TECHNICAL UNIVERSITY OF CLUJ-NAPOCA

ACTA TECHNICA NAPOCENSIS

Series: Applied Mathematics, Mechanics, and Engineering
Vol. 67, Issue Special I, February, 2024

FINITE ELEMENT ANALYSIS OF INCREMENTAL FORMING PROCESS AND EXPERIMENTAL VALIDATION

Mihai-Octavian POPP, Gabriela-Petruța RUSU, Mihaela OLEKSIK, Valentin OLEKSIK

Abstract: *The incremental forming process still requires further research before it can be implemented on a large scale in the industry, as certain unresolved drawbacks persist. Understanding material deformation necessitates knowledge of the fracture limit diagram, enabling the implementation of an analytical model that predicts the moment of material fracture. This paper presents a theoretical model of the single-point incremental forming process, subsequently validated through specific experimental tests. Line and cross tests are performed to assess and compare the outcomes with the model's predictions. The validation process aims to establish a correlation between experimental findings and analytical predictions, facilitating the efficient application of the incremental forming process in industry.*

Key words: *Incremental forming, finite element analysis, fracture limit diagram.*

1. INTRODUCTION

In the case of incremental forming process, deformability of the materials used is higher than in the case of other cold plastic deformation processes. The forming limit of the sheet blanks can be defined as the maximum level of deformation reached before the material failure.

Many researchers performed different tests to determine the forming limit diagram (FLD) for conventional forming process, such as deep drawing, but for incremental forming process there are not so many explicit tests on how to determine it. Still, some researchers defined also a curve for the incremental forming limit diagram (IFLD) [1]. These curves are defined as a relationship between major and minor strains of the parts produces. In conventional forming process the FLD is described as a V shape curve, whereas in IFLD as a straight line, as can be seen in the Figure 1. There are also some contradictory findings about these curves, some authors tried to implement the IFLD in order to predict the material failure but showed that it is not possible due to complex stresses that appear on the material [2]. Jeswiet observed that the IFLD depends on the maximum wall angle reached before material failure [3]. This wall angle depends on the material used, sheet blank thickness and technological parameters used [4].

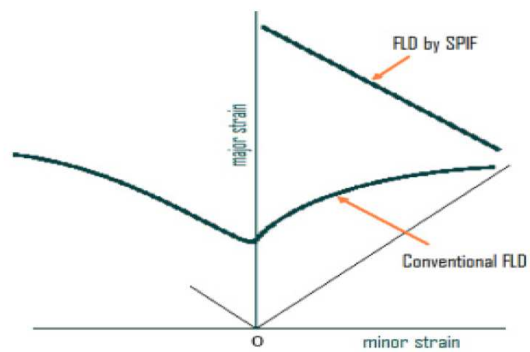

Fig. 1 Conventional FLD and IFLD

On the other hand, Filice took another approach to predict material failure through online monitoring of the forming forces and succeeded in the case of frustrum cone parts [5]. Szekeres showed that this principle cannot be applied in the care of frustrum pyramid parts [6].

The aim of this paper is to present a simple method to determine the IFLD by means of general tests in order to predict the maximum deformability of the material regardless of the type of parts produced. Furthermore, a finite element analysis model is presented in which the IFLD is implemented after experimental tests are performed, in order to compare the maximum depth reached before material failure from the theoretical model and experimental results.

2. MATERIAL AND METHOD

2.1 Experimental setup

In this paper, experimental tests were performed on 0.8 mm thickness DC01 steel sheet blanks. The tests involved a simple line test and a cross test in order to be able to determine a general IFLD for single point incremental forming process (SPIF). The sheet blanks were 250 by 250 mm and the punch used had a semispherical head with a diameter of 10 mm. The line and cross tests had a maximum width of 85 mm. The major and minor strains were measured with the help of a 3D strain analysis system ARAMIS. KUKA KR210 serial robot was used to deform the sheet blanks, which has an active payload of 2kN.

