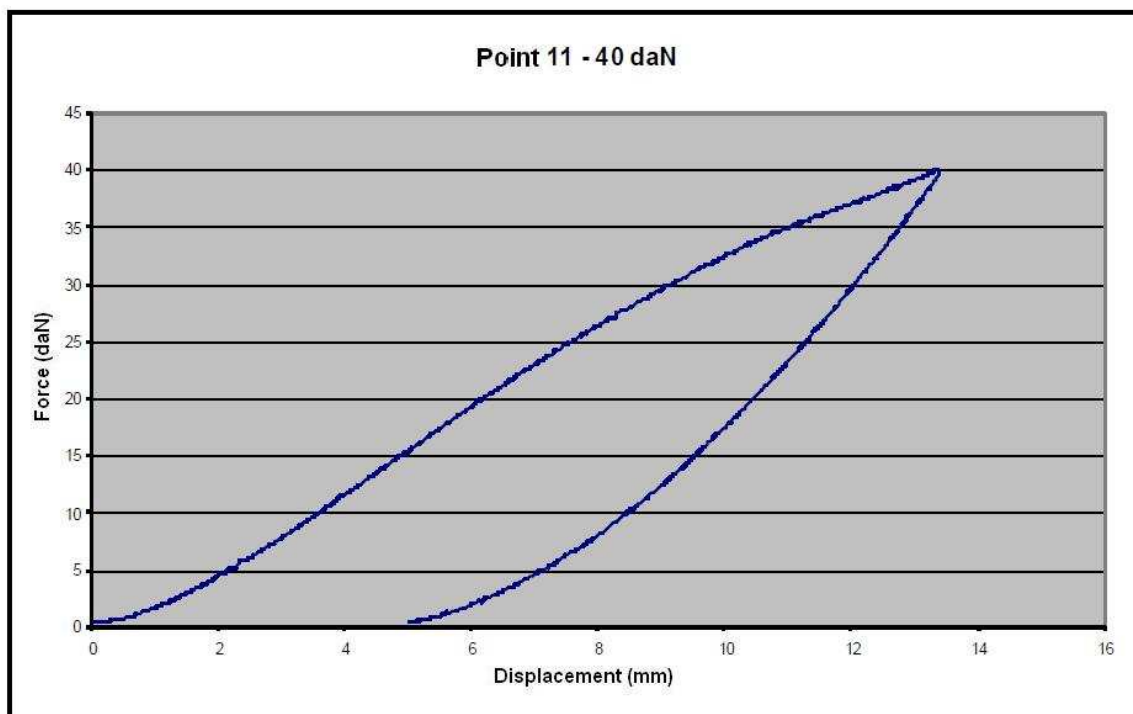


Figure 3- Force x Displacement graph of point-11.

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:

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

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:

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

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

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

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 critical element size on the mesh and the wave propagation speed.

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.

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. Figure 5 show de graph of true stress x true plastic stress of BH220, and other data are given in Table 1. As the velocity of stamping is low, the data of strain rate and temperature variation were ignored.

Table 1: Steel BH220 material data.

	Young's Modulus E (GPa)	Poisson Coefficient ν	Density ρ (Ton/mm ³)	Lankford Coefficient \bar{R}
BH220	207	0.28	7.85E-09	1.338

