Influence of computational parameters and nonlinear unloading behavior on springback simulation

Sérgio Fernando Lajarin & Paulo V. P. Marcondes

Journal of the Brazilian Society of Mechanical Sciences and Engineering

ISSN 1678-5878
Volume 35
Number 2

J Braz. Soc. Mech. Sci. Eng. (2013)
35:123-129
DOI 10.1007/s40430-013-0014-1

Volume 35 • Number 2 • June 2013

Journal of the Brazilian Society of

Mechanical Sciences and Engineering

Springer

ABCM
Brazilian Society of Mechanical Sciences and Engineering

Springer

J Braz. Soc. Mech. Sci. Eng. (2013) 35:123–129
DOI 10.1007/s40430-013-0014-1

TECHNICAL PAPER

Influence of computational parameters and nonlinear unloading behavior on springback simulation

Sérgio Fernando Lajarin · Paulo V. P. Marcondes

Received: 12 August 2011 / Accepted: 14 March 2012 / Published online: 26 March 2013

Abstract The advanced high-strength steels (AHSS) have become an interesting alternative to the automotive industry that aims to reduce the thickness of the components without compromising the structural performance and absorption capacity of impact. However, their mass utilization is still limited due to computational problems to predict the occurrence of the phenomenon of springback. Nonlinear unloading behavior following plastic deformation is identified as the factor that compromises the computational prediction of AHSS sheet forming. Moreover, it is observed sensitive dependence on correct choice of numerical parameters on the quality of the springback prediction such as the number of through-thickness integration points (NIP), coefficient of friction, the number of element on tool radius and simulation punch velocity. The aim of this study is to evaluate the degradation of elastic modulus and how the choice of numerical parameters can affect the computational prediction of AHSS springback. For this purpose, a user subroutine has been implemented in ABAQUS to describe the degradation of the modulus of elasticity during unloading, and several numerical parameters have been evaluated. It was possible to observe the influence of considering the degradation of elastic modulus during simulation and the large or small influence of some computational parameters.

Keywords Springback · AHSS · DP600 · Elastic modulus degradation · Simulation

List of symbols

e	Engineering strain (%)
n	Hardness coefficient
R	Tool radius
t	Sheet thickness (mm)
K	Plastic resistance constant (MPa)
μ	Friction coefficient
θ_1	Springback of wall opening angle (°)
θ_2	Springback of flange angle (°)
ρ	Sidewall curl radius (mm)
σ	True stress (MPa)
ε	True strain

1 Introduction

The automotive industry undergoes a constant pressure related to environmental requirements, which mainly aim the reduction of emission of greenhouse gasses in the atmosphere. An alternative found by the car manufacturers is to reduce the mass of the vehicle to lower fuel consumption and, therefore, greenhouse gas emissions. On the other hand, the automotive industry is part of a highly competitive market that requires cost reduction and improved safety features and performance. The advanced high-strength steels (AHSS) have become an interesting alternative to satisfy both requirements: it reduces the thickness of components without compromising the structural performance and absorption capacity of impact. However, the replacement of conventional steels by AHSS implies changes in the forming process planning and

---

Technical Editor: Alexandre Abrão.

S. F. Lajarin (✉) · P. V. P. Marcondes
Departamento de Engenharia Mecânica, Universidade Federal do Paraná, Curitiba, PR Caixa Postal 19011, Brazil
e-mail: espanhol@ufpr.br

P. V. P. Marcondes
e-mail: marcondes@ufpr.br

design of the tools. In addition, the stamping springback is the major problem to be solved in these steps.

Springback is an elastically driven change in shape and form of a part upon unloading after the part is formed, and their prediction by means of finite element methods (FEM) does not show good results because of the difficulty of describing the behavior of these steels during plastic strain and also by the sensitivity setting of computational parameters.

