VERIFICATION OF SPRINGBACK ANALYSIS ACCURACY IN DEEP DRAWING PROCESS WITH FOCUS ON INITIAL SETTING OF NUMERICAL SIMULATION

PAČÁK Tomáš, TATÍČEK František, CHRÁŠŤANSKÝ Lukáš

Czech Technical University in Prague, Czech Republic, EU
Tomas.Pacak@fs.cvut.cz, Frantisek.Taticek@fs.cvut.cz, Lukas.Chrastansky@fs.cvut.cz

Abstracts

Computation of the springback phenomenon in numerical simulations, together with its prediction, is a very complex subject. Final results of the springback analysis are highly dependent on the initial FE settings. Therefore, with the initial settings, final accuracy of the springback prediction can be influenced significantly. There are plenty of options in terms of initial FE settings of the numerical simulation, for better understanding, the options were divided into two major categories. First group defines the element type, BEM type (Bending Enhanced Membrane), EPS type (Elastic Plastic Shell) or modified shell element type. Second group defines the density of the meshing (radius penetration, max element angle, master element size, etc.). This article focuses on springback analysis comparison in AutoForm R6 based on various FE settings. For experiment, outer panel of fifth doors from SEAT Ateca was used.

Keywords: Springback, numerical simulation, large metal stamping, AutoForm

1. INTRODUCTION

During the deep drawing process, the punch is pushed into the die cavity, the blank is plastically deformed and the specific shape of the punch and the die is transferred to it. As soon as the loads are removed, the elastically-driven change of the product occurs. This phenomenon is called springback. Springback results into geometrical inaccuracies of the obtained product shape. In order to compensate the springback it is important to carefully consider all factors prior to the stamping process, otherwise a reject product occurs. Apart from the geometrical error, springback leads to increase in the costs in pre-production tools design. Tools design and tools construction are one of the most time consuming steps in new car type developing process. Therefore, finding an effective and reliable method for springback prediction is very important [1].

Nowadays, numerical methods are commonly used for process simulation and evaluation. Software like AutoForm, PAM-STAMP, etc. are powerful tools in the pre-production step. The problem is still in the accuracy of these methods, which is still far from exact. In terms of springback prediction and compensation, approximate accuracy of the numerical methods is up to 62%, depending on the simulation settings [1]. The accurate prediction and analysis of the springback is very important to the tools design and following tools production. If the experimental trial and error process were replaced by a reliable numerical procedure, the pre-production time and costs could be decreased drastically [2].

2. SPRINGBACK PREDICTION AND ITS ACCURACY

Even though the overall accuracy of the numerical simulation outputs is very high, use of the numerical simulation in springback prediction and compensation is not yet so optimal. When springback analysis or compensation is to be expected, the numerical simulation must be individually designed from the very beginning of the process. The reason is that the overall accuracy of the springback analysis, in comparison with the reference geometry, is highly influenced by the settings of finite element mesh. For example by element type, meshing tolerance, radius penetration, number of initial and final elements, etc. [2].

2.1. Simplification in numerical simulations

One of the reasons, why the accuracy of the springback analysis is still not perfect, is that nowadays software simplifies the problem computation. In order to reduce the complexities involved in the springback analysis, the following simplified assumptions are made [3].

- A rigid plastic, strain - hardening, and anisotropic material is assumed
- Bending deformation is expected to occur under the plane strain condition
- The Bauschinger effect and the strain rate are neglected to remain consistent in the deformation behavior between tension and compression
- Hill's theory of plastic anisotropy is used to describe the normal anisotropic characteristics of metal sheets

2.2. Initial finite element settings of the numerical simulation

In numerical simulations, the option to choose from various mesh settings is possible. For example, in software AutoForm, one has the option to choose from the predesigned methods of settings: Concept Evaluation (CE), Concept Evaluation+ (CE+) and Final Validation (FV). With these settings, the element type of the FEM is selected. Method CE uses the Bending Enhanced Membrane elements (BEM), where these are mostly applied for the first tryout simulations. In contrary CE+ and FV methods use Elastic Plastic Shell elements (EPS). Shells (classical or modified) are more accurate however additional time for the computation is required. The differences of the initial FE settings are presented in **Figure 1** and in **Table 1**.