Before incremental forming process, uniaxial tensile test was performed on DC01 test parts until failure in order to determine the mechanical characteristics of the material. After the engineering stress-strain curve was obtained, it was transformed in true stress-strain curve which was later introduced in ABAQUS software for the theoretical model.

2.2 Theoretical model

Consistent with the aim of this paper, there was also a theoretical model developed in ABAQUS/EXPLICIT in order to study the material failure. The finite element analysis (FEA) model consists of two retaining rings, which were implemented as discrete rigid, a punch which is considered analytical rigid and the sheet blank which was modeled as deformable part. The deformable part was meshed with S4R elements, whereas the discrete rigid parts were meshed with R3D4 elements. The elements for the sheet blanks had a dimension of 1.5 x 1.5 mm. Besides the true stress-strain curve, it was also implemented the IFLD obtained from experimental tests. The plastic behavior was described with the help of exponential strain hardening law. The numerical simulations were performed without anisotropic model.

3. RESULTS AND DISCUSSION

3.1 Experimental results

The uniaxial test was performed on six DC01 specimens and the engineering stress-strain curves were obtained. In Table 1 are presented the mechanical properties of DC01 used for the simulations.

Table 1

Experimental tests and depth of failure

Tensile stress at Yeld [MPa]	Strenght coefficient K [MPa]	Strain hardening exponent n	Ultimate tensile stress [MPa]
196.21	521.52	0.23	286.35

After uniaxial tests, two incremental forming tests were performed as described in Table 2.

Table 2

Experimental tests and depth of failure

Case number	Type of toolpath	Punch diameter [mm]	Depth of failure [mm]
L1	line	10	16,43
C1	cross	10	17,89

Figure 2 and 3 shows the depth measured with the help of ARAMIS software after the moment of failure. During the tests it was observed that the material failure occurred at the moment when the punch enters deeper into the material, when the vertical step is in progress. Due to the fact that the vertical step is 1 mm, which is quite a big value for incremental forming process, it can be observed that the damage produced by the punch is more noticeable in the case of L1 than in the case of C1.

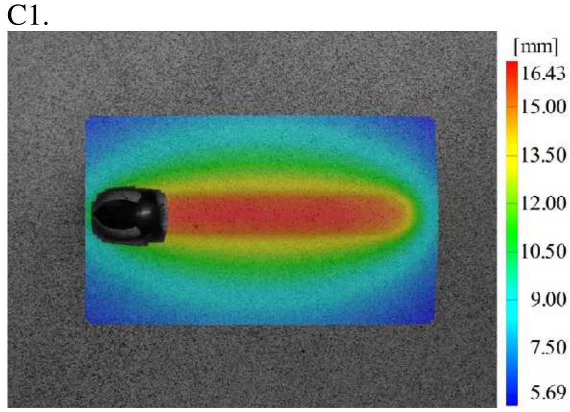

Fig. 2. Depth of failure for case L1

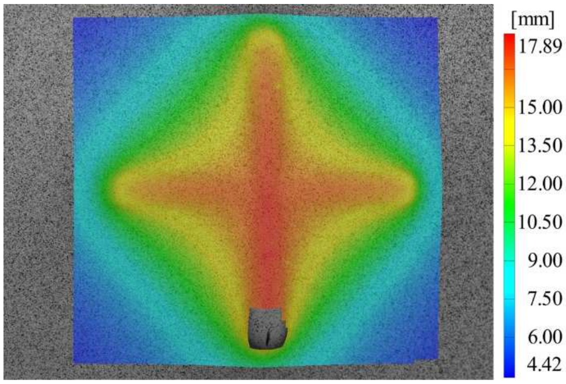

Fig. 3. Depth of failure for case C1

In case of L1 test the material failure happened at the beginning of the vertical step, whereas in the case of C1 it happened at the end of the vertical step. Furthermore, the maximum depth of 17,89 mm was reached in the case of C1, regardless it was used the same punch and vertical step. After the material failure, the tests were stopped and from ARAMIS were extracted the major and minor strains. With these points it can be predicted the IFLD of DC01 steel during SPIF process. Figure 4 shows the IFLD for the line test of DC01 steel, whereas the figure 5 shows the IFLD for cross test.

