Numerical Simulation of Springback for Transiting Part of Combustion Chamber

Ba Nyar Thein, Winn Wah Wah Aye
Department of Airport Management, Myanmar Aerospace Engineering University

***Abstract*- Springback is an important factor influencing the quality of formed sheet metal part in sheet metal forming process. Geometrical changes occur when the forming load is removed since the elastic deformation is recovered. Analytical methods have been used for analyzing parts of two-dimensional or simple three-dimensional geometry. However, for complex three-dimensional parts, it is almost impossible to apply analytical methods. Therefore, numerical simulation of finite element method (FEM) have been developed and widely used in sheet metal forming as a result of rapid progress of computational time and costs.**

The influence of numerical parameters such as friction coefficient, die corner radius and integration points on springback for the aerospace part is considered in this work. In order to characterize the materials, a mechanical test is conducted. That test include standard uniaxial tension test. Furthermore, since process conditions have an obvious effect on springback and one of the most important of these is the stabilization effect. The effect of anisotropy and combined hardening model was applied for the aerospace part and reasonable values are proposed.

***Keywords* - Springback, Combined hardening, Friction Coefficient, Integration Points, Die corner radius**

I. INTRODUCTION

Springback is an important factor influencing the quality of formed sheet metal part in sheet metal forming process. Geometrical changes occur when the forming load is removed since the elastic deformation is recovered. Analytical methods have been used for analyzing parts of two-dimensional or simple three-dimensional geometry. However, for complex three-dimensional parts, it is almost impossible to apply analytical methods. Therefore, numerical simulation of finite element method (FEM) have been developed and widely used in sheet metal forming as a result of rapid progress of computational time and costs.

The application of the finite element simulation for metal forming applications began as a pre-industrialization time before 1990. An important breakthrough occurred when successful forming simulation codes suitable for handling real industrial problems appeared on the market [1]. In 1994, M. Karima discussed the opportunities and challenges in the implementation of Sheet Metal Stamping Engineering Technologies (SMSET), from the user, trainer and developer perspectives. The technology includes all science based aspects that would enhance the understanding and the engineering of metal stampings [2]. To look at the present situation of finite element simulation system introduced to industry and to reconsider real requirements from industrialengineers, which may give important information to a researcher for further development of a finite element system was purposed by A. Makinouchi [3]. Today, several commercial codes are already available for forming simulation [4] to [6].

Springback occurs when stamped sheet components are released from forming tools. It means that the final shape of the drawn part will depart from the shape imported by the forming tool. In many industries, springback plays an important role in stamping field. To meet the needs of the manufacturing industry; prediction of defects and modification of design in the design stage are needed, as well as in order to reduce development costs and lead times. The accuracy of springback prediction is affected by many influence parameters in both forming and springback process. Many researchers [7-11] tried to investigate the numerical factors which influence the accuracy of springback simulation by using the different ways. The accurate simulation of springback, however, has still been proven difficult. Springback simulation is the last step of numerical simulation of sheet metal forming, consequently, any calculation errors resulting from previous simulation of forming processes will be accumulated and influence the springback analysis. Therefore, the accuracy of springback simulation is not only related to springback analysis itself, but also strongly dependent on the accuracy of forming processes [12].

In this paper, FEM simulation of Transition part of combustion chamber forming process is mainly performed by commercial FEM code ABAQUS software. While it is possible to perform springback analyses within ABAQUS/Explicit, ABAQUS/Standard is much more efficient at solving springback analyses. Since springback analyses are simply static simulations without external loading or contact, ABAQUS/Standard can obtain a springback solution in just a few increments. [13] At the present, an investigation into the use of static implicit finite element analysis to analyze the springback process is presented. Several forming processes are analyzed using ABAQUS explicit finite element analysis. Influence factors such as friction coefficient, die corner radius, integration points and stabilization effects are studied.

II. BASIC CHARACTERISTICS AND STRESS-STRAIN CURVE OF MATERIAL

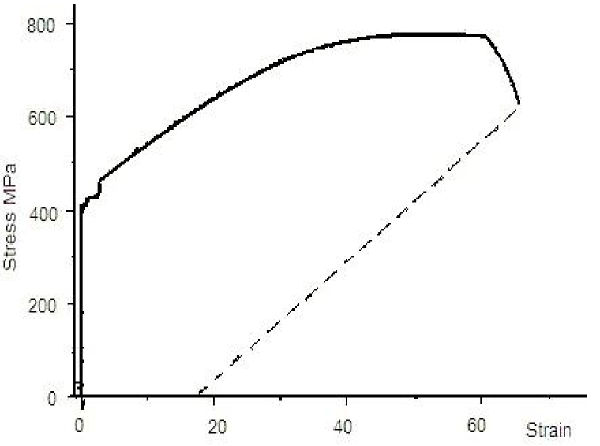


Figure 1. Experimental Stress-Strain Curve.