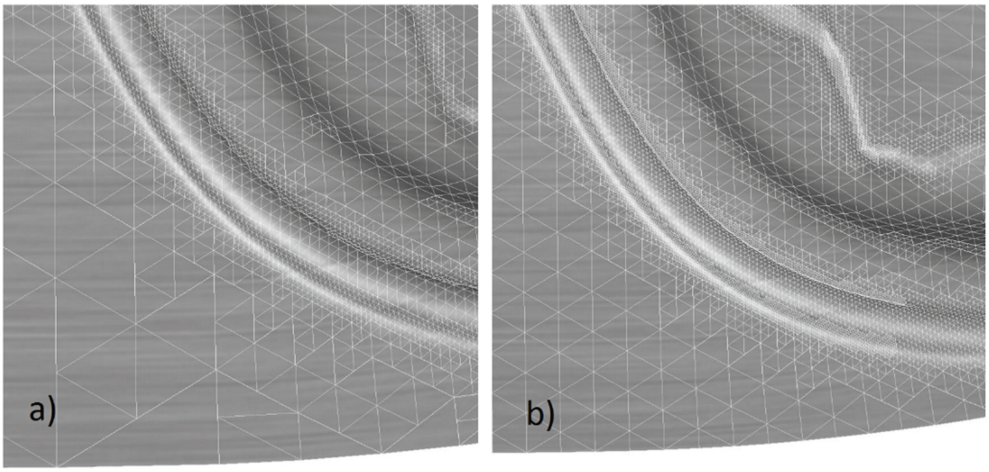

Figure 1 Example of the mesh differentness with use of various settings: a) CE and b) FV methods

Table 1 Verification of the various FE settings used for the numerical simulation

Setting Type	CE	CE+	FV
Element type	BEM-5	EPS-5	EPS-11
Tolerance Settings and Accuracy			
Stitching Distance (mm)	0.5	0.4	0.25
Meshing Tolerance (mm)	0.1	0.05	0.05
Max Side Length (mm)	50	30	10
Radius Penetration (mm)	0.22	0.22	0.22
Max Element Angle (°)	30	30	22.5
Max Refinement Level (-)	8	7	6
Initial Max element size (mm)	40	20	10
Min Element Size (mm)	0.31	0.16	0.16

Nowadays, shell elements are widely used in the sheet metal forming simulations. According to the various articles and guides, it is recommended to use the shell elements (classical or modified) for the springback analysis and for the following compensation. With the shell element type, it should be possible to reach the accurate and comparable results. Use of the membrane element type should in the contrary make the accuracy of the springback analysis less reliable [4, 5].

3. ACCURACY OF SPRINGBACK ANALASYS IN DEEP DRAWING EXPERIMENT

Previously commented statements about the FE settings should not be verified only in laboratory conditions (simple bending, cup drawing, etc.) but rather on the more complex parts (from real production). Based on this theory, experiment for varication was designed and later conducted on a car body part from SEAT Ateca. The part shown in **Figure 2** is an outer panel from the fifth door. Springback prediction with use of numerical simulations is not yet perfectly exact therefore it is only better to firstly conduct these experiments on parts with less complex geometry.

The panel used for the experiment is manufactured from a cold rolled low carbon steel DC05 with electrolytically coated zinc (R_e = 150 MPa, R_m = 293 MPa, n = 0.263, A_g = 22.4 %). Thickness of the part is 0.65 mm.

