IOP Conf. Series: Materials Science and Engineering **280** (2017) 012021 doi:10.1088/1757-899X/280/1/012021

An Investigation and Prediction of Springback of Sheet Metals under Cold Forming Condition

A Elsayed[1], M Mohamed[1,2*], M Shazly[3] and A Hegazy[1]

[1] Department of Mechanical Engineering, Helwan University, Helwan, Egypt
[2] Impression Technologies Ltd, Coventry, UK
[3] Mechanical Engineering Department, The British University in Egypt, Cairo, Egypt

*Corresponding Author: m.mohamed@impression-technologies.com

Abstract. Low formability and springback especially at room temperature are known to be major obstacles to advancements in sheet metal forming industries. The integration of numerical simulation within the R&D activities of the automotive industries provides a significant development in overcoming these drawbacks. The aim of the present work is to model and predict the springback of a Galvanized low carbon steel automotive panel part. This part suffers from both positive and negative springback which physically measured using CMM. The objective is to determine the suitable forming process parameters that minimize and compensate the springback through robust FE model. The analysis of the springback was carried out following (Isotropic model and Yoshida – Uemori model) which are calibrated through cyclic stress strain curve. The material data of the Galvanized low carbon steel was implemented via lookup tables in the commercial finite element software Pam-Stamp(TM). Firstly, the FE model was validated using the deformed part which suffers from springback problem at the same forming condition. The FE results were compared with the measured experimental trails providing very good agreement. Secondly, the validated FE model was used to determine the suitable forming parameters which could minimise the springback of the deformed part.

1. Introduction

Sheet metal forming process is one of the most important manufacturing processes in the automotive sector. Achieving higher performance, higher productivity and lower cost is the main drive behind the development of the sheet metal forming technology in the automotive sector [1]. It is well known that the sheet forming processes, mainly cold forming, suffer from defects such as thinning, wrinkles, cracks and geometrical inaccuracies due to springback. Springback is the elastic recovery that causes change in the shape of the sheet metal part upon completion of the forming process and removal of the part from the tooling [2].

The springback phenomenon causes dimensional deviations and often resulting in rejection of the part due to failure in meeting dimensional tolerance requirements. Moreover, in many cases the springback may cause a delay in assembly time due to the additional post forming process to compensate the springback or additional machines with higher capacity to fit the parts correctly in the assembly [1].

A common countermeasure method against springback is to design the forming tool in a way that can compensate for the springback. However, calculating the required compensation accurately is a challenging task even for highly experienced tool designers. Therefore, in practice, engineers mainly rely on trial and error approach to compensate for the springback [2].

Numerical simulations with an adequate material model with its parameters, tooling and part geometries is an effective alternative approach to the time consuming and expensive trial and error method. Finite element (FE) simulations provide an accurate prediction of springback which resulted in an optimized tool design that produces a sound part shape after forming at first attempt.

The current work is an attempt to investigate the springback of a case study which a part that experiences delays in assembly. The part, shown in figure 1a, is provided by INDE GROUP FOR ENGINEERING INDUSTRIES [3]. The main aim of the work is to develop an FE model coupled with an adequate with suitable material model to predict and compensate for the springback problem for the investigated part by testing different process parameters.

2. Analysis of the formed part

The Fog lamp part considered in this work is produced from 1.5mm low Galvanized low carbon steel. The part is produced in two stages; the first stage is a blanking process to produce the desired blank shape and the second stage is the forming process. The part is cold formed using 300 bar hydraulic press as shown in figure 1(b). The current forming parameters are a punch pressure of 130 bar and a cushion pressure (back pressure) of 70 bar with a punch speed of 15 m/s.

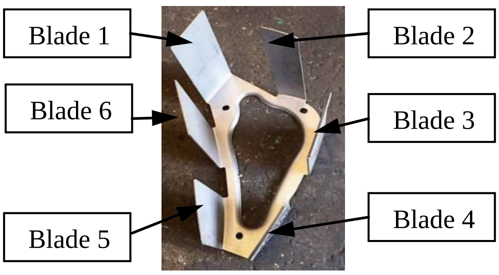

Figure 1. Fog lamp part investigated in the present work, INDE GROUP FOR ENGINEERING INDUSTRIES