In metal forming processes using finite element code software, the first need to know the basic characteristic ofmaterial ,for example: Young's Modulus of Elasticity E, Yield Strength σ_s , Tensile Strength σ_b, Poisson's ratio μ, true stress-strain relation σ-ε and so on. Therefore, according to GB3076-82 'steel sheet metal traction test method' (national standard of China), unidirectional tensiletest was carried on material department's electronic tensile material testing machine (MTS) of Beihang University (BUAA) to get the test data. Engineering stress and strain are defined by the following equations,

$$\text{Engineering stress, } \sigma_{nom} = \frac{F}{A_0} \tag{1}$$

$$\text{Engineering strain, } \varepsilon_{nom} = \frac{L - L_0}{L_0} \tag{2}$$

where, $A_0$is the original cross-sectional area of the specimen, $L_0$is the original distance between the gage marks, and L is the distance between the gage marks after force F is applied. Material performance data is obtained as shown in Table. I. The stress-strain curve result as shown in Fig.1 by using the experimental data after testing.

TABLE I
BASIC MECHANICAL PERFORMANCE PARAMETERS

Material	Yield strength $\sigma_{0.2}$ (MPa)	Tensile strength σ_b (MPa)	Poisson's ratio, μ	Young's Modulus (MPa)
N 263	369.313	1176.782	0.371	132276

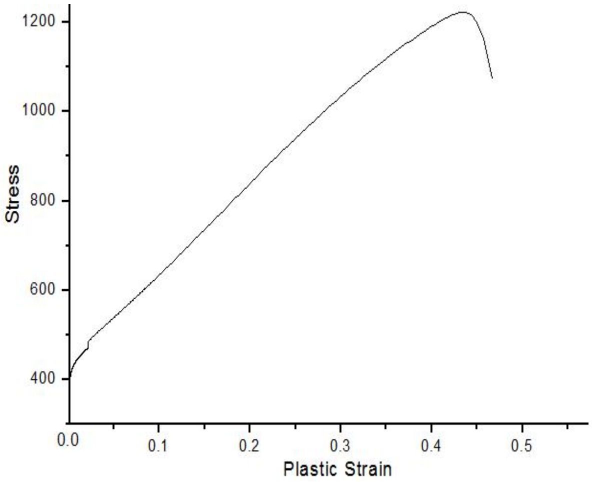


Figure 2. True Stress-Strain Curve.

The transiting part of combustion chamber component uses the sheet metal plate, N263. It is one kind of nickel heat-resisting alloy material, and thickness is 4.8mm. The material is assumed to follow a power law. True stress and true strain can be determined by the following equations:

$$\varepsilon = \ln\left(1 + \varepsilon_{nom}\right) \tag{3}$$

$$\sigma = \sigma_{nom}\left(1 + \varepsilon_{nom}\right) \tag{4}$$

where, ε is the true strain and σ is the true stress. True stress-strain curve is presented in Fig. 2.

III. NUMERICAL STUDY OF INFLUENCE FACTORS ON SPRINGBACK SIMULATION

In the implicit finite element analysis used in our simulations, the static equilibrium equations are solved by using the Newton-Raphson iteration method. The Newton-Raphson iteration technique operates as follows, Abaqus [2009]:

$$\vartriangle u^{i+1} = \vartriangle u^{i} + K_t^{-1}(F^i - I^i) \tag{5}$$

where$u$ is the nodal displacement vector, K_t is thecurrent tangent stiffness matrix, F is the external load vector, and I is the internal force vector. The internal force vector is calculated by summing the internal force vectors calculated from every element:

$$I = \sum_k Ik \tag{6}$$

whereas the I_k is calculated as:

$$I_k = \int B_k^T \sigma_k dV_k \tag{7}$$

Where B_k is the stress-displacement matrix for the kth element, σ_k is the stress tensor in vector form, and V_k isthe element volume.