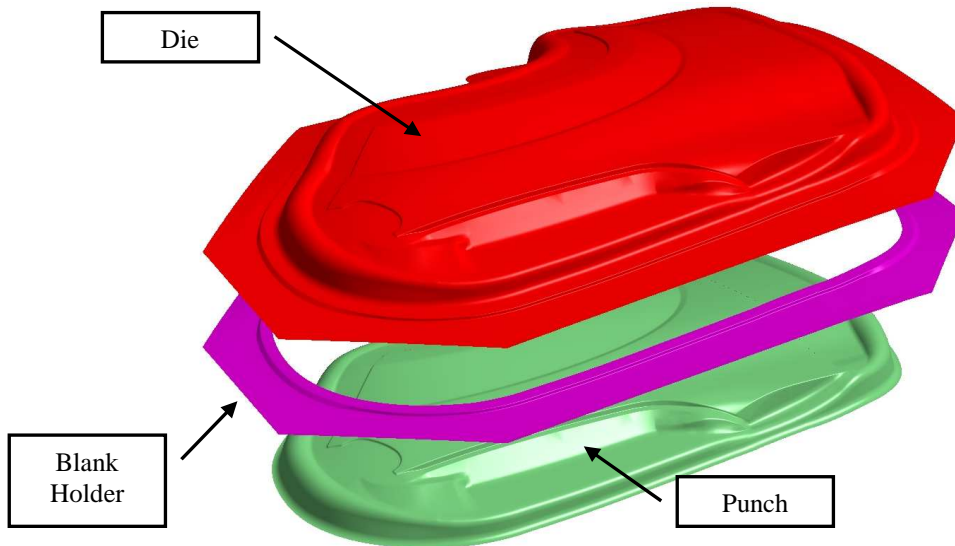


Figure 4 - Model of fender simulation forming.

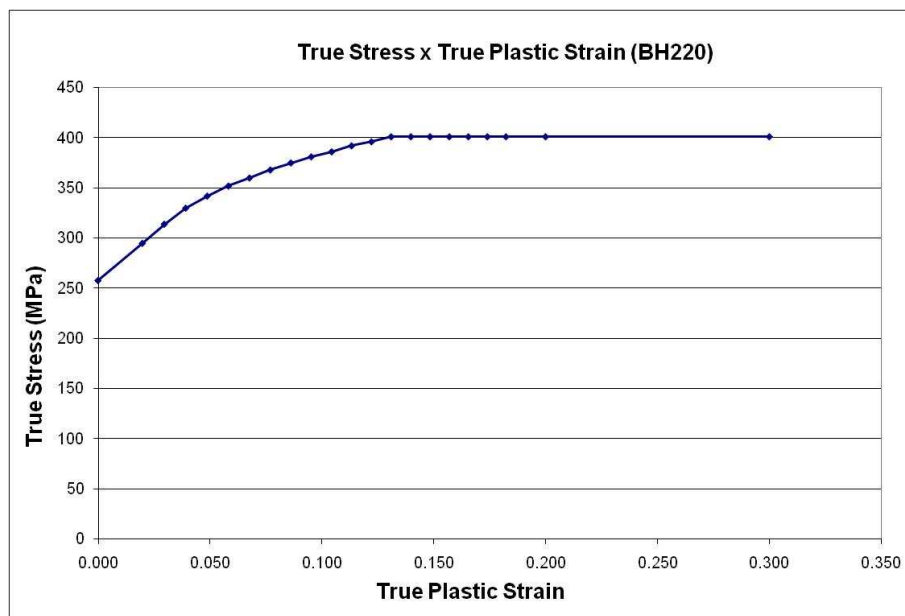


Figure 5 - BH220 - True stress x true plastic strain.

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 Fig. 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).