Nonlinear unloading behavior following plastic deformation is identified as the factor that compromises the computational prediction of AHSS sheet forming and has been widely researched. Morestin et al. [2] investigated the elastic modulus decrease of diverse kinds of steel after plastic strain and found that the elastic modulus can decrease 17.5 % of its value at only 5 % of plastic prestrain for high-strength steel. Cleveland and Ghosh [3] noticed that inelastic strain released from the formed state could be a major source of additional strain recovery. They established that for 7 % of plastic prestrain, the elastic modulus can lose 19 % of its value for high-strength steel. Perez et al. [10] and Cobo et al. [4] explained the elastic modulus decrease in high-strength steel in terms of microplastic strain. During tensile plastic strain, many dislocations are created and pushed, which causes pile-ups, tangles and trapped into high-density dislocation walls to form. Many of these dislocations, especially those forming part of pile-ups, are repellent in character and only the applied stress keeps them very closed to each other. When the stress drops, these dislocations go back; this produces an extra strain and thus, a decrease in the measured elastic modulus. Yang et al. [14] explained that the elastic modulus decrease is due to microcracks created by the hardening. They observed that the elastic modulus is not uniform in grain and the main elastic modulus decrease is located at the grain boundary. They concluded that the movement of the mobile dislocations and the dislocations pile-up is the major source of the elastic modulus decrease.

Moreover, it is observed sensitive dependence on correct choice of numerical parameters on the quality of the springback prediction such as the number of through-thickness integration points (NIP) [11], the integration algorithm [8, 9], the number of element in radius tool [7] and simulation velocity [13] (Firat et al. [12]). Li et al. [5] explored a variety of issues in the springback simulations and they conclude that more contact nodes are necessary for accurate springback simulations than for forming simulation, approximately one node per 5° of turn angle versus 10° have been recommended for forming. However, Meinders et al. [7] have argued that an accurate result of springback can be achieved with 9 elements in contact with the tool radius—one node per 10°. Li et al. [5] reported that three-dimensional nonlinear shell and solid elements are preferred for springback prediction. Wagoner and Li [11] evaluated the integration points through thickness in the simulation of springback in draw bending of high-strength aluminum sheet and conclude that to analyze the springback with 1 % numerical error it is required up to 51 points through-thickness integration points for shell/beam type elements, and more typically 15–25 points, depending on *R*/*t*, sheet tension and friction coefficient. Xu et al. [13] contradicts saying that too many or too few number of integration point has disadvantage for the explicit solution in springback simulation and usually seven integration points is the best value. They also studied different punch speeds on draw bending of a U-hat model and concluded that the speed cannot exceed 1 m/s. In agreement with this result, Firat et al. (2007) also showed better results in simulations with similar speed.

Faced with the need to accurately simulate the phenomenon of springback, the main purpose of the present study has been to evaluate the degradation of elastic modulus and how the numerical choice of parameters can affect the computational prediction of AHSS springback. For this purpose, a user subroutine has been implemented in ABAQUS to describe the degradation of the modulus of elasticity during unloading, and several numerical parameters have been evaluated. It was possible to observe the influence of considering the degradation of elastic modulus during simulation and the large or small influence of some computational parameters.

2 Methodology

The FE analyses of Numisheet'93 U-channel forming process have been performed in the Abaqus software (Fig. 1a). The channel geometry and the forming process investigated in this paper may be found in [6]. This model is used because it can simulate similar forming conditions that occur in the industry. The blank geometry is a metal strip of size 300 × 35 × 1.5 mm. Due to the symmetry conditions, only the half of the tooling and blank were included in the simulations with appropriate boundary conditions.

To reduce the processing time, shell elements (S4R) with four nodes and six degrees of freedom were used to describe the blank [1]. The punch, die and blank holder were considered as rigid elements. The computational and process parameters set as default were: punch velocity of 1 m/s, 9 elements in contact with tool radius, blank holder force of 2.5 kN, 9 integration points through thickness and clearance between punch and die of 1.5 t. The contacts between the rigid bodies—punch, die and blank holder—and sheet surfaces were defined by the penalty contact method using a friction coefficient of 0.1—Coulomb's