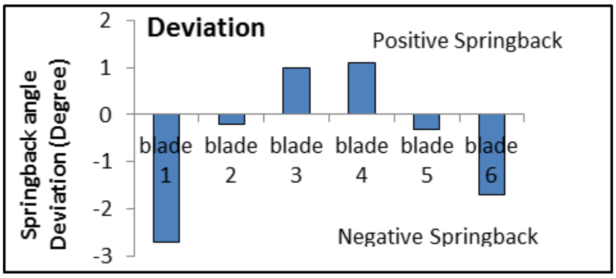

Figure 2. springback deviation from the CAD data

In order to characterize and analysis the springback behavior of the formed part, the physical part is scanned using Coordinate Measuring Machine (CMM). The required angle for all blades is 92° and by comparing the scanned and CAD parts, springback deviation was obtained as shown in figure 2, it is found that blade 1 has the largest deviation with 2.7° while the blades 6, 4 and 3 exhibits angle deviations of 1.7°, 1.1° and 1° respectively. Blades 2 and 5 show minimum deviation in the blades' angles.

The analysis of the formed part in figure 2 shows that the part is affected by both positive and negative springback. Blades 3 and 4 are affected by positive springback (i.e. the produced angles are more than the required angles). This type of springback occurs due elastic recovery during unloading. The existence of this type means poor tool design and/or improper springback compensation. On the other hand, blades 1 and 6 are affected by negative springback (i.e. the produced angles are less than the required angles). This type of springback usually occurs due to overloading and over stamping for the part during forming stage to compensate for the positive springback especially in closed bending dies [4].

The key factor in the forming process that directly affects the amount of springback in this part is the combination of the forming and cushion (back) pressures. Applying low pressures in forming and/or cushion pressures may cause insufficient forming and less pressure on the bent section leading to positive springback. On the contrary, applying high forming and/or cushion pressures means over stamping which leads to negative springback. In complex shapes, high pressures can cause thinning and cracking which must be taken into consideration during the design process. In the present work, there are three factors in addition that affect the springback and must be taken into account in the analysis process. First, blades have different orientations with respect to the parent sheet which is an important factor if the martial used has an anisotropic behavior. Second, the lengths of the blades are different, and as the blade length increases the springback sensitivity increases. Third, not all blades have straight rectangular cross section, some blade have curved cross section which affects the springback value [2].

3. Material Model

3.1. Material models and calibration
The accuracy of springback analysis is highly dependent on the predictions of stress levels at the final stage of stamping and during springback. The uneven stress distributions along the cross section of the component which occurs due to the stretch bending and subsequent unbending when drawn over a die corner are directly related to the material's hardening, Bauschinger effect, and cyclic hardening characteristics. As shown in figure 3, isotropic hardening plasticity models usually model an overrated cyclic hardening behavior under reverse loading. Moreover, it can't model the Bauschinger effect during cyclic deformation, the rapid change of material work hardening and early re-yielding. As a result, kinematic and combined hardening material models are introduced to overcome the limitations of isotropic models. These advanced material models have limited application in industrial models due to the increased number of material parameters that should be calibrated which require complex material testing and advanced mathematical techniques to determine these parameters. In addition, these complex material tests require high cost as compared to those used in the isotropic hardening models [4][5]. In order to predict the springback accurately, two different material models in Pam-Stamp(TM), isotropic hardening and kinematic hardening material model.

*3.1.1. Isotropic hardening model.*In the isotropic hardening model, the hardening part of the material behavior obeys Hollomon law, equation (1), while the plasticity law obeys Hill's plasticity [4].

$$\sigma = k\,\varepsilon^n \tag{1}$$

where $\boldsymbol{k}$ is the strength coefficient, $\boldsymbol{n}$ is the work hardening, $\acute{\varepsilon}_{\mathbf{p}}$is the effective plastic strain and $\boldsymbol{\sigma_y}$ is the yield stress [5]. The input parameters for the model are given in Table 1.

*3.1.2. Yoshida-Uomuri (Y-U) model.*Y-U a model is a two-surface model that assumes the kinematic hardening of the yield surface $\boldsymbol{f_0}$ with backstress $\boldsymbol{\alpha}$ within the bounding surface $\boldsymbol{F}$ with back stress $\boldsymbol{\beta}$ of mixed isotropic–kinematic hardening. The centre of yield surface ***f*** is moving with the back stress $\boldsymbol{\alpha}$. The boundary of $\boldsymbol{F}$ is expanding due to the plastic strain hardening which is expressed by the (B+R) where $\boldsymbol{B}$ is the bounding surface $\boldsymbol{F}$ initial size and $\boldsymbol{R}$ it's isotropic hardening component[6].