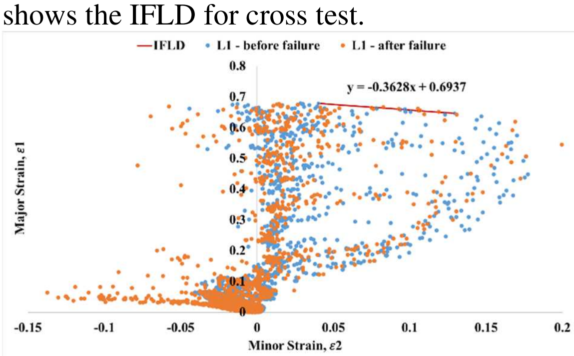

Fig. 4. IFLD for line test

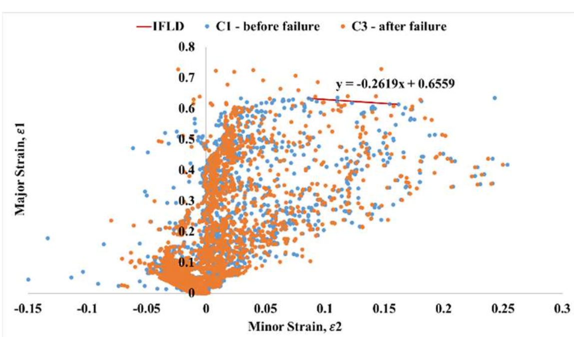

Fig. 5. IFLD for cross test

The two IFLDs showed in Figure 5 and 6 were implemented in ABAQUS software in order to simulate the material failure. Besides the IFLD in order to visualize the material failure in FEA software it is necessary to specify the damage evolution criteria. This parameter was extracted from the stress-strain curve.

From the comparison of the two IFLDs it can be observed that the IFLD obtained from the line test presents higher strain values than the IFLD from cross test. This is not consistent with the failure depth reached in experimental test, due to the fact that in the case of line test the 3d strain analysis system did not capture the exact moment of material failure as can be seen from Figure 3. Thus, there were obtained higher major strains in the case of line test, which is not completely true. Still, one must take in consideration that for both tests the IFLDs obtained are very close in terms of major and minor strain values.

3.2 Simulation results

After the IFLDs implementation in ABAQUS, the numerical simulations were performed and the depth of material failure was extracted from both tests. In Figure 6 can be seen the depth of material failure in case of line simulation and Figure 7 shows the depth in case of cross simulation.

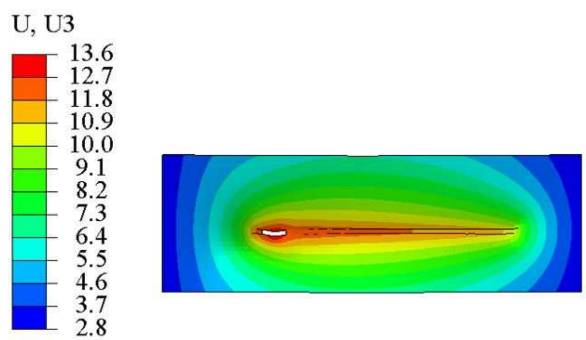

Fig. 6. Depth of failure for line simulation

From the simulation of the incremental forming process of line and cross test it can be observed that for the line simulation the maximum depth reached is 13.6 mm with a difference of 2.83 mm from the experimental test. In case of cross simulation, the maximum depth reached is 13.7 mm with a difference of 4.19 mm from the experimental test.

Another statement is that for both simulations the depth reached in the moment of failure is very close, one can say that is almost the same.

This is due to the fact that the IFLDs implemented are also very similar. If it is taken into account that for the line test the major strains are higher, this should have influenced the maximum depth and should been higher for the line simulation. It is not the case still due to the mesh size and the frequency of writing the output.