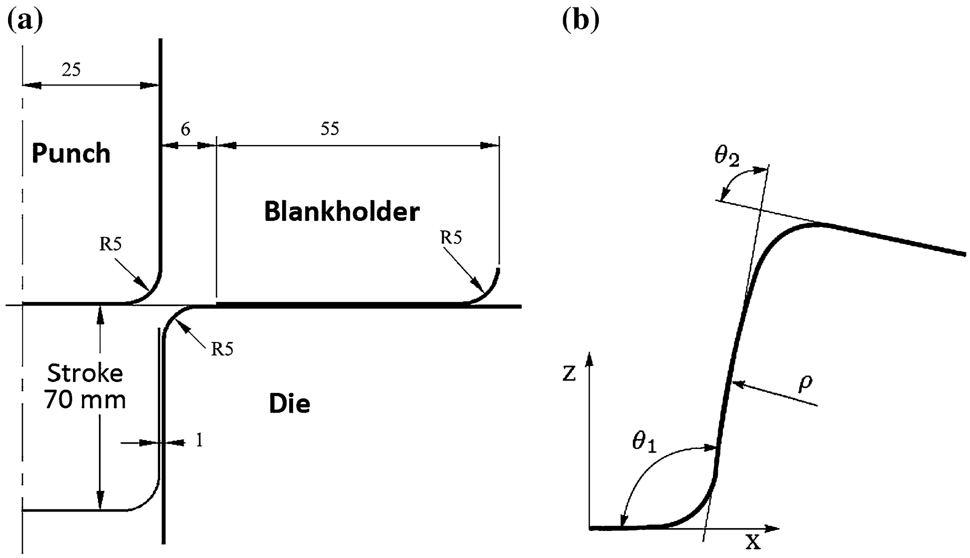

Fig. 1 Numisheet'93 U-channel tool: **a** tool design and **b** springback measured parameters

Table 1 DP600 mechanical properties

Material	Yield strength (MPa)	Tensile strength (MPa)	Elong. (%)	*K* (MPa)	*n*	Density (Kg/m^3)	Poisson	Initial elastic modulus (GPa)
DP600	385	602	23	975	0.17	7,800	0.3	208

model. In this work, each of these parameters has been changed to assess their influence on springback response during simulation.

Three measurements, namely, the springback of wall opening angle, the springback of flange angle and sidewall curl radius—shown in Fig. 1b—have been used to characterize the total springback. Only the cross-sectional shapes of formed parts were considered.

3 Material

The material used for the springback simulation was an AHSS DP600—thickness of 1.5 mm. It is a material of great interest in automotive industry for body structural parts. The material constants are summarized in Table 1. The true stress–strain profile was described with Power Law (Fig. 2). The material elasticity properties were assumed to be isotropic and von Mises yield function was used to describe the sheet metal deformation.

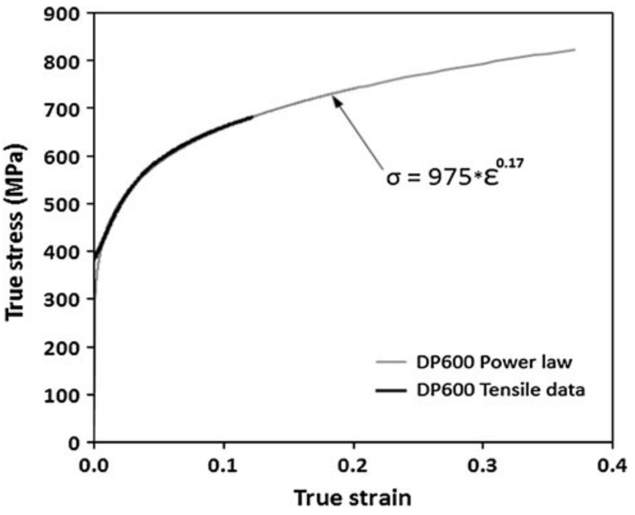

Fig. 2 DP600 uniaxial tension test—obtained by the extrapolated Power Law

The Poisson's constant and the initial elastic modulus were given as 0.3 and 208 GPa, respectively. In order to evaluate the influence of nonlinear unloading behavior, a user subroutine was utilized with the objective to analyze the influence of the variation of the modulus of elasticity with plastic strain. The subroutine used was based on a model available in [1]. The modulus of elasticity variation tests were performed with similar procedure described by Cleveland and Ghosh [3], and Perez et al. [10]. Starting from the initial state ($e = 0$ %) the specimens were subjected to uniaxial tensile test until the plastic strain was introduced, i.e., allowing that the initial elastic behavior was recorded. Subsequently, the elastic behavior was also recorded during unloading. This cycle was repeated every 24 h increasing the deformations in the following percentages: 0.5, 1.5, 3, 9 and 13 %. The tests were performed on a universal tensile machine with extensometer of 50 mm stroke. A graphic of the variation of the modulus of elasticity as a function of different true strains was plotted and their points were inserted into the Abaqus software as a data table to be interpolated by the user subroutine.