$$\mathrm{f_0} = \varphi(\sigma - \alpha) - \mathrm{Y} = 0 \tag{2}$$

$$\mathrm{F} = \varphi(\sigma - \beta) - (\mathrm{B} + \mathrm{R}) = 0 \tag{3}$$

Where $\boldsymbol{\varphi}$ is the anisotropic yield function, $\boldsymbol{\sigma}$ is the Cauchy stress and $\boldsymbol{Y}$ is the yield strength. The isotropic hardening, which represents the expanding rate of boundary surface of ***F*,** is a function of plastic strain rate $\boldsymbol{P}^{\circ}$ and is defined by

$$\mathrm{R}^{\circ} = \mathrm{m}\,(\,\mathrm{Rsat}\; - \mathrm{R}\,)\mathrm{P}^{\circ} \tag{4}$$

while the Bauschinger effect is expressed in terms of the back stress $\boldsymbol{\alpha}$ which consists of two components $\boldsymbol{\beta}$ and $\boldsymbol{\alpha*}$, defined by following equations:

$$\alpha^{\circ} = \beta^{\circ} + \alpha^{*\circ} \tag{5}$$

$$\alpha^{*\circ} = \mathrm{c}\left[\left(\frac{\mathrm{a}}{\mathrm{Y}}\right)(\sigma - \alpha) - \sqrt{\frac{\mathrm{a}}{\overline{\alpha *}}}\,\alpha^{*}\right]\mathrm{P}^{\circ} \tag{6}$$

$$\beta^{\circ} = \mathrm{m}(\tfrac{2}{3}\mathrm{b}\;\varepsilon^{\mathrm{p}} - \beta\,\mathrm{P}^{\circ}) \tag{7}$$

$$\mathrm{a} = \mathrm{B} + \mathrm{R} - \mathrm{Y} \tag{8}$$

Where $\boldsymbol{Y}$, $\boldsymbol{C}$, $\boldsymbol{B}$, $\boldsymbol{m}$, $\boldsymbol{b}$ and $\boldsymbol{Rsat}$ are material parameters of Y-U model. Besides these 6 parameters, another parameter $\boldsymbol{h}$ is used to express hardening stagnation and considered as an adjusting parameter. The Young's modulus used in Y-U model varies with plastic strain which can be described by

$$E(\bar{\varepsilon}^p) = E_0 - (E_0 - E_a)\left[1 - \exp(-\xi\,\bar{\varepsilon}^p)\right] \quad (9)$$

Where E_0 and E_a are the Young's module for initial and infinitely large pre-strained materials, respectively. $\bar{\varepsilon}^p$ denotes effective plastic strain and ξ is a material constant [4,6].The two models (Isotropic hardening and Y-U model) are calibrated by fitting their mathematical (Numerical) expression with cyclic stress strain curve as shown in figure 3. The experimental cyclic stress strain curve was obtained from Yoshida and Uemori [6]. Tables 1 and 2 shows the material's constants for the calibrated material models; isotropic hardening and Y-U model, respectively.

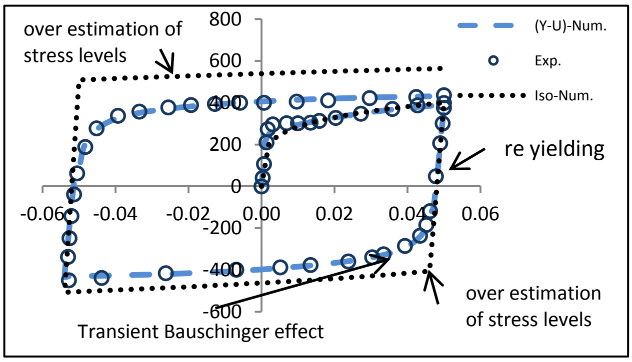

Figure 3. Calibration of the models by fitting the data with stress strain curve of the material.

Table 1. Material constants of the isotropic model for mild steel

Strength coefficient (k)	750MPa
Work hardening (n)	0.21
Young's modules(E)	207 GPa
Possion ratio (v)	0.3
Density	7.85 g/cm^3

Table 2. Material constants of Y-U model for mild steel

Y	**C**	**B**	**m**	**b**
250	260	276	18	100
Rsat	**h**	**E0**	**Ea**	**ξ**
98	0.5	207000	160000	11

4. FE Model

4.1. Implementation of the material models

In the present work, Pam Stamp(TM) commercial finite element software is used to model and predict the stamping and springback stages of the investigated part. Both material models (Isotropic and Y-U model) are implemented in Pam Stamp(TM) via lookup tables. In order to validate and test the implementation process of the material models, one element cyclic test simulation is performed and the obtained cyclic stress strain curves are compared with their numerical curves as shown in figure 4. It should be noted here that the simulated stress strain curves of both isotropic and Y-U models are identical to their numerical curves.