Figure 3: shows a finite element model for the forming process. ABAQUS/Implicit, commercial finite element code was used for simulation. The material properties, the friction between the material and the tools and the blankholder force affect the deformation of the material. The tooling is composed of a punch, a die and blankholder; all tools are modeled as rigid bodies on the purpose of saving CPU time cost. The blankholder force (BHF) is varying during the FE analysis. The final punch stroke is taken equal to 246 mm. The material parameters are chosen to represent the N-263 metal. The blank is simulated using 4 node reduced-integration shell elements, (S4R). Coulomb friction model with various friction coefficients are used to simulate between the blank and the tools. The plastic behavior of the material is modeled using the Power law; with a strength coefficient, K of 1758.01266 MPa and a strain hardening exponent, n of 0.43566. Several integration points (Ip)are used to investigate the amount of deviated shape and then, suitable Ip is advised.

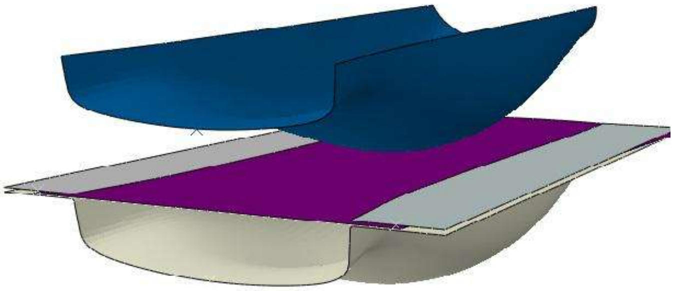


Figure 3.FEM model without blankholder.

A. *Influence of the friction coefficients on hardening model*

Friction is an important factor in sheet metal springback. We have to use the various friction coefficients such as 0.08, 0.12 and 0.15 on all part of in this forming process and study the results obtained from different values. Material modeling has been the important factor in numerical simulation of sheet metal forming, where, the combined hardening model is widely use to have an apparent influence on springback simulation.

Figure 4: represent the values obtained for the springback parameters at both end of the left and right of the part while using combined hardening. It is important to note that the values of the parameters for low coefficient of friction are made to get a large shape error and also the value of high friction coefficient may be a springback. However, coefficient of friction 0.12 and 0.15 are relied on the stable condition. According to these results, friction is a very sensitive parameter for this complex shape of the part.

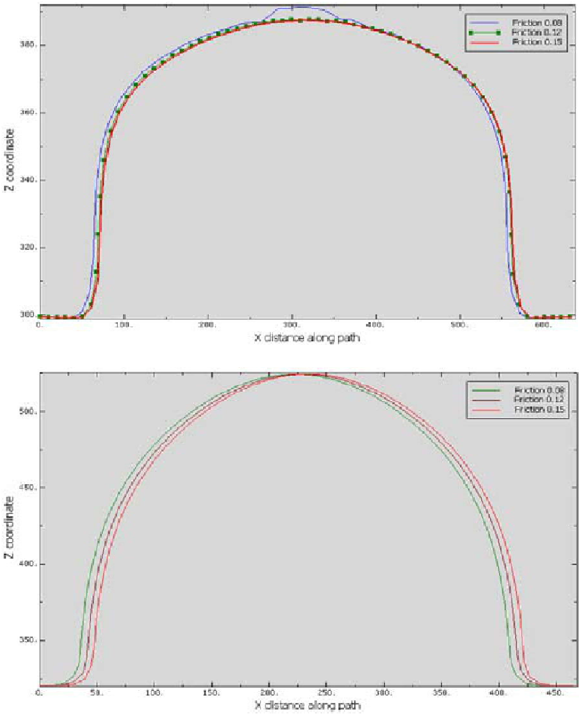


Figure 4.The effects of Friction Coefficients Curve forsimulated model.

B. *The influence of the die corner radius*

This study has focused on the springback occurring in the sheet metal deep drawing process by controlling die corner radius. Four types of corner radius 5, 10, 15 and 20 mm are used to describe the die corner. A smaller die corner radius is expected to be shape error and also larger die corner radius can be springback as shown in Figure 5(a) and(b).