4 Results

4.1 Influence of the computational punch velocities

At each step of simulation of an explicit model a virtual speed should be set. It will mean the speed at which step will be executed—this parameter shows impact on the efficiency of calculations. In the simulation of U-draw bending hat a too large punch velocity will increase the dynamic effect in simulation, because the stamping is a static or quasi-static process, while too small punch velocity will increase the CPU time. Three computational punch velocities of 1, 5 and 10 m/s were used to study the influence of punch velocity.

Figure 3a shows the results of the change angle and sidewall curling for different punch velocities and with and without the use of a user subroutine. It can be found that the increase of the forming velocity does not really affect the θ_2. However, the forming velocity showed a significant effect on the springback on the wall (θ_1). The ρ also showed some variation with the punch velocity ranged between 1 and 5 m/s, but has leveled up to 10 m/s. In other words, punch velocity less than 5 m/s makes the blank more curved. The variation of elastic behavior—through the user subroutine implementation—showed influence mainly on the results of θ_1 and ρ. Figure 3b shows the component shape after springback.

4.2 Influence of the number of element in contact with the tool radius

The size and number of elements in the contacting area showed direct influence on the stress field after forming. The contact area on the radius of the tooling will represent, generally, the effects of the bending—that drives the springback. To assess the influence of the number of elements in the contact area—sheet and tool radius—five simulations were performed using 5, 7, 9, 13, 18, 25 and 31 elements in the contacting area of the tool radius with the sheet.

Figure 4a shows the springback with different number of element in contact with the tool radius. Figure 4b shows the part shape after springback. As can be seen, the springback is greatly influenced by the number of elements in contact area. A larger number of elements show higher θ_1 and ρ and lower θ_2—the springback was more pronounced. With 18 elements in contact area with the radius the springback stabilizes and according to Li et al. [5] is the indicated value for springback simulation.

4.3 Influence of the number of integration points (NIP)

In order to reduce the computational time on sheet metal forming simulation, the shell element is usually used instead of solid elements. In the numerical simulation of the shell element, integration points should be used on the middle plane in the thickness direction. In order to study how the choice of NIP may influence on springback, the integration points of 3, 5, 7, 9, 15, 25 and 31 were analyzed (Fig. 5).

Figure 5b represents the part shape after springback obtained with different NIP. It can be seen that the

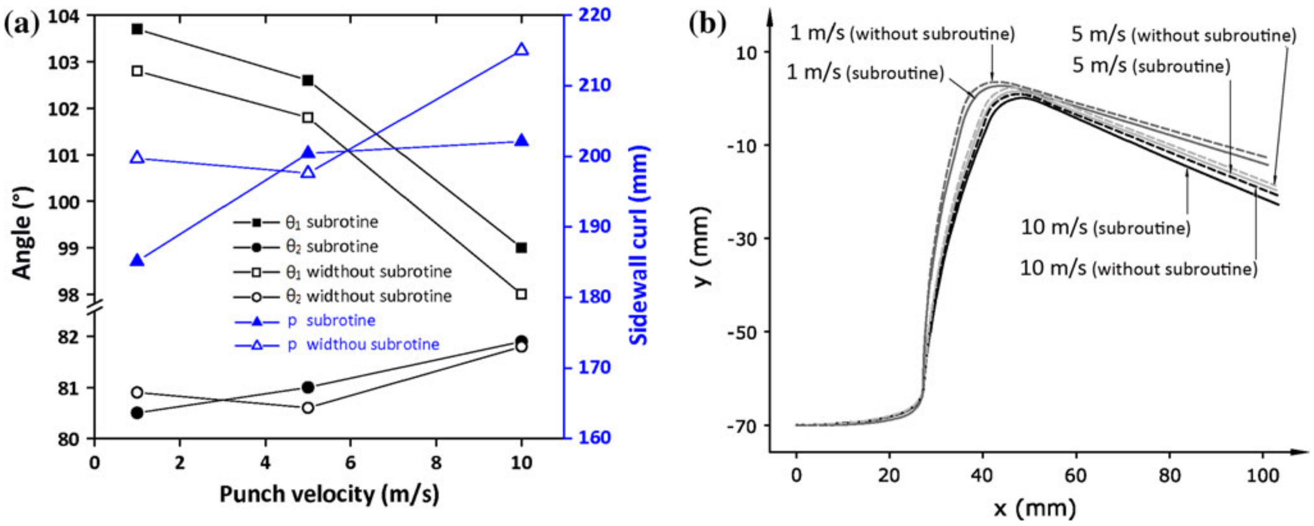

Fig. 3 Influence of computational punch velocity on springback: **a** springback variation and **b** part shape after springback

