THE ANNALS "DUNĂREA DE JOS" UNIVERSITY OF GALAȚI
FASCICLE V, TECHNOLOGIES IN MACHINE BUILDING, ISSN 1221-4566
2005

Numerical Simulation of Forming Limit Curves using Reduced Scale Samples

Viorel Păunoiu[1], Dumitru Nicoară[1]
Ana Maria Cantera Lopez[2], Pedro Arroyo Higuera[2],
[1] Dunărea de Jos University of Galați, Romania
[2] University of Valladolid, Spain

Abstract

In this paper, a numerical simulation of an alternative solution to Haşek conventional method for forming limit curves determination is proposed. New types of samples, for the experimental research, were used, which dimensions are reduced three times in comparison with the conventional one. This numerical study was made with DYNAFORM software. The Belytschko-Lin-Tsay shell element based on a combined co-rotational and velocity-strain formulation was chosen to analyze the elasto-plastic process with complex geometrical nonlinearity. The geometrical modeling of the die-binder-blank-punch assembly request the calculations of the characteristic profiles of the working surfaces. A series of conclusions obtained from the numerical simulation of the forming limit curves using the new types of samples and a comparison with experimental one, are shown at the end.

Keywords: numerical simulation, FLD, press forming, FEM, formability

1. Introduction

Finite Element Analysis (FEA) is a powerful simulation tool for analyzing complex three dimensional sheet metal forming problems related to potential forming defects such as tearing, wrinkling and spring back. It can be used during the die design stage or as a troubleshooting tool in the production mode.

Sheet metal formability should be thought of as a systematic process which includes the consideration of material properties, die design, forming process control by appropriate setting and adjustment of the restraining force generated by draw beads and binders, and lubrication related issues.

The forming limit curves, FLDs, is one of the method in examining the failure potential, which include a good representation of material's stretchability and the easiness when used for trouble shooting.

The DYNAFORM-PC solution package was developed to study the issues stated above and to assist the die designer and stamping engineer in meeting rapid prototyping requirements.

In the paper are presented the numerical simulation of an alternative solution to conventional methods for forming limit curves, using a special option of DYNAFORM software.

2. FLD in Dynaform

In DYNAFORM, to properly display Forming Limit Diagram (FLD) results, the shape and location of the Forming Limit Curve (FLC) must be determined. This FLC can be determined two different ways: by defining mechanical properties *n*, *r-bar*, and t_{mat} of the material, or by explicitly defining the *(x, y)* points that represent the stress-strain (σ-ε) curve for the material in question.

The mechanical property denoted by *n* is the strain hardening (or work hardening) exponent, defined as material parameter P2 within the pre-processor. The mechanical property denoted by *r-bar* is the average plastic strain ratio. In the pre-processor, the plastic strain ratio is represented in three directions; r_{00} along the rolling direction, r_{45} at 45 degrees to the rolling direction, and r_{90} at 90 degrees to the rolling direction. The average plastic strain ratio is:

$$r - bar = \frac{r_{00} + 2r_{45} + r_{90}}{4} \qquad (1)$$

True strain (ε_{true}) is differentiated from engineering strain (ε_{eng}) by dividing the change in size of the tested sample by its instantaneous cross-sectional area as opposed to the original cross-sectional area.

The size and location of the FLD display window in the post-processor can be set by defining pixel sizes and locations (figure 1.)

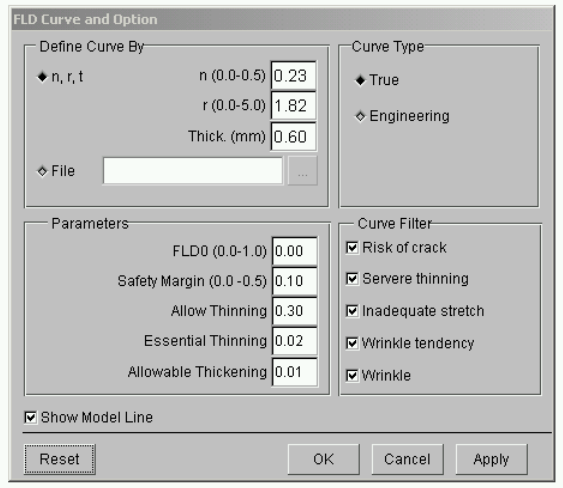

Fig. 1. FLD window in POSTPROCESSOR, [1]

The *Option* section allows the user to turn the mode lines on or off. These mode lines include areas undergoing uniaxial compression, pure shear, uniaxial tension, and balanced biaxial stretch forming.