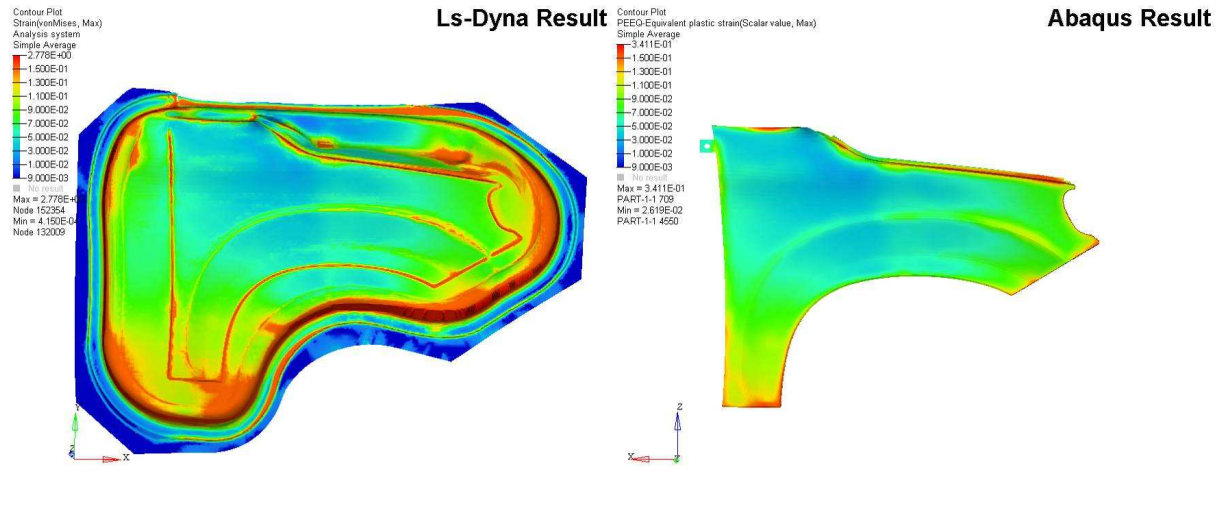


Figure 6a - Distribution of effective plastic strain in the original mesh (LS-Dyna) and the result of mapping on the new mesh.

In the Fig. 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).

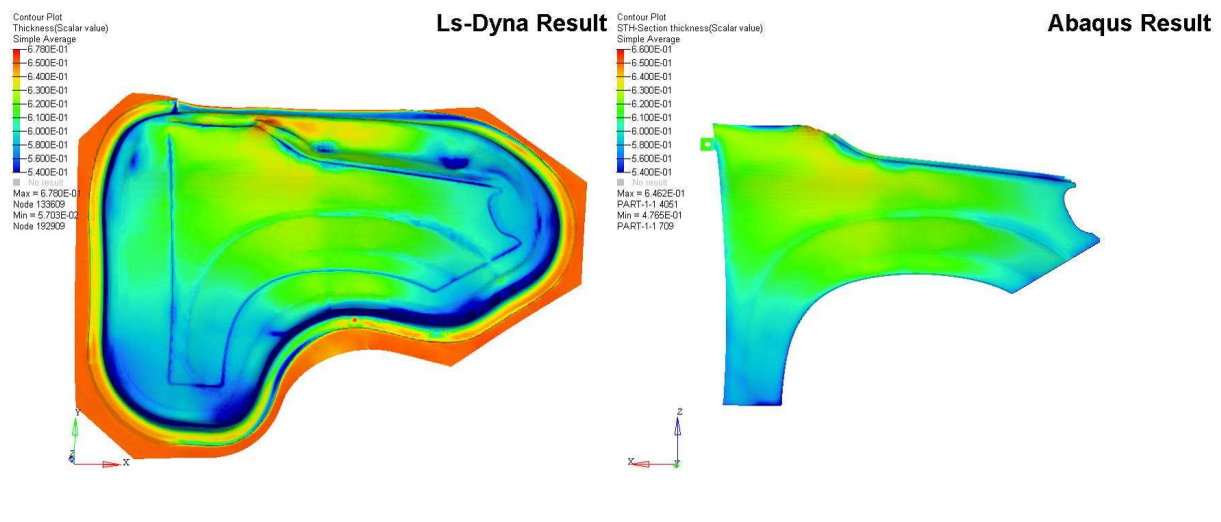


Figure 6b - Distribution of final thickness in the original mesh (LS-Dyna) and the result of mapping on the new mesh.

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 shell element used (S4R with reduced integration) provides robust and accurate solutions in all loading conditions for thin and thick shell problems. Their formulation considers the thickness change as a function of in-plane deformation. 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 Fig. 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.