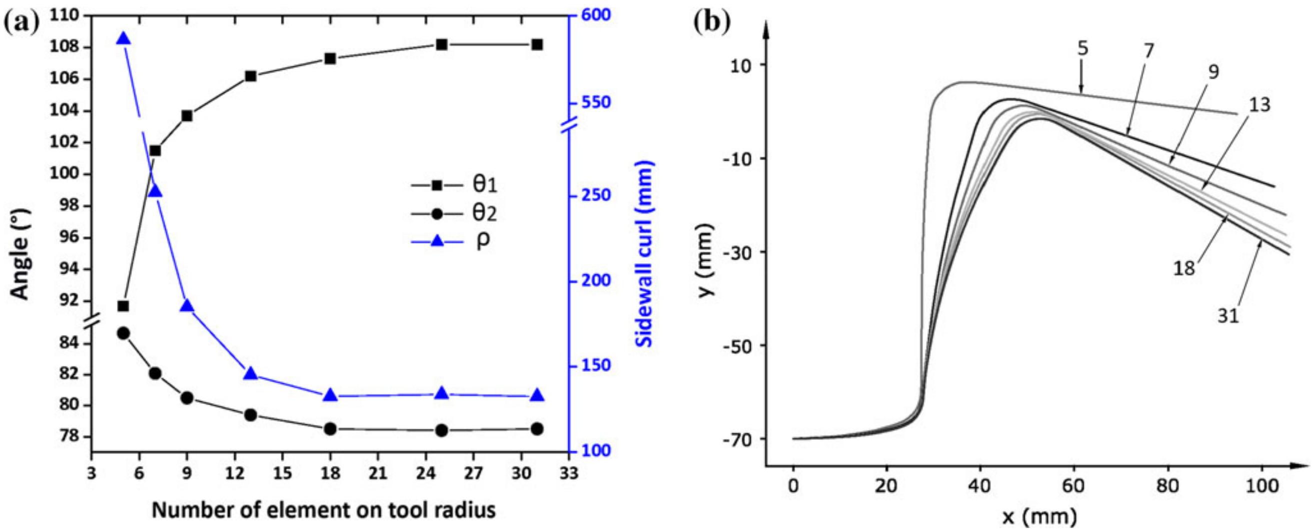

Fig. 4 Influence of the number of element in the contact area with the tool radius: **a** springback variation and **b** part shape after springback