The location of the intersection between the FLC and the ε_{maj}-axis can be specifically determined by setting the FLD_0, between 0.0 and 1.0 major strain.

The setting for *Safety Margin* determines what range of strain below the FLC is defined as representing areas of the sheet that are at risk of cracking. Generally, this is set to 10 % strain.

The setting for *Allow Thinning* determines what percent thinning of material will represent cracks and failure. Generally, it is advised to set this to about 20 %, though some customers may have other maximum thinning requirements.

The *Essential Thinning* setting determines what minimum percent thinning of material is required to determine any change in material strain composition.

The *Allowable Thickening* setting determines what percent gathering of material is necessary before the material is defined as wrinkling. This should normally be set to about 10 %, through some material will gather to as much as 50 or 60 percent, under the right forming conditions.

The remaining settings allow the user to activate or deactivate the different contour color bands. The user can display a contour map as simple as only *Safe* and *Crack,* or can also include any combination *of Risk of crack, Severe thinning, Inadequate stretch, Wrinkle tendency,* and *Wrinkle.*

3. Simulation models

The model was created by generating the real geometry of the active element profiles: die, punch and binder and also the profile and dimension of the blanks.

In the assembly of the simulation there are three active parts and the blank. As it can see in the figure 2. a circular blank is placed over a die opening with a comer radius and a diameter. The blank is held in place with a binder which applies the binder force for preventing the formation of folds. A punch, with a diameter slightly smaller than the die and binder diameters and a corner radius moves downward.

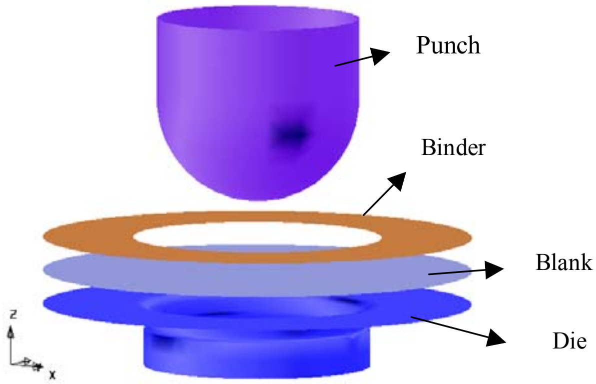

Fig. 2. – Simulation Press Device Assembly

It was considered a die which active parts are of an interior diameter equals to 35 mm and a flange diameter of 70 mm. The die has 1228 finite elements. The hemispherical shape punch is of a diameter equal to 35 mm. The punch has 458 finite elements. The binder has a diameter equal to 35 mm and a flange diameter of 70 mm. The binder has 370 finite elements.

Finally, there were considered 7 different blank geometries with a contour diameter equals to 70 mm and a radius 50 mm symmetric circular trimming centred to obtain shapes as presented in figure 3 and table 1.

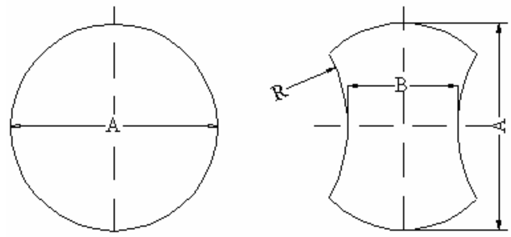

Fig. 3. - The Blanks

Table 1. The samples dimension

Number of specimen	1	2	3	4	5	6	7
B(mm)	0	63	40	33	20	13	7

All the blank shapes are presented in figure 4.