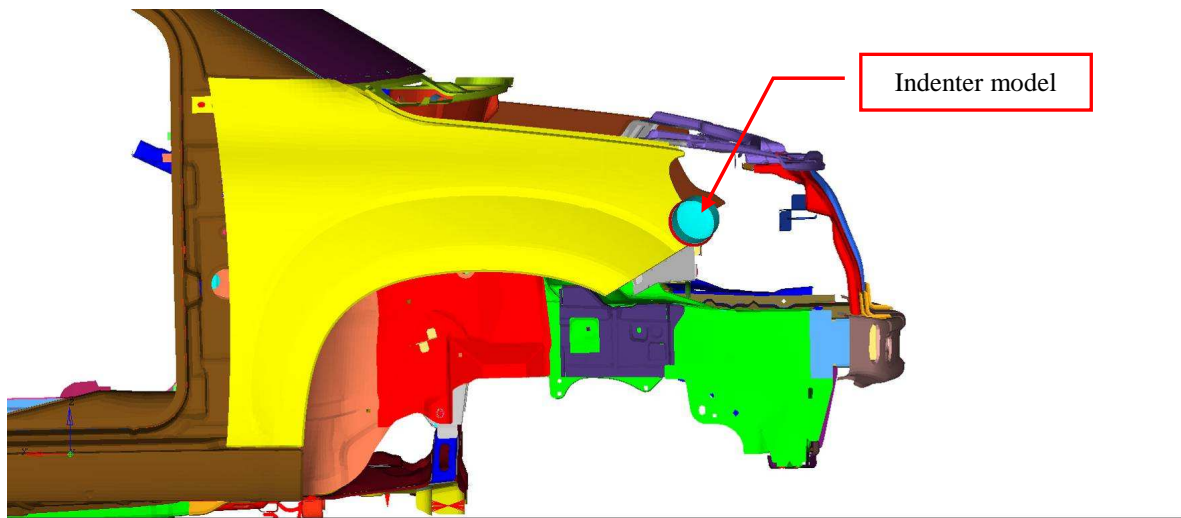


Figure 7 - FEM palm-printing analysis model.

The results of Von Mises Stress for the fender with the nominal data (thickness from blank and no plastic strain) can be seen in Fig. 8a. Figure 8b is showed the result of Von Mises Stress for the fender, considering the effective plastic strain and thickness from the stamping process simulation.

It can be observed a change in stress results considering the forming effects regarding to the results with original blank data. This is due to the hardening in this region of analysis, about 1 %. This small hardening change significantly the "stress x strain" curve for the results considering the forming effects. The graph in Fig. 9 show how the new curve for steel BH220 would be with hardening effect (about 1%).