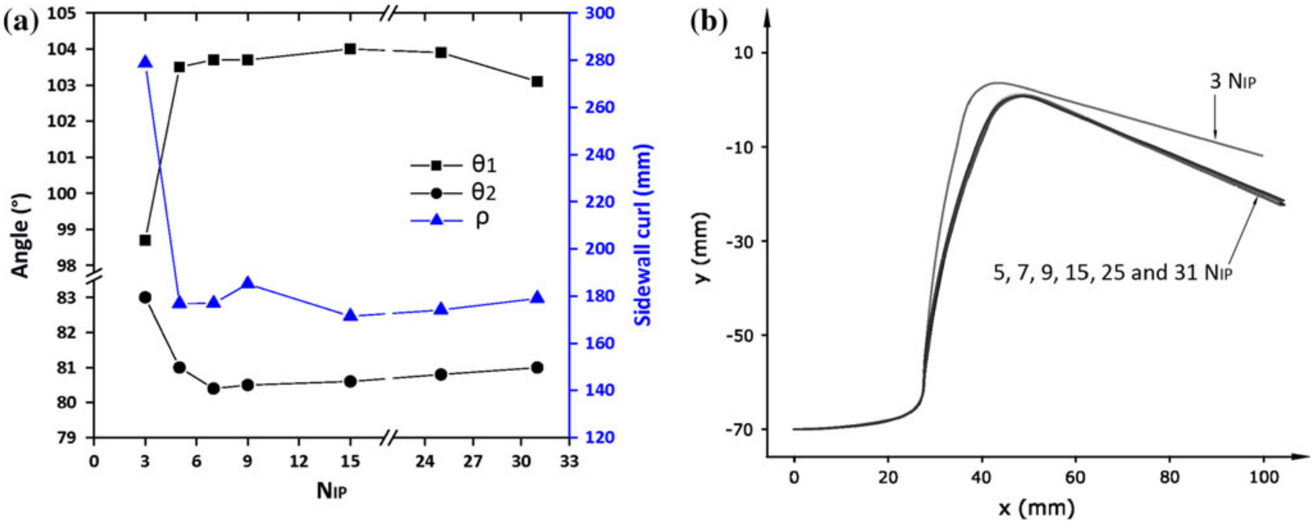

Fig. 5 Influence of NIP on the springback: **a** springback variation and **b** part shape after springback

springback was little influenced by the NIP. The ρ and θ_1 stabilized after 5 NIPs and θ_2 after 7 NIPs. In resume, three integration points are not enough, five integration points are the minimum acceptable and seven integration points can propitiate the best solution—increasing the number of integration points also increases the CPU time. The utilization of NIPs 7 and 31 spent, respectively, 738 and 2,943 s for simulation—increase on time of 398 %.

4.4 Influence of the coefficient of friction

Friction is an important factor influencing springback but the full behavior is still unknown. Friction coefficient is a difficult parameter to measure experimentally, probably, because it is different on the flat and curved parts of the punch and die. So it is very common—on sheet forming simulation—to use everywhere in the model just one friction coefficient, defined by Coulomb's model. To analyze the influence of the friction factor on springback, the coefficients of 0, 0.025, 0.1, 0.144 and 0.2 were evaluated.

Figure 6a shows that θ_1 and ρ are strongly influenced by the friction coefficient. The higher the value, the greater the wall opening angle and the sidewall curl is increased. The increased friction between the sheet surfaces and the die produces a greater flow of tension causing compression on the outer surface of the sheet—sidewall curling is observed. However, θ_2 showed little influence of the friction factor, because the slip is small in that region of the sheet—radius of the die—unlike what happens with the wall region. Figure 6b represents the component shape after springback with different friction factors showing that the friction is a very sensitive parameter.

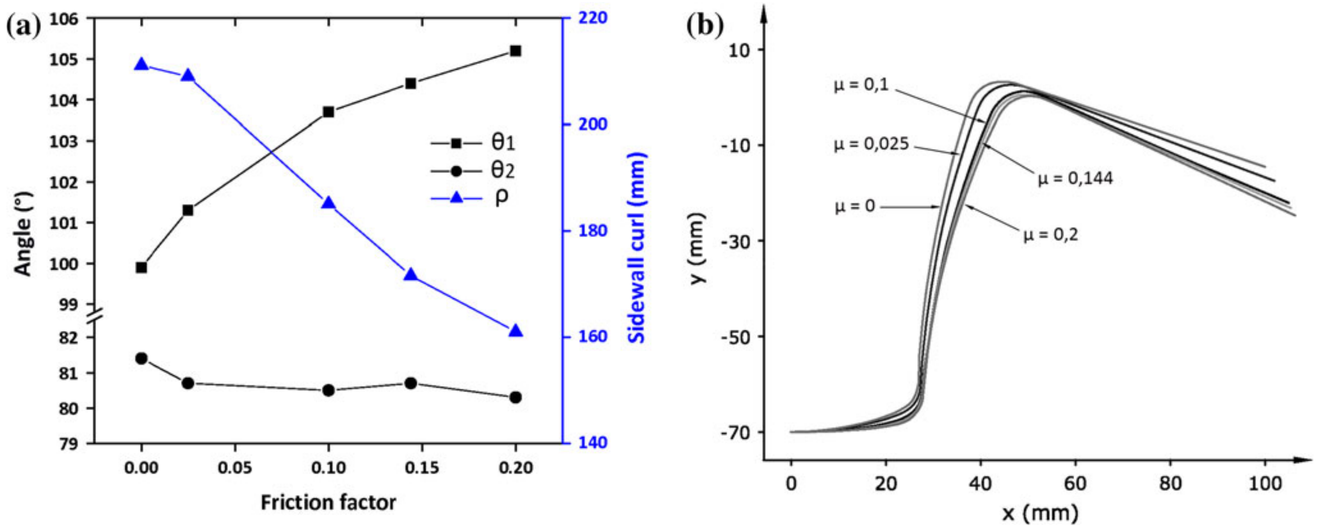

Fig. 6 Influence of friction coefficient on springback: **a** springback variation and **b** part shape after springback