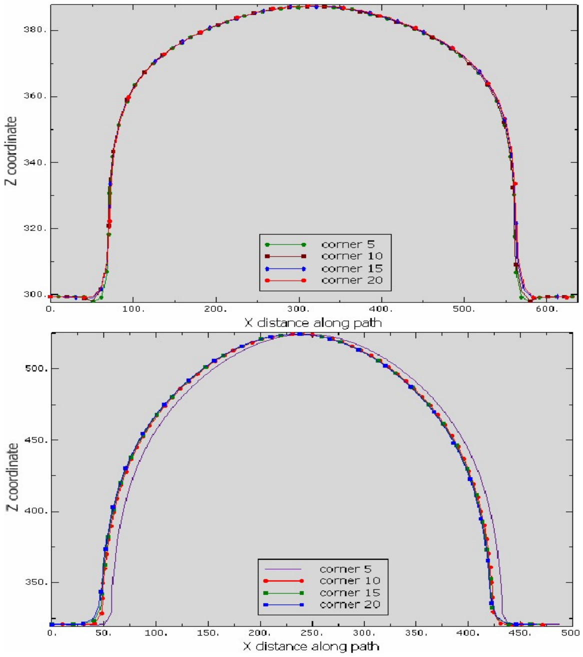


Figure 5.Comparison of die corner radius for springback.

C. The Influence of the Integration Points

In this section, the number of integration points through sheet thickness based on S4R element has been investigated in springback simulation. There are two types of integration schemes commonly recommended in element formulation; which are Gauss integration scheme and Simpsonintegration scheme. Simpson integration scheme is employed by ABAQUS element model due to its accurate description of numerical integration. For the influence of the number of integration points through thickness, there are many papers published, which were described the different number of through thickness integration points (varying from 5 to 51).

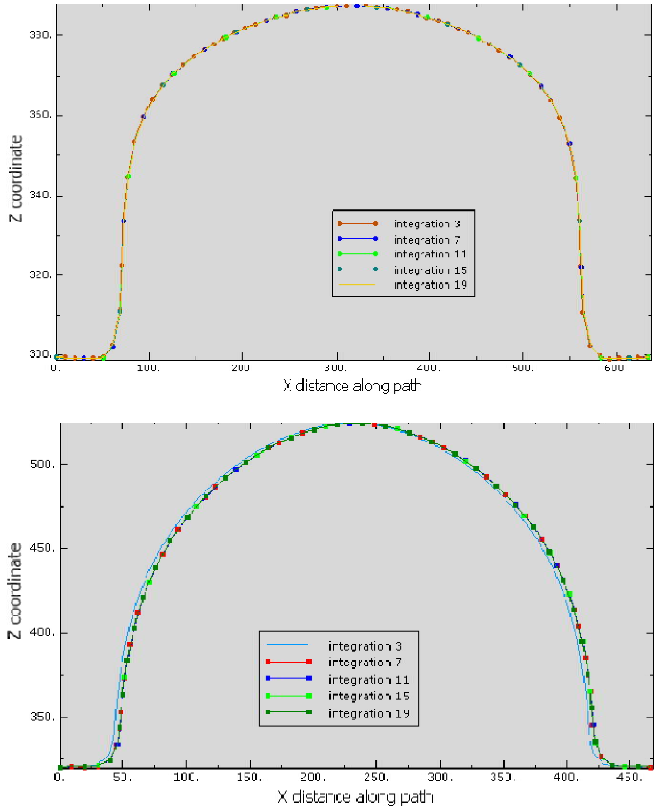
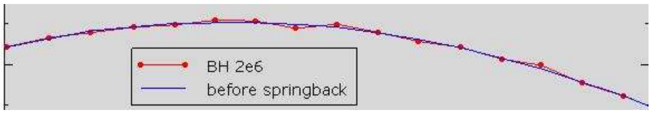


Figure 6.The influence of the number of integration pointsthrough thickness.

Figure 6: shows the simulated springback varies with number of integration points. It can be seen that poor springback result will be obtained when the number of integration points (Ip) is less than 5, however, when the Ip is 5, the simulated springback starts to be stable, and there is nearly no further improvement after Ip is over 5. Note that larger Ip will result in more time consumption. Seven integration points are allocated along the thickness direction of the blank to take up bending deformation effectively.

D. The Influence of the Blankholder Force

Effects of blankholder force on springback were also investigated. To examine this effect, several different BHFs were utilized from a relative low value 2e5 N to a quite high value of 10e6 N without material failure. When the blankholder force is larger than 10e6, fracture occurs in the part region. Figure 7: shows the effect of BHF on the springback for the simulated part. It is shown that increasing the BHF will result in a reduction in the springback. It wasobserved that the wrinkle occurred with the lower BHF was 2e6 (see section A-A for BHF 2e6).