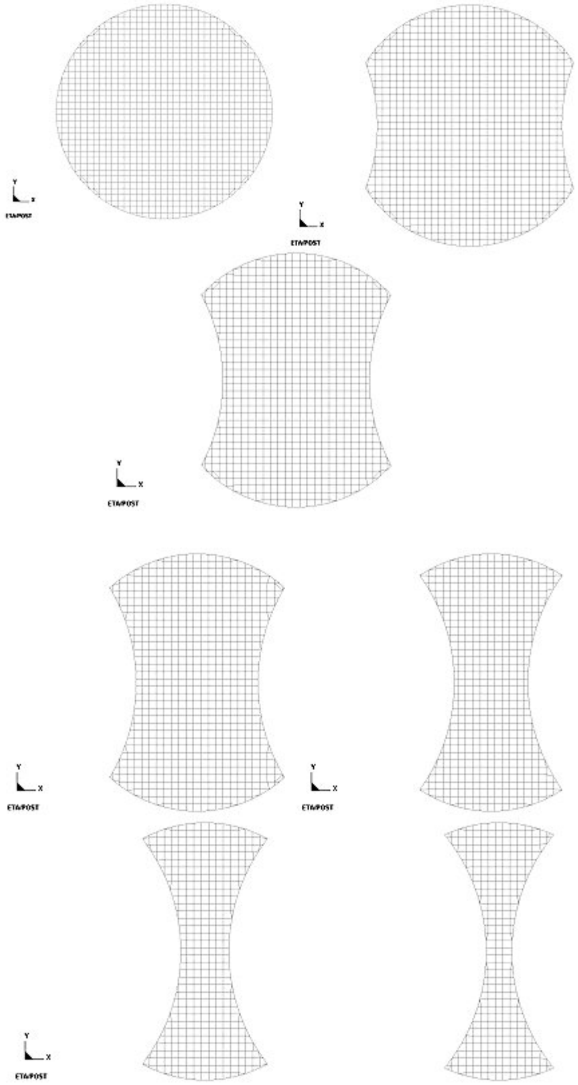

Fig. 4. The blanks geometry

The elements used in meshing are shell element type ones. The blanks were created using an elasto-plastic material. It was checked the coincident nodes, warpage and overlapped elements. After this, the final blank’s models design is finished. The number of finite elements in each blank is shown in the table 2.

Table 2. – Blank finite elements

Number of blank	1	2	3	4	5	6	7
Number of finite elements	1013	905	771	763	497	385	293

The material chosen for the analysis was Material-Type 36 in DYNAFORM: CQ mild STEEL which properties are consistent with the real one.

This is an anisotropic material which uses the Hollomon power law hardening rule as follows:

$$\sigma = K\varepsilon^n \tag{2}$$

where K is the strength coefficient and n is the work hardening exponent.

The values of k and n are as follow: $K = 479.3$, $n = 0.226$

The anisotropic properties of the material are: $r_{00} = 1.45$; $r_{45} = 1.1$; $r_{30} = 1.73$.

The selected simulation value for binder-piece friction parameter was, as suggested by the program: 0,125: however, punch-piece friction value was chosen to be really low: 0,001, with the aim to match the experimental conditions, instead of using the program suggested value.

For the punch it was selected a velocity of 5000 mm/s in a stroke distance of 15 mm. The force applied by the binder was chosen to be 200000 N. The material simulated thickness matches the real thickness of 0.7 mm.

3. Numerical results

In the following pictures are presented the FLD curves only for specimens 1, 2 and 7 simulated in DYNAFORM post-processor. Each curve has associated the figure of the sample for the step considered. There are considered two steps for each specimen, the first one shows behaviour just before cracking appears and the second one shows behaviour after the cracking occurred.

Figures 5 and 6 present the FLD curves for the first sample. As it could be seen the deformation mode (figure 5) corresponds to the equi-biaxial stretching at the pole of the stretched dome. The degree of thinning is 41.84 %.