5 Discussion

This work found that the punch velocities used in the simulation shows a considerable influence on springback results and according to Xu et al. [13] and Firat et al. (2007), 1 m/s is the best option. In addition, the number of elements in contact with the tool radius showed great influence on springback. With 18 elements in contact area the springback stabilizes and according to Li et al. [5] is the indicated value for springback simulation. In addition, the friction coefficient showed strong influence on springback and the value used should be exactly the same observed experimentally between the sheet and tool. In contrast, the number of integration points (NIP) showed little influence on springback. Three integration points are not enough, five integration points are the minimum acceptable and seven integration points can propitiate the best solution.

6 Conclusion

In order to improve the quality in predicting, the springback phenomenon is needed to better describe the behavior of the material and choose the appropriate computational parameters. In this study, it was observed that the nonlinear unloading behavior described by means of a user subroutine showed an improvement in the results. It should be highlighted that the results of the present work are only valid for experiments using DP600 steel.

Regarding the choice of computational parameters, the following conclusions can be pointed out:

- This work found that the simulation speed shows a considerably influence on springback and according to Xu et al. [13] and Firat et al. (2007), 1 m/s is the best option.
- The number of elements in contact with the tool radius showed great influence on springback. With 18 elements in contact area the springback stabilizes and according to Li et al. [5] is the indicated value for springback simulation.
- NIP showed little influence on springback. Three integration points are not enough, five integration points are the minimum acceptable and seven integration points can propitiate the best solution.
- The friction coefficient showed strong influence on springback and the value used should be exactly the same observed experimentally between the sheet and tool.

Acknowledgments The authors thank the financial support from CNPQ and CAPES Agencies for the scholarships (Brazil) and the steel samples supplied by Usinas Siderúrgicas de Minas Gerais S.A. (USIMINAS).

References

1. ABAQUS/CAE (2009) User's Manual, Version 6.9
2. Morestin F, Boivin M (1996) On the necessity of taking into account the variation in the young modulus with plastic strain in elastic-plastic software. Nucl Eng Des 162:107–116
3. Cleveland R, Ghosh AK (2002) Inelastic effects on springback in metals. Int J Plast 18:769–785
4. Cobo R, Pla M, Hernández R, Benito JA (2009) "Analysis of the decrease of the apparent young's modulus of advanced high strength steels and its effect in bending simulations". In: Proceedings of the IDDRG 2009 International Conference, Golden, CO, USA
5. Li KP, Carden WP, Wagoner RW (2002) Simulation of springback. Inter J Mech Sci 44:103–122

6. Makinouchi AE, Nakamachi E, Onate E, Wagoner RH (eds) (1993) NUMISHEET'93. "The Institute of Physical and Chemical Research"
7. Meinders T, Konter AWA, Meijers SE, Atzema EH, Kappert H (2006) A sensitivity analysis on the springback behaviour of the unconstrained bending problem. Int J Form Proc 9(3):365–402
8. Narasimhan N, Lovell M (1999) Predicting springback in sheet metal forming: an explicit to implicit sequential solution procedure. Finite Elem Anal Des 33:29–42
9. Papeleux L, Ponthot J-P (2002) Finite element simulation of springback in sheet metal forming. J Mater Process Technol 125–126:785–791
10. Perez R, Benito JA, Prado JM (2005) Study of the inelastic response of TRIP steels after plastic deformation. ISIJ Inter 45:1925–1933
11. Wagoner RH, Li M (2007) Simulation of springback: through-thickness integration. Int J Plast 23:345–360
12. Firat M, Mete OH, Kocabicak U (2007) A sensitivity analysis of springback deformations in stamping process. 4th International Conference and Exhibition on Design and Production of Machines and Dies/Molds, Cesme, Turkey
13. Xu WL, Ma CH, Li CH, Feng WJ (2004) Sensitive factors in springback simulation for sheet metal forming. J Mater Process Technol 151(1–3):217–222
14. Yang M, Akiyama Y, Sasaki T (2004) Evaluation of change in material properties due to plastic deformation. J Mat Proc Tech 151(1–3):232–236