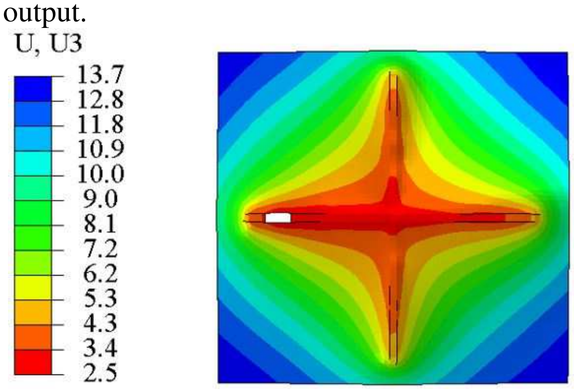

Fig. 7. Depth of failure for cross simulation

4. CONCLUSION

Line and cross tests were performed for the purpose of this paper and afterwards the IFLDs obtained were implemented in FEA software and numerical simulations were performed. The simulations showed a prediction of material failure earlier than in the case of experimental tests. This is a good approach in case of plastic deformation processes because the purpose of a simulation is to predict the material failure with a safety factor. Still the behavior of material failure in simulation can be improved by using a finer element mesh size, a higher frequency of writing output. In this research the simulation of line and cross tests was achieved and the moment of material failure was predicted within the FEA model with an accuracy of less than 3 mm for the line test and less than 4 mm for the cross test.

5. REFERENCES

[1] Centeno, G., Bagudanch, I., Martínez-Donaire, A. J., García-Romeu, M. L., Vallellano, C., *Critical analysis of necking and fracture limit strains and forming forces in single-point incremental forming*, Materials & Design, 0261-3069, 2014.

[2] Oleksik, V., Bologa, O., Breaz, R., Racz, G., *Comparison between the numerical simulations of incremental sheet forming and conventional stretch forming process*, International Journal of Material Forming, 2008.

[3] Jeswiet, J., Micari, F., Hirt, G., Bramley, A., Duflou, J., Allwood, J., *Asymmetric Single Point Incremental Forming of Sheet Metal*, CIRP Annals, 0007-8506, 2005.

[4] Rosca, N., Oleksik, M., *Simulation of the Single Point Incremental Forming of Polyamide and Polyethylene Sheets*, 9th International Conference on Manufacturing Science and Education -MSE 2019- Matec Web of Conferences 290, 2261-236X, 2019

[5] Filice, L., Ambrogio, G., Micari, F., *On-line control of single point incremental forming operations through punch force monitoring*, CIRP Ann Manuf Techn, 0007-8506, 2006.

[6] Szekeres, A., Ham, M., Jeswiet, J., *Force measurement in pyramid shaped parts with a spindle mounted force sensor*, Key Engineering Materials, 2007.

ANALIZA CU ELEMENTE FINITE A PROCESULUI DE DEFORMARE INCREMENTALĂ ȘI VALIDARE EXPERIMENTALĂ

Rezumat: Procesul de deformare incrementală necesită cercetări suplimentare înainte de a putea fi implementat pe scară largă în industrie, deoarece există anumite dezavantaje nerezolvate. Înțelegerea comportamentului materialului necesită cunoașterea diagramei limitei de rupere, permițând implementarea unui model analitic care prezice momentul ruperii acestuia. Această lucrare prezintă un model teoretic al procesului de deformare incrementală într-un singur punct, validat ulterior prin teste experimentale specifice. Procesul de validare își propune să stabilească o corelație între constatările experimentale și predicțiile analitice, facilitând aplicarea eficientă a procesului de deformare incrementală în industrie.

Mihai-Octavian POPP, Assistant researcher, Ph.D. Student, Faculty of Engineering, "Lucian Blaga" University, Sibiu, Romania, mihai.popp@ulbsibiu.ro
Gabriela-Petruța RUSU, Assistant researcher, Ph.D. Student, gabriela.rusu@ulbsibiu.ro
Mihaela OLEKSIK, Assoc.Prof.Dr.Eng., mihaela.oleksik@ulbsibiu.ro
Valentin OLEKSIK, Prof.Dr.Eng., valentin.oleksik@ulbsibiu.ro