Numerical simulation of material failure in single point incremental forming process

G P Rusu¹, M O Popp¹,²*, M Oleksik² and C Rodean³
¹ Lucian Blaga University of Sibiu, Department of Industrial Machinery and Equipment, Emil Cioran Street No. 4, Romania
² Lucian Blaga University of Sibiu, Department of Industrial Engineering and Management, Emil Cioran Street No. 4, Romania
³ Continental Automotive Systems SRL, Salzburg No. 8, Sibiu, Romania

E-mail: mihai.popp@ulbsibiu.ro

Abstract. The sheet metal parts obtained through single point incremental forming process show a thinning and also there is a high possibility of material failure. In order to avoid the failure of the material during the process, it is a good practice to use numerical simulation through finite element method to prevent this phenomenon. The aim of this paper is to investigate the prediction of the failure moment of the aluminium sheet metal using various failure criteria. Commonly used failure criteria are: Tresca or maximum shear stress criterion, von Mises yield criterion, Hill yield criteria and various invariants of the Cauchy stress tensor. For the numerical simulation of the single point incremental forming process, the semi-finished part is discretized using shell elements which allow determination of material thinning and also it is possible to observe the moment of material failure. This paper will treat also the influence of the wall angle over the moment of material failure.

1. Introduction
The incremental sheet forming process has an increasing popularity among the researchers around the world due to its high potential for manufacturing complex shape parts through its flexibility. This process takes place with the help of a parabolic or hemispherical punch that presses on the surface of the metal sheet and deforming it after various complex trajectories. These trajectories can be described by means of numerical processing centers or established by industrial robots. Due to the fact that for manufacturing of one part through incremental sheet forming takes a considerable amount of time, this process is mainly used for prototype parts and small series production. In addition, there can be manufactured parts for aerospace and medical industry.

In the paper of Nasulea et al. a new software tool was developed in order to reduce the time for preparing all the necessary data for the FEM analysis of incremental sheet forming process. This software extracts the coordinates of the interpolation points of the toolpath generated from a CAM software. In addition, the positioning time of the tool is also revealed and together with the points, coordinates are written into an input type file that can be understood by ANSYS. This brings a great advantage because the lost time for generating manually the toolpath can be removed. In the end of the paper 6 numerical simulations in ANSYS were performed, in order to validate the integration of the software developed by the authors in incremental sheet forming process. The parts used for numerical
Simulations were made out of DC05 sheet metal with 1 mm thickness and had a truncated shape with a wall angle of 40°, 50° and 60° [1]. The formation of the pieces is accomplished by combined phenomena occurring in the material, such as bidirectional stretching and bending. These two phenomena together can cause an excessive thinning of the material which can lead to material failure in different areas. The dangerous areas can be predicted by using numerical analysis through finite element method. Martins et al. have conducted a study that presents numerical simulations of the incremental sheet forming process from which various factors have emerged that contribute to the appearance of material failure during the process [2].

In the paper Yanamundra et al. a comparison between results obtained from a numerical simulation of ISF process and the experimental results was performed. The authors used the same toolpath for the finite element analysis of a trapezoid shape part as well as for the experiment performed. During the experiment, the forces on 3 axis were measured using a dynamometer. In the end of the research, forces obtained experimentally were compared with the forces resulting from the numerical simulation and the conclusion further more demonstrates that it is a good practice to use finite element analysis methods before deforming the sheet metal parts [3].

In the paper done by Blaga et al. there are treated subjects as the effects of the material thinning of a truncated shape part. Experimentally, incremental deformation of 0.7 mm DC04 steel plate was made, a material commonly used in body parts in automotive industry. The paper also treats results obtained using three types of trajectories for deforming the part. At the same time, there are measured the forces in the punch that take place during the material bending. It is compared and analysed these forces divided by 2 axis: on X axis to determine forces that appear in a horizontal plane during deformation and on Z axis corresponding to the punch, all these for 3 types of trajectories used. The paper ends with conclusions about the 3 different trajectories used for manufacturing the truncated part, thus showing that at the space spiral trajectory the forces that appear in the punch are lower compared to the other 2 trajectories [4].

In Oleksik’s paper are presented the influences of different geometric parameters on the thinning of the material. Experimental research were performed based on the influence of the wall angle of the piece. The paper also exposes the influence of the diameter of the punch that was used for the single point incremental forming process [5].