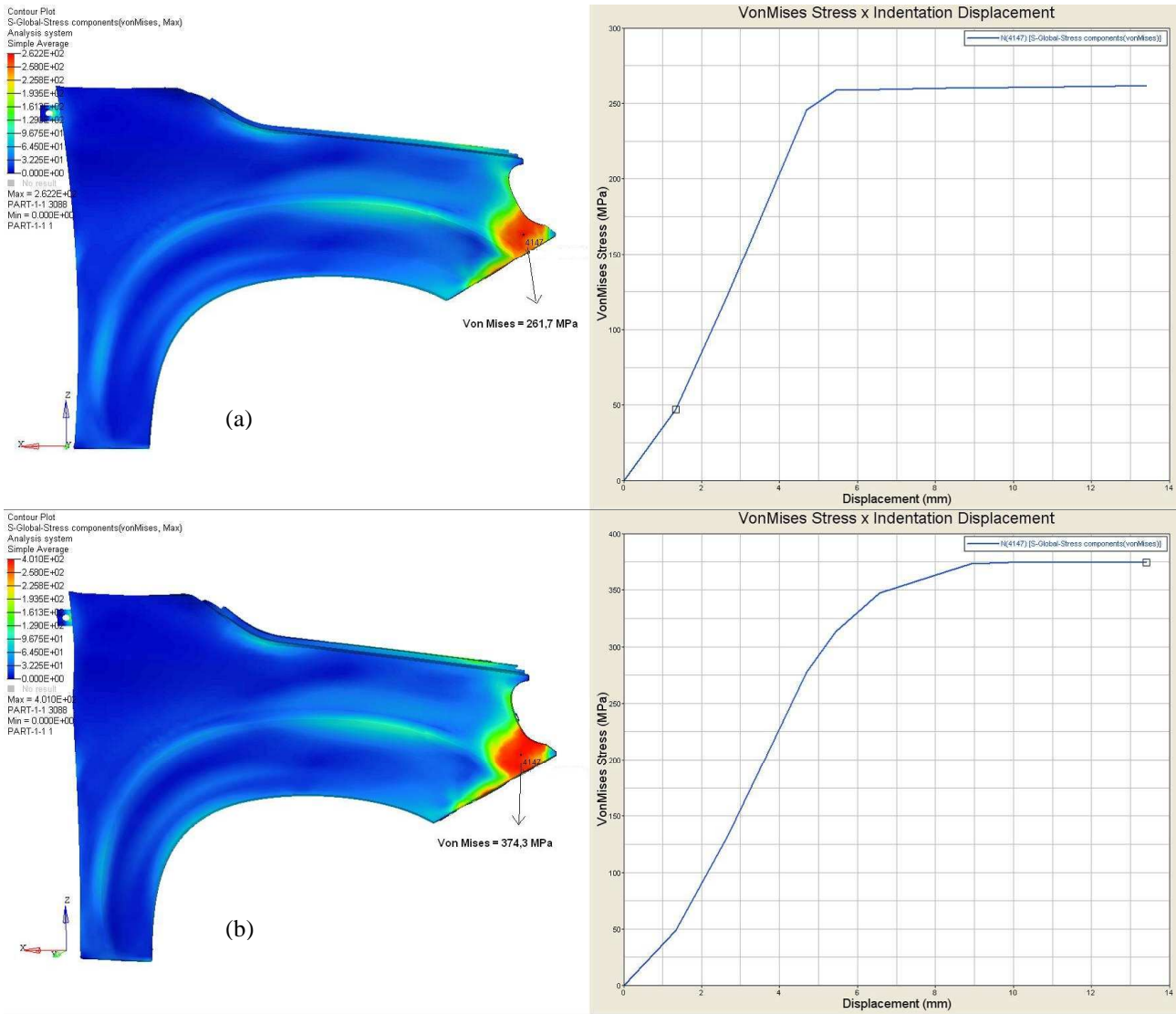


Figure 8 - Von Mises stress in contact local of indenter with the fender: a) nominal data; b) both effective plastic strain and thickness from forming process.

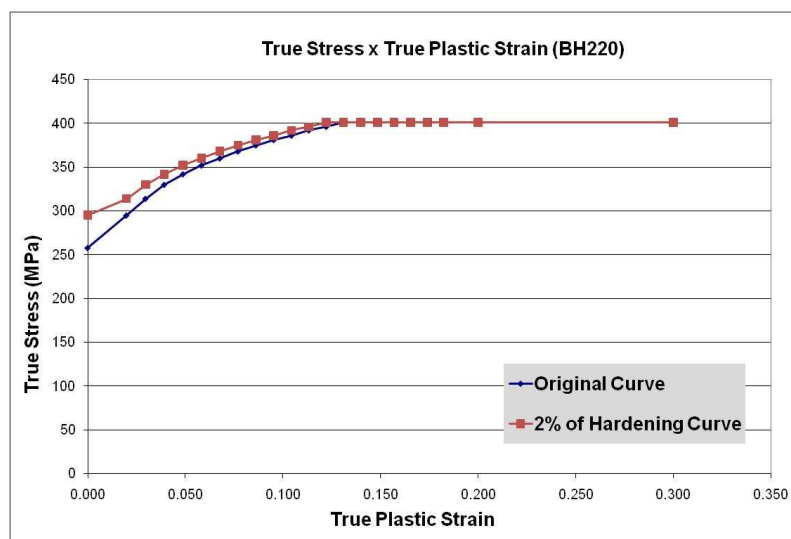


Figure 9- Original and hardening curve of steel BH220.

In Table 2, it can observe that the force behavior along the indenter displacement is more prominent in the simulation considering only thickness than in the simulation considering only hardening, for the same displacement (13.4mm). 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". The reduction in thickness affects directly the stiffness of the material, reducing the force required to perform the test during the elastic phase. In this type of analysis, the fender undergoes a very small local plastic strain (see Fig. 8). As the steel used (BH220) has the hardening rate fairly low, the influence of the residual plastic strain is not prominent.

Therefore, 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.

Figure 10 shows 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.

Table 2: Force for a displacement of indenter of 13.4mm.