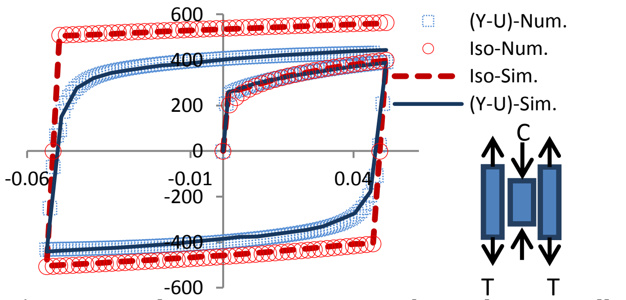

Figure 4. Cyclic stress strain curves obtained numerically and from one element cyclic test simulation T=Tension,

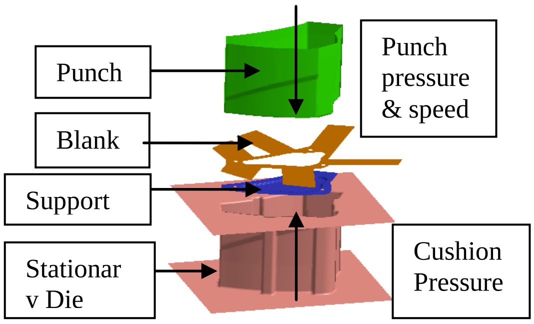

Figure 5. Tool setup with Boundary conditions the FE model.

4.2. Model setup

The blank of the part was modeled using shell elements with an element size of 3mm, thickness 1.5 mm using fully integrated analysis. The tool identification and the boundary conditions are applied as shown in figure 5. In the initial trails in the FE model, the blank is formed by a rigid punch with forming pressure of 130 bar and punch speed 15 mm/s. The support carries the blank with cushion pressure of 70 bar. The friction coefficient between all contacting surfaces is taken as 0.17.

5. Results and Discussions

5.1. Simulation of the Part with Punch Pressure of 130 Bar and Cushion Pressure of 70 Bar
Table 3 shows the predicted angles by the two selected models for the selected part. The deviations in the measurements of all angles between the formed and the simulated part are shown in figure 6. It can be seen that, the minimum deviation is obtained from Y-U model and hence is considered to be more reliable in predicting true material behavior.

Figure 7 shows a comparison between the FE model using the Y-U Model (F) and the produced part scanned by CMM (S) under the same forming conditions to validate the developed FE model for further investigations. As shown in figure 7, the FE model shows good agreement with the formed part. The maximum deviation measured is 0.7^o indicating that the FE model could be adopted for further analyses.

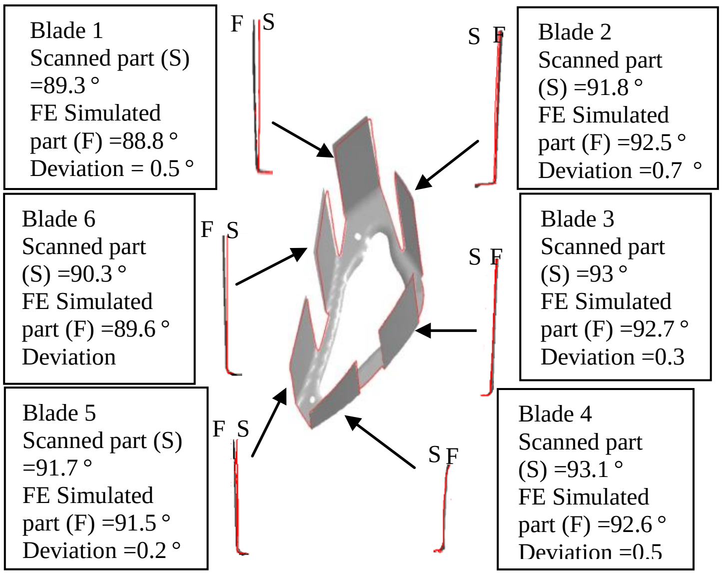

Figure 7. Comparison between the FE simulated (3D part- symbol (F)) and physical scanned part (wire frame – symbol (S)) formed at punch pressure of 130 bar and cushion pressure of 70 bar.