(a) Section A-A for BHF 2e6

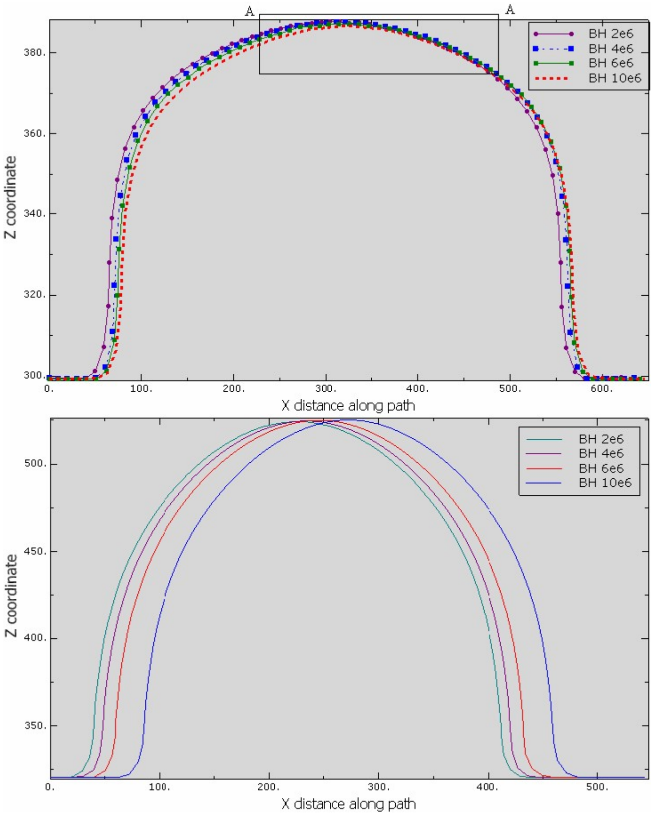


Figure 7.Effect of blankholder force (BHF) on springback.

When the blankholder force is too small, deformation is limited and often the desired final part shape cannot be achieved. In current investigation, the final shape of the part shape will remain normal between 4e6N and 10e5N.This phenomenon suggests that for a given material, tooling geometry and friction, there may be an optimal BHF to balance springback against failure.

IV. FINAL VALIDATION

Based on the above-mentioned investigations, the optimized numerical parameters, which intend to provide a robust and accurate springback simulation, are suggested.

TABLE II
OPTIMIZED NUMERICAL PARAMETERS USED IN SPRINGBACK SIMULATION

Process parameters	Value
Punch stroke	246 mm
Blankholder force (BHF)	4000 kN
Friction coefficient	0.15
Sheet thickness	4.8 mm
Integration point	7
Die corner radius	20 mm
Lankford value (R)	1.5479

Table II. shows the optimized numerical parameters suggested in this work, named as final validation settings. With these values, a robust and accurate springback simulation is expected to be obtained. Figure 8: illustrates the simulation result of springback based on the optimized numerical parameters.

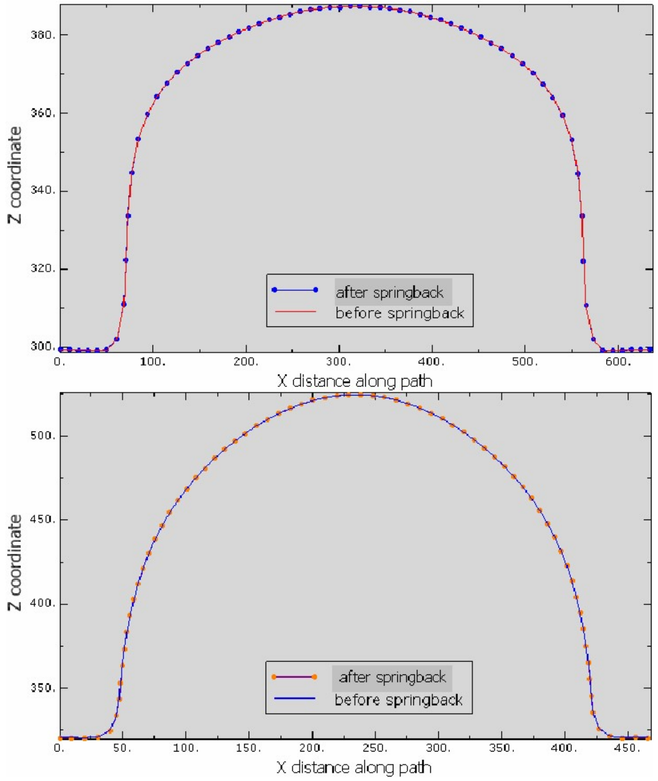


Figure 8.The simulation result of the springback for combinedmodel.