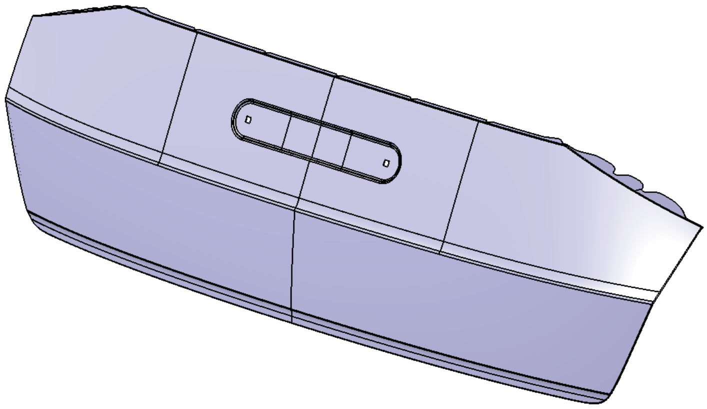

Figure 2 SEAT Ateca outer panel part used in experiment

3.1. General analysis of the simulations with various FE settings

At first, the numerical simulation was designed with use of AutoForm R6. The experiment was conducted only on the drawing operation. The reason is that here, the settings of FE influences the springback the most. In the simulation, various methods of settings were applied (CE, CE+ and FV). After the simulations had been computed, a general analysis was performed. The first analysis was focusing mainly on the distribution of the finite elements during the computation and its influence on the overall simulation. The following **Table 2** shows the general information about the performed simulations. In all three cases, simulation was solved with use of 8 Cores.

Table 2 General analysis of the simulation with use of various methods of settings

SEAT Ateca outer panel simulation	CE	CE+	FV
Number of Initial Elements	1 168	4 640	18 496
Number of Initial Nodes	637	2 425	9 457
Number of Final Elements	181 680	201 099	337 346
Number of Final Nodes	86 145	93 954	154 786
Computation Time (h:mm:ss)	0:46:57	1:05:01	1:36:53

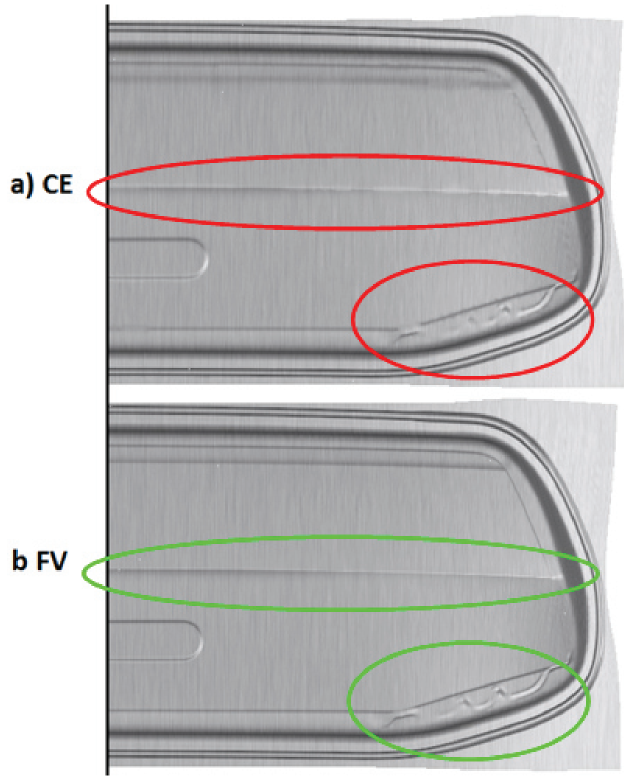

Figure 3 Surface analysis comparison between the CE and FV methods

Table 2 shows that the FE settings highly influences the number of finite elements, which results into growth of computation time and data consumption. The experiment was applied on the part with less complex geometry, where the time needed for the simulation does not play the most important role. In the contrary, when a simulation of e.g. side panel is required, the time needed for the computation is crucial (usually from 2 to 48 h). Besides the time and data usage, the FE settings influences also other parameters. For example quality of a surface is one of a many parameters, which is highly dependent of the mesh settings. Membrane elements are not suitable for the forming on small radiuses. **Figure 3** shows, how impropriate is use of the CE method with the membrane elements. On the first **Figure (a)**, poor surface results are shown. In the contrary, the simulation with the FV settings **(b)** shows ideal surface results without any roughening. Such an analysis is vital for the future tools design because the quality of the surface results must be taken into account.

3.2. Comparison of springback analysis with the reference part geometry from production

In order to compare the springback analysis correctly with the reality, reference geometry had to be selected. Best suitable option was use of the scanned part, taken from real drawing process. The press line had to be stopped after the drawing operation, so that the part could be taken from the production, scanned and then converted to CAD data. Following important step was making the decision which virtual springback analysis should be used. AutoForm offers various options for the springback analysis. For example “Free Springback” with no gravity taken into account or analysis with the gravity taken into account, such as “Real Measurement” or “Constrained Springback” (part is measured with use of clamps). Since the gravity plays serious role in the springback analysis, Real Measurement method was used.