Analysis	Punch Force (N)	Error
Nominal	436.9	9.2%
Only thickness	414.1	3.5%
Only hardening	440.7	10.2%
Both	417.5	4.4%
Experimental	400.0	

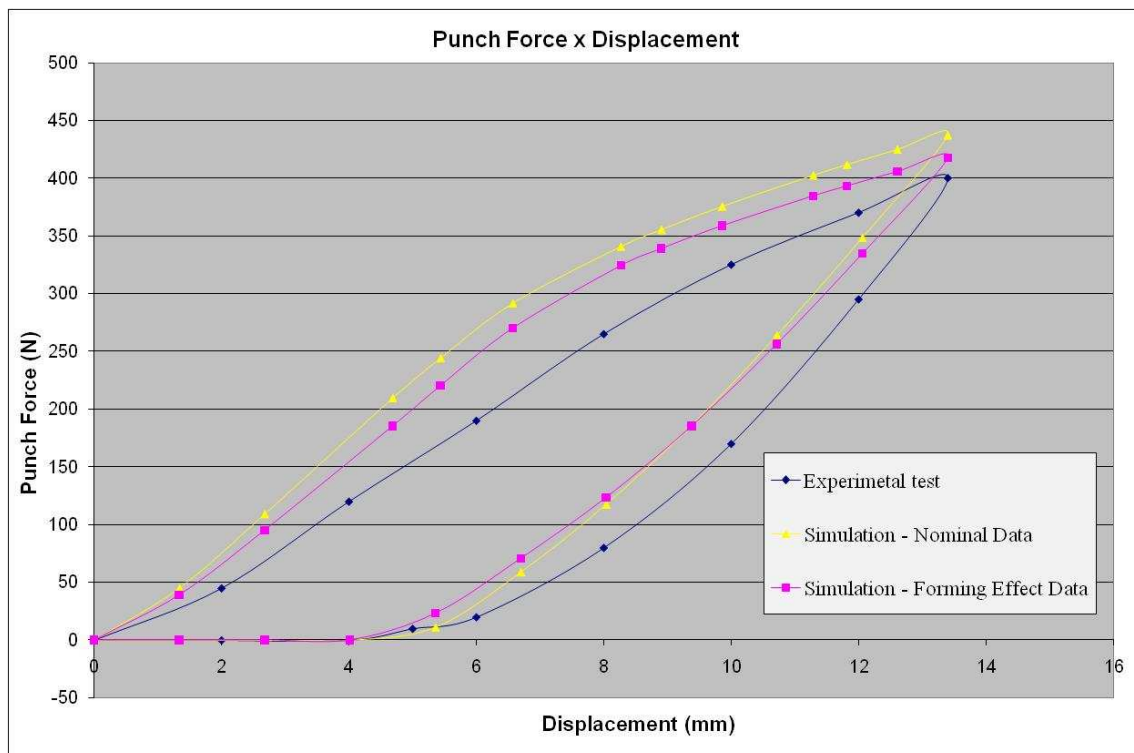


Figure 10 - Graph of punch force over displacement of indenter.

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 analyzed and results similar to this were found. 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.

5. REFERENCES

- Belytschko, T., Liu, W. K., Moran, B., 2000, " Nonlinear Finite Elements for Continua and Structures", John Wiley & Sons.
- Dagson, N., 2001, "Influence of the Forming Process on the Crash Response of a Roof Rail Component", M.Sc. Thesis, Department of Solid Mechanics, Linkping University.
- Dutton, T., Richardson, P., Knight, A., Sturt, R., 2001, "The influence of residual effects of stamping on crash results", 3rd European LS-DYNA Conference. Paris.
- Gaddekar, G.B., Raj, P., Kshirsagar, S., Anilkumar, C., 2006, "Inclusion of forming effect in crash simulations using one step forming simulations", Altair CAE User Conference - Innovation Through Simulation. August 3-5, Bangalore.
- Hoek, T. V., 2006, "The History Influence of Forming on the Predicted Crash Performance of a Truck Bumper", M.Sc. Thesis. Eindhoven.
- Kim, H., Hong, S., Hong S., Huh, H., 2003, "The Evaluation of crashworthiness of vehicles with forming effect", 4th European LS-Dyna Users Conference.
- Lanzerath, H., Ghouati, O., Wesemann, J., 2001, "Schilling, R., Influence of manufacturing processes on the performance of vehicles in frontal crash", 3rd European LS-DYNA Conference, Paris.
- Owen, D. R. J., Hinton, E., 1986, "Finite Elements in Plasticity: Theory and Practice", Pine-ridge Press Limited, Swanse.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.