Springback analyses can suffer from instabilities that adversely affect convergence. Thus, include automatic stabilization to prevent this problem. Use the appropriate value for the dissipated energy fraction.

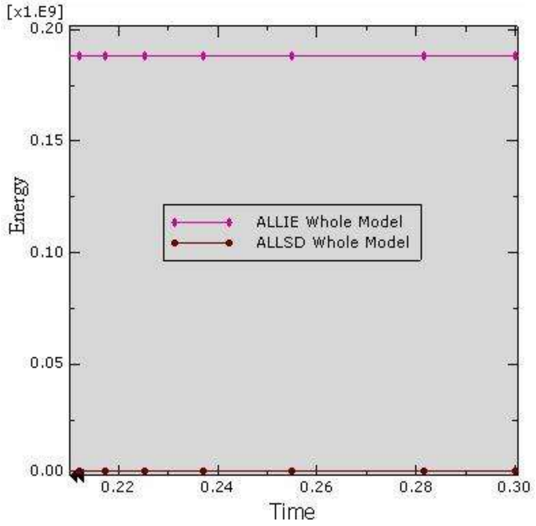


Figure 9.Comparison of ALLIE and ALLSD

We should also plot the blank's internal energy ALLIE and compare it with the static stabilization energy ALLSD that is dissipated. The stabilization energy should be a small fraction of the internal energy to have confidence in the results. Figure 9: shows a plot of these two energies; the static stabilization energy is indeed small and, thus, has not significantly affected the results.

The result of the springback is necessarily dependent on the accuracy of the forming stage preceding it. In fact, springback results are highly sensitive to errors in the forming stage, more sensitive than the results of the forming stage itself.

V. CONCLUSION

The springback simulation is an important process, which is not only influenced by springback computation itself, but also depends on the accuracy of previous forming simulation. To obtain the accuracy of springback calculation, there are many numerical parameters are influenced that it is not easy to obtain an accurate springback simulation.

In this paper, the influences of numerical parameters on springback simulation are investigated and the optimized numerical parameters are suggested.

REFERENCES

[1] M.Tisza,"Numerical modeling and simulation in sheet metal forming,"Journal of Materials Processing Technology 151 (2004) 58-62.
[2] M. Karima, "Practical application of process simulation in stamping," Journal of Materials Processing Technology 46 (1994) 309-320.
[3] A Makinouchi, "Sheet metal forming simulation in industry," Journal of Materials Processing Technology 60 (1996) 19-26.
[4] M.Kawka, T.Kakita, and A.Makinouchi, "Simulation of multi-step sheet metal forming processes by a static explicit FEM code," Journal of Materials Processing Technology 80-81 (1998) 54-59.

[5] Dong Hongzhi*, and Lin Zhongqin, "Investigation of sheet metal forming by numerical simulation and experiment," Journal of Materials Processing Technology 103 (2000) 404-410.

[6] T. Pepelnjak, G. Gantar, and K. Kuzman , "Numerical simulations in optimization of product and forming process," Journal of Materials Processing Technology 115 (2001) 122– 126.

[7] Sang-Wook Lee, Yoon-Tae Kim; A study on the springback in the sheet metal flange drawing, Journal of Materials Processing Technology 187–188 (2007) 89- 93

[8] Luc Papeleux, Jean-Philippe Ponthot; Finite element simulation of springback in sheet metal forming. Journal of Materials Processing Technology 125–126 (2002) 785–791

[9] R.H. Wagoner, M. Li; Simulation of springback: Through-thickness integration. International Journal of Plasticity 23 (2007) 345–360

[10] Peng Chen, MuammerKoc; Simulation of springback variation in forming of advanced high strength steels. Journal of Materials Processing Technology 190 (2007) 189–198

[11] M. Kadkhodayan; Numerical Prediction of Springback and Side-Wall Curl in U-bending of Anisotropic Sheet Metals.Iranian Journal of Mechanical Engineering Vol. 6, No. 2,December 2005

[12] Dorel Banabic; Sheet Metal Forming Processes, Technical University of Cluj-Napoca, Research Centre on Sheet Metal, Romania

[13] ABAQUS 6.9.1 User Manual.