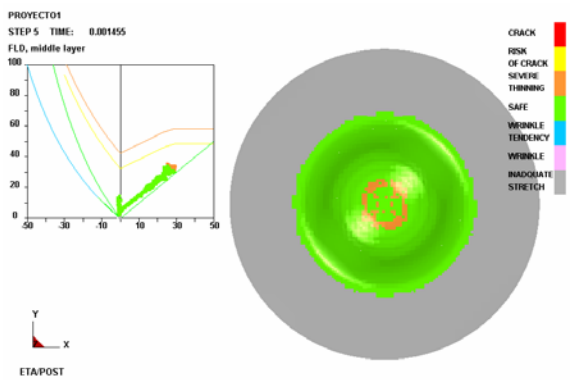

Fig. 5. – Specimen 1 before cracking

At the fracture the degree of thinning is 63.59 %. The fracture appears on the lateral wall of the punch (figure 6).

Figures 7 and 8 presents the FLD curves for the sixth sample. The deformation mode (figure 7) tends to the uniaxial extension. The degree of thinning is 21.41 %.

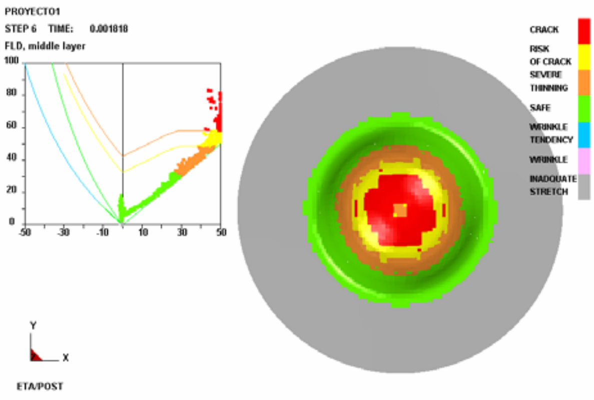

Fig. 6. – Specimen 1 after cracking

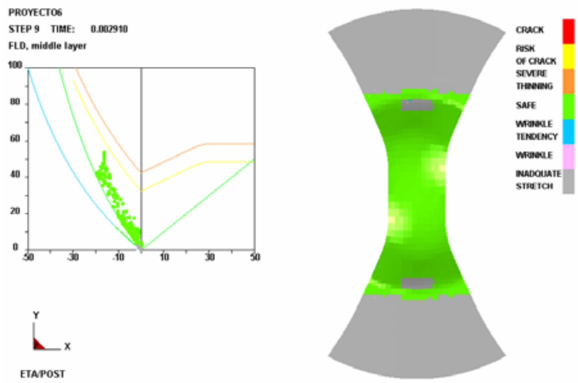

Fig. 7. – Specimen 6 before cracking

At the fracture the degree of thinning is 28.92 %. The fracture appears in a more limited area on the lateral wall of the punch (figure 8).

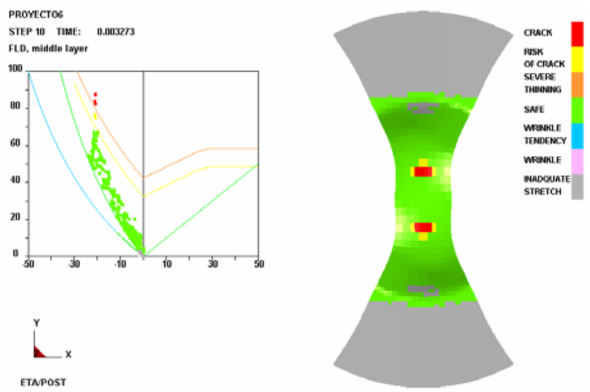

Fig. 8. – Specimen 6 after cracking

Figures 9 and 10 presents the FLD curves for the seventh sample. The deformation mode (figure 9) corresponds to the uniaxial extension. The degree of thinning is 25.36 %.

At the fracture the degree of thinning is 57.94 %. The fracture appears in the middle of the sample, The phenomenon of necking is clearing present (figure 10).