Figure 4 shows the results from the Real Measurement analysis in the comparison with the scanned part taken from drawing operation. Straightaway from the springback analysis shown in **Figure 4**, one can notice, how the results vary, depending on the initial finite element settings. Visible differences can be seen between the membrane (CE) and shell (CE+ and FV) element type.

The amount of springback in normal direction was measured with the use of RPS (**Table 3**). RPS (Reference Point System) is a measurement tool used for 3D data which utilizes a unique method of measurement by establishing a reference point at the location of measurement. [6]

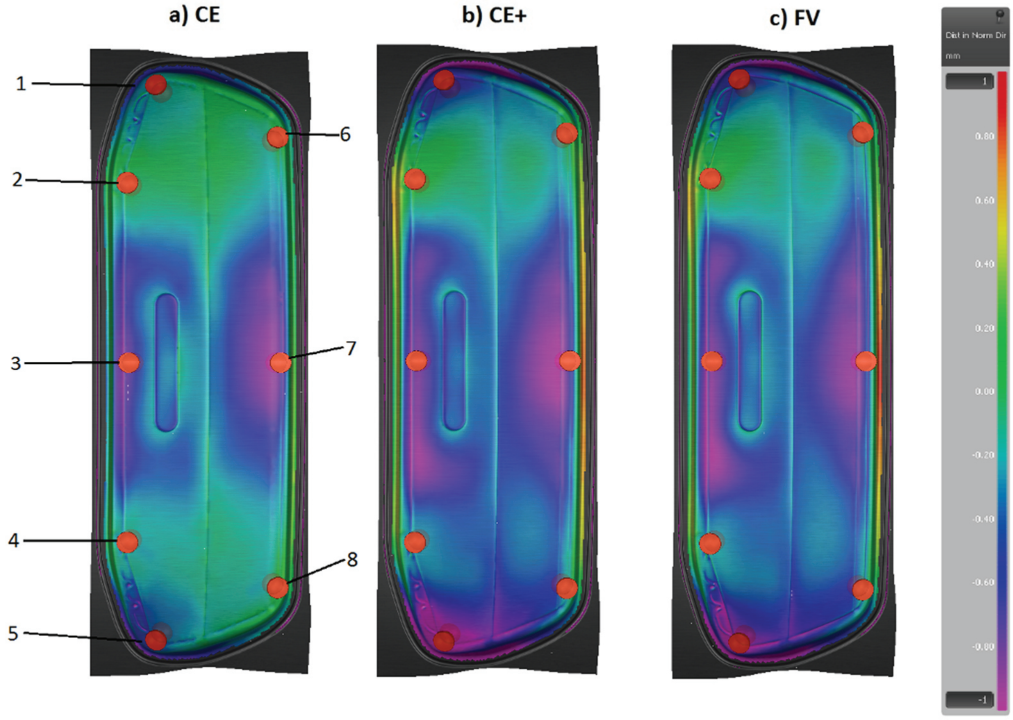

Figure 4 Comparison of springback analysis with use Real Measurement in RPS points

Table 3 Exact amount of springback with use of Real Measurement in RPS points

RPS point	Change in Normal Direction [mm]		
	CE (BEM)	CE+ (EPS)	FV (EPS)
1	-0.15	-0.31	-0.25
2	0.07	-0.08	0.03
3	-0.54	-0.83	-0.78
4	-0.22	-0.29	-0.28
5	-0.37	-0.91	-0.72
6	-0.15	-0.26	-0.34
7	-0.92	-0.82	-0.91
8	-0.12	-0.65	-0.55

As was mentioned in the beginning of this article, shell elements are recommended for the springback analysis and its following compensation. Yet this experiment proved that even with the membrane elements, accurate results can be achievable. In the experiment, BEM element type together with the coarse mesh settings achieved even more accurate results when compared with the scanned part. From the both CE+ and FV methods (both use shell element type), FV was slightly more accurate. Nevertheless, the results are almost identical, even though the mesh settings is not equal. The same experiment was conducted with outer panel of ŠKODA Superb fender, with very similar results (BEM element type showed more accurate springback analysis than settings with EPS element type) [4, 5].