TABLE 3. Blades' angles Predicted by the two models.

	ANGELS					
	1	**2**	**3**	**4**	**5**	**6**
Y-U	**88.8**	**92.5**	**92.7**	**92.6**	**91.5**	**89.6**
Isotropic	**93**	**94.2**	**93.6**	**93.8**	**88.9**	**92.7**
Scanned Part	**89.3**	**91.8**	**93**	**93.1**	**91.7**	**90.3**

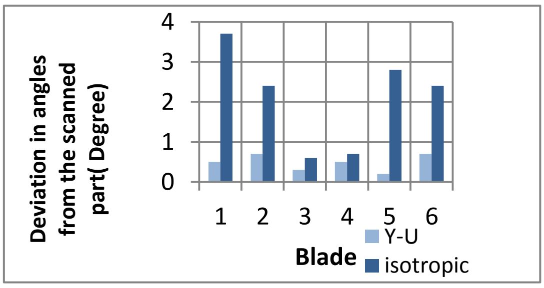

Figure 6. Deviation of the predicted angles by the material models from the formed part in degree.

FE model coupled with precise material model (Y-U model) is strongly able to predict the springback accurately. In the next step of the work, the FE model is used to determine suitable pressures combination which minimizes the springback. Several simulation trails with different forming conditions are carried out.

It is difficult to identify precisely the optimum pressure combination, but it is found that a punch pressure of 50 bar and a cushion pressure of 20 bar is the most suitable combination, which enabled the part to be formed with a minimum springback effect. To explore this further, the comparison between the FE model and scanned part at defined suitable condition is given below.

5.2. Prediction of the Part with a Forming Pressure of 50 Bar and a Cushion Pressure of 20 Bar
Figure 8 shows a comparison between the FE model and produced part for the proposed process parameters and the old ones. The FE model predicts the angles of blades 1, 3, 4, 5 and 6 successfully with very a maximum deviation 0.3^o and a deviation of 0.8^o for blade 2. In addition, figure 8 shows a comparison between the scanned physical part and the CAD data of the desired part (referred as (C)). The FE model results for blades 3 and 4 show angles identical to the design angle of 92^o, blades 2 and 5 have deviations of 0.5^o and 0.1^o, respectively. However, deviations, between the FE model and the CAD data, in angle values can be seen for the blade 1 and 6. These deviations are summarized in figure 9.