DYNAFORM Post-Processor gives the possibility to measure the deepness the specimen reaches. Furthermore it is possible to compare the deepness reached in each specimen after and before cracking so that a simulated value for the admissible deepness in the deep drawing process is established. The values are collected in table 3.

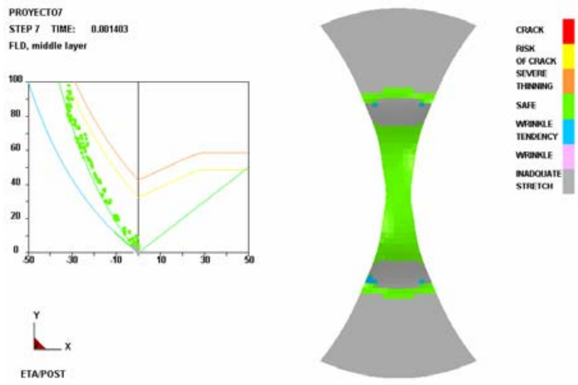

Fig. 9. – Specimen 7 before cracking

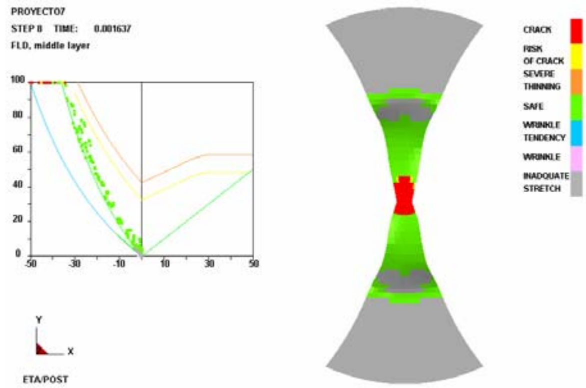

Fig. 10. – Specimen 7 after cracking

Table 3. – Deepness Reached (mm)

Specimen Number	Deepness before Cracking	Deepness after Cracking
1	11.696	15.277
2	12.423	13.550
3	8.797	10.610
4	8.814	10.581
5	10.625	12.463
6	12.450	13.588
7	11.198	13.370

Other major advantage given by DYNAFORM Post-Processor is to take measures on each node. Using this function, a measurement of the strain in a selected line is been made with the aim to compare them to the experimental readings. For the specimens 1, 6 and 7, there are presented the strains values (tables 4-6) both from simulations (1) and from experimental work (2).

Table 4. – Strains for Specimen 1

Measured Point	ε_1		ε_2	
	1	2	1	2
1	0.516	0.468	0.421	0.362
2	0.466	0.383	0.392	0.282
3	0.380	0.330	0.334	0.230
4	0.366	0.306	0.306	0.147
5	0.217	0.259	0.192	0.091

Table 5. – Strains for Specimen 6

Measured Point	ε_1		ε_2	
	1	2	1	2
1	0.560	0.461	-0.218	-0.237
2	0.412	0.383	-0.172	-0.166
3	0.335	0.330	-0.145	-0.189
4	0.259	0.306	-0.103	-0.175
5	0.109	0.259	-0.044	-0.148

Table 6. – Strains for Specimen 7

Measured Point	ε_1		ε_2	
	1	2	1	2
1	0.421	0.493	-0.050	-0.051
2	0.290	0.340	-0.018	-0.019
3	0.289	0.274	-0.017	-0.011
4	0.184	0.208	-0.005	-0.005
5	0.134	0.034	-0.003	-0.004

4. Conclusions

Using DYNAFORM program, the deformation of different types of samples was simulated, for studying the different modes of strain states.

For each sample, with the help of the program, it was obtained the Forming Limit Curve. The forms of these curves are changing in connection with the form of the samples, the strain state variation covers biaxial stretching, plane strain and uniaxial tension.

It was verified that the reduced scale samples could be use for identified the different strains states in material.

The obtained major and minor strains values for both simulation and experimental work are presented in tables 4 to 6. It results a good correlation between the experimental and numerical results and the FLC-s assure the estimations of the material behaviour during deformations.