4. CONCLUSION

Springback occurrence in sheet metal working is very common. Springback may be mitigated via changes in the forming process itself or by die design alterations. Unfortunately in terms of deep drawing and following forming operations, approach to the springback, its prediction and analysis is much more complex. One option, how to approach to the springback, is to use FEM as a foundation of many numerical simulations software. Nowadays numerical methods are very accurate in terms of process analysis. However in terms of springback analysis, accuracy is the major drawback of this method, since it is still far from exact [7].

Presented experiment was focused on the initial settings of numerical simulation through the FE mesh and its influence on the springback accuracy. In the experiment, outer panel from SEAT Ateca and later, outer fender from ŠKODA Superb, were used. In the experiment, numerical simulations with various settings were conducted, analyzed and later compared with the scanned parts as a geometry references. The comparison was performed on the semi-finished parts after the drawing operation. The experiment proved that in terms of surface quality, use of the shell elements and the better the mesh, the better are the results. But in terms of springback analysis, similar statement was not proved. The springback analysis was more influenced by element type, than by the density of the mesh. Moreover, simulation with the BEM element type shoved more accurate springback prediction, than simulations with EPS element type. Before it will be possible to reliably use specially designed standalone modules for springback compensation, e.g. AutoForm Compensator, the accuracy of springback analysis must be improved. The improvement must be endeavored both by developers and by customers, who work with numerical simulations in daily basis. In future, step by step methodology for working with springback analysis and compensation should be designed.

ACKNOWLEDGEMENTS

The research was financed by SGS16/217/OHK2/3T/12. Sustainable Research and Development in the Field of Manufacturing Technology references.

REFERENCES

[1] JINGJING, Xu, et al. Springback Prediction in Sheet Metal Forming Combined Finite Element Method with Date Mining Technique [online]. 2002, 9 [2016-06-03]. http://www.ansys.com/staticassets/ANSYS/staticassets/resourcelibrary/confpaper/

[2] *ASM Handbook: 14b Metalworking: Sheet Forming.* 14th edition. Ohio: ASM International, 2006. ISBN ISBN-13: 978-0-87170-710-9.

[3] LEU, D.K., ZHUANG Z. W. Springback Prediction of the Bending Process for High-Strength Steel Sheets. *Journal of Mechanical Science and Technology.* 2016, Vol. 30, No. 3, 1077-1084. DOI: 10.1007/s12206-016-0212-8. ISSN 1738-494x. http://link.springer.com/10.1007/s12206-016-0212-8

[4] JEONG-WHAN, Y, et. al. Springback Prediction For Sheet Metal Forming Process Using A 3D Hybrid Membrane/Shell Method. *International Journal of Mechanical Sciences* [online]. 2002, Vol. 21 [cit. 2016-04-10]. ISSN 2133 - 2153. http://mamel.snu.ac.kr/Papers/051%20Spring-back%20Prediction%20for%20Sheet%20Metal%20Forming%20Processes%20Using%20a%203D%20Hybrid%20Membrane%20Shell%20Method.pdf

[5] HAMA, T., BANU M., Effects of Finite Element Type on Springback Simulation of a High Strength Steel Sheet. In: *The Annals of Dunărea De Jos" University of Galaţi Fascicle V*, Technologies in Machine Building [online]. 2009, s. 6 [cit. 2016-04-10]. ISSN 1221- 4566. http://www.tcm.ugal.ro/tmb/2009/l13.pdf

[6] BENTHACK, Da., *Auswahlkonzepterstellung zur Einfuhrung eines optischen Messsystems zur Erfassung von Bauteil- oder Umformwerkzeuggeometrien im Presswerk und Betriebsmittelbau eines Automobilherstellers*. Hamburg, 2012. HAW Hamburg.

[7] *ASM handbook: Metalworking: Sheet Forming.* 14th edition. Materials Park, Ohio: ASM International, 2015. ISBN 16-150-3827-2, Page 23.