In order to predict the moment when the material failure appear, in finite element analysis programs, failure criteria are commonly used, such as: Tresca or maximum shear stress criterion, von Mises yield criterion, Hill yield criteria and various invariants of the Cauchy stress tensor. In the case of incremental sheet forming process analysis, the material failure is determined with the help of a forming limit diagram (FLD) that separates the deformation zone from the material failure zone.

2. Numerical simulation of the incremental sheet forming through finite element analysis
The aim of this paper is to determine when material failure appears in single point incremental forming process during the manufacture of a truncated piece, and one way to predict this is through finite element analysis, method that gains more and more popularity among the companies the design phase.

Two numerical simulations of the process for manufacturing two truncated parts with a wall angle of 55° and 65°, both with a depth of 25 mm, were performed in order to achieve the purpose of this paper. Figure 1 presents the trajectory and the dimensions of the truncated con with a wall angle of 55°. Both truncated parts have a radius of 40 mm on the top of the truncated con.

Figure 1. Trajectorie and dimensions of the truncated cone with a wall angle of 55°
The numerical simulation is performed using ABAQUS / Explicit finite element analysis program. The model used consists of a rigid analytical retaining ring, a deformable SHELL aluminium sheet and a rigid analytical tool. The thickness of the sheet for this model is 1mm. The aluminium sheet is meshed with 4-node, quadrilateral, stress / displacement shell elements with reduced integration and a large-strain formulation (S4R). The material used as a semi-finished product is AA5052, which is an aluminium alloy and has the following chemical concentration: Mg – 2.2%, Cr – 0.18%, Si – 0.14%, Fe – 0.31%, Cu – 0.01%, Mn – 0.05% and Zn – 0.001%. This material has the following properties: Yield strength is 243.4 MPa, ultimate tensile strength is 272.5%, percentage of elongation is 13%. In the program was introduced the plasticity of the material which was determined with the help of von Mises flow criterion. According to this criterion, the deformation enters plastic flow when the potential deformation energy for shape change reaches a critical value. The total energy of deformation per volume unit is calculated using the formula:

\[ U_1 = \frac{1}{2} (\sigma_1 \varepsilon_1 + \sigma_2 \varepsilon_2 + \sigma_3 \varepsilon_3) \]  

and the energy associated with the hydrostatic deformation can be calculated as follows:

\[ U_p = U_v = \frac{1}{2} \rho \delta = \frac{1}{6} (\sigma_1 + \sigma_2 + \sigma_3) (\varepsilon_1 + \varepsilon_2 + \varepsilon_3) \]  

where \( \delta \) is the specific deformation. The deformation energy associated with shape change is the difference of the two energies:

\[ U_f = U - U_v \]

During the incremental forming process, the tool deforms plastic the aluminium sheet metal and thus results in two types of characteristic deformations. In the first step, when the punch moves into a horizontal plane, the strain is biaxial stretched type, and in the second step when the tool penetrates the sheet metal, the combined bending and stretching appears. [6]

In order to determine the material failure during the numerical simulation of the incremental forming process, the forming limit diagram for the material used is introduced in ABAQUS. This curve is represented by using on the abscissa the values of the minor strain and on the ordered axis the values of the major strain. It has been ascertaining that the deformation curve for deep-drawing cannot be applied to the incremental forming process due to the dynamics of the forces and stresses that appears in the material. In figure 2, it can be observed the difference between the forming limit diagram used for deep-drawing and the forming limit diagram used in case of single point incremental forming process. In Mugendiran’s paper is presented a comparison between the forming limit diagram specific for the truncated square pyramid and cone produced through SPIF [7].

Figure 2. Comparison between FLD for deep-drawing and SPIF
Having implemented the deformation curve specific to the incremental forming process, we can predict the occurrence of the material failure with the help of numerical simulations through finite element analysis method. Figure 3 presents the results specific to the criterion used, which with an imposed limit shows the initiation moment of material failure.

![Figure 3. Forming limit diagram damage initiation criterion](image)

( a ) with a wall angle of 55°     ( b ) with a wall angle of 65°

In view of these considerations in the two numerical simulations performed, it can be observed in figure 4 that the material failure appears at different depths for the two inclinations used for the truncated part. In the case of the part with a wall angle of 55°, the moment of material failure appears at a depth of approximate 23.6mm whereas at the part with a wall angle of 65° at 17.5mm.

![Figure 4. Displacement on the z direction](image)

( a ) with a wall angle of 55°     ( b ) with a