In general is easy to realize that experimental values are higher than the simulated ones. It is possible to be caused by an accumulated error in the measuring procedure. Anyway, the deviation keeps a tendency so it would be easy to correct the errors as soon as the readings were taken by different researchers and maybe in different microscopes so that an average of the readings would have been made. Other way to interpret these results is to accept that the computer simulation offers a more accurate and adjusted measurement than the experimental results.

References

[1] *** DYNAFORM, *Application manual*

[2] *** LS DYNA, *Theoretical manual*

[3] Păunoiu, V, Nicoară, D., *Sheet metal forming technologies*, University Book Press București (2004)

[4] Knapke. J., *Evaluation of a variable-configuration-die sheet metal forming machine*, Ph.D. thesis, MIT, 1988

[5] Păunoiu, V., Nicoară, D., *Simulation of friction phenomenon in deep drawing process*, Analele Universității Dunărea de Jos, fasc. VIII, Tribology, Proc. Conf. ROTRIB'03, vol. I, ISSN 1221-4590, 2003

[6] Lopez Cantera A. M., Higuera Arroyo P., Păunoiu, V., Nicoară D., *Experimental researches regarding the forming limit curves using reduced scale samples*, Analele Universității Dunărea de Jos, fasc. V, Technology in Machine Building, ISSN 1221-4566, 2005

[7] Lopez Cantera A. M., Higuera Arroyo P., - *Numerical and experimental simulation of the forming limit curves*, Final Project, Erasmus Scheme, Universitatea Dunărea de Jos din Galați, 2005

[8] Banabic, D., Dorr, I.R., *Deformabilitatea tablelor metalice. Metoda curbelor limtă*, Editura OIDICM, Bucuresti, ISBN 973-95641-1-9, 1992

Simularea numerică a curbelor limită de ambutisare utilizând epruvete de dimensiuni reduse

Rezumat

În această lucrare se propune o simulare numerică a unei soluții alternative la fmetoda convențională Haşek de determinare a curbelor limită de ambutisare. Pentru cercetarea experimentală s-au folosit noi tipuri de epruvete ale căror dimensiuni sunt micşorate de trei ori în comparație cu cele convenționale. Studiul numeric s-a făcut cu software-ul DYNAFORM. Pentru a analiza procesul elasto-plastic cu neliniarități geometrice complexe s-a ales elementul tip placă şi membrană Belytschko-Lin-Tsay. Modelarea geometrică a ansamblului matriță-semifabricat-placă de reținere-poanson a presupus determinarea profilelor caractersitice ale suprafețelor de lucru. În final sunt prezentate o serie de concluzii obținute din simulările numerice şi din comparația acestora cu rezultatele încercărilor experimentale.

Numerische Simulation der Formsbegrenzung Kurven mit Eineverringerten Skala-Proben

Zusammenfassung

In diesem Papier wird eine numerische Simulation einer Ausweichlösung Haşek zur herkömmlichen Methode für die Formung von von Begrenzung Kurven Ermittlung vorgeschlagen. Neue Arten der Proben, für die experimentelle Forschung, wurden benutzt, die Maße dreimal im Vergleich mit dem herkömmlichen verringert werden. Diese numerische Studie wurde mit DYNAFORM Software gebildet. Das Belytschko-Lin-Tsay Oberteilelement, das auf einer kombinierten Corotations- und Geschwindigkeit-Belastung Formulierung basierte, wurde beschlossen, um den elastoplastischen Prozeß mit komplizierter geometrischer Nichtlinearität zu analysieren. Das geometrische Modellieren des Matrize-Blech-Niederhalter-Stempel Antrags die Berechnungen der charakteristischen Profile der Funktion Oberflächen. Eine Reihe Zusammenfassungen, die von der numerischen Simulation der bildenbegrenzung Kurven mit den neuen Arten der Proben und einem Vergleich mit experimentellem erreicht werden, werden am Ende gezeigt