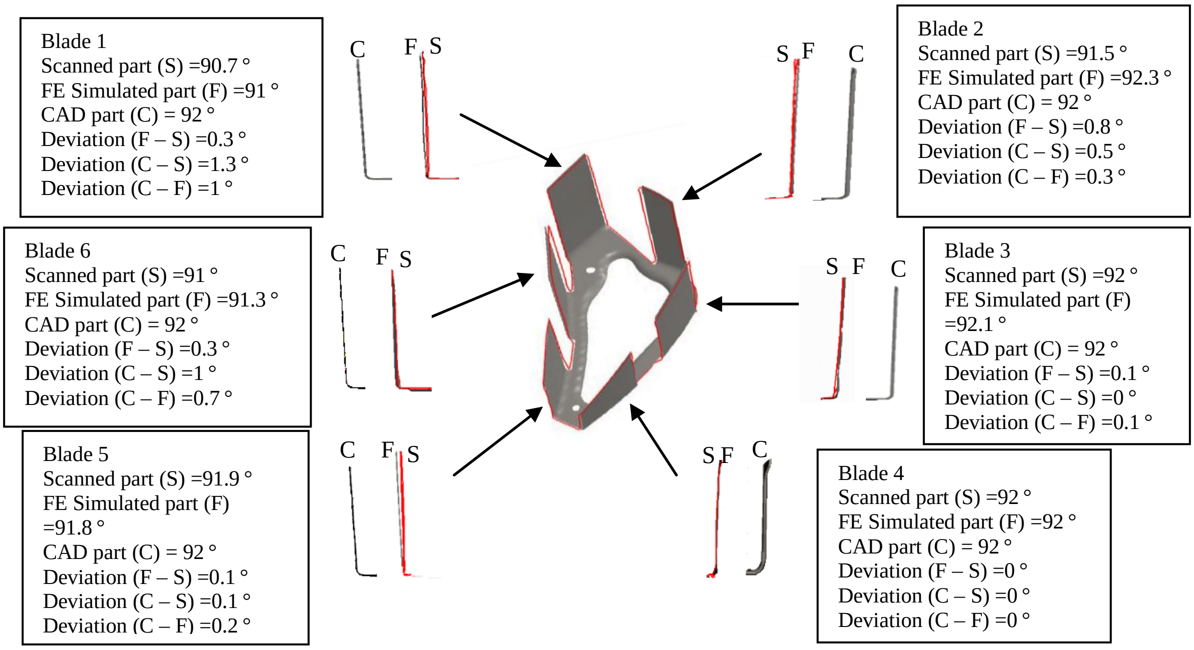

Figure 8. Comparison between the CAD Data (symbol(C)), FE simulated part (3D part-symbol (F)) and physical part (wire frame – symbol (S)) formed at punch pressure 50 bar and cushion pressure 20 bar.

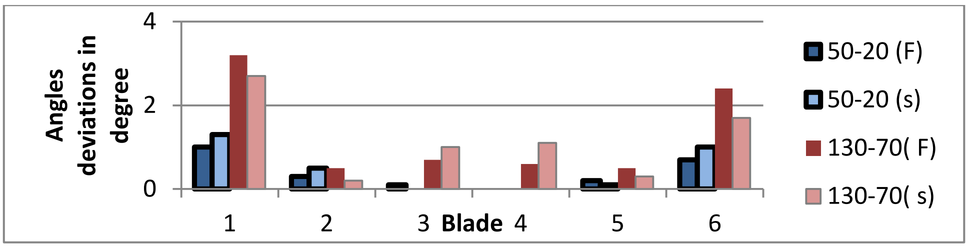

Figure 9. Deviation of the scanned (s) and FE simulated parts (F) at the two conditions (130-70) and (50-20) from the CAD part.

FE simulations coupled with advanced material models have a significant impact in developing the forming process and predicting the springback. As shown in figure 9, calibrating the FE model with the scanned formed part, fabricated with forming and cushion pressures of 130 and 70 bar (S), provide a robust model that was able to predict the springback under different forming conditions and by several simulation trails the suitable forming conditions of forming and cushion pressures of 50 and 20 bar (F) has been found to be the best pressure combinations and resulted in good agreements between the FE simulation and its experimental trail.

However, there is a little deviation between the FE simulations and the scanned parts from the actual forming process. These deviations can results from different sources of errors including inaccurate characterization of the anisotropy value of the material data that were obtained from the work of Yoshida [6] and/or errors in CMM measurements. However, despite of these errors and the resulted deviations, the results provide improvements in production line of the proposed part where the delay in assemble process caused by springback has been reduced.

6. Conclusions

In this paper, the forming process of an automotive part which affected by positive and negative springback was modeled using Pam Stamp$^{(TM)}$ and validated by comparisons against experimental trails. The accuracy of springback analysis depends on the predictions of stress levels at the final stage of stamping, and also at the springback stage. Two different material models (Isotropic and Y-U model) were used to predict the springback. It was found that Y-U model is considered to be more accurate material model in simulating bending processes and predicting springback providing maximum deviation measured (0.7^{o}) which is indicating that the FE model could be adopted for further analyses. By using the validated FE model coupled with advanced material models, the suitable pressure combination is determined which enabled the part to be formed with a minimum springback effect. The right compensation for the springback obtained by the FE simulation improves the productivity and minimize the delay caused by springback while assembling process.

Acknowledgments
The authors would like to thank INDE GROUP FOR ENGINEERING INDUSTRIES for providing opportunity to use the tool and the equipment.

References

[1] Ishikawa T and Mori K 2014 Springback analysis of high strength dual-phase steels 11th *Int. Conf. on Technology of Plasticity* Vol 81 (Japan : Congress Center) pp 975-980
[2] Yoshida T 2002 Material modeling for accuracy improvement of the springback prediction of high-strength steel sheets Nippon Steel Technical Report Vol 102
[3] INDE GROUP FOR ENGINEERING INDUSTRIES, 6OCT City, Cairo, Egypt. http://www.inde.com.eg/
[4] Pam Stamp user's guide manual 2017
[5] Ma N, UmezuY, Watanabe Y and OgawaT 2008 Springback prediction by yoshida-uemori model and compensation of tool surface using jstamp 7th *Int. Conf.and Workshop on Numerical Simulation of 3D Sheet Metal Forming Processes*
[6] Yoshida F and Uemori T 2002 A model of large-strain cyclic plasticity describing the bauschinger effect and work hardening stagnation *Int. Journal Of Plasticity* Vol 18 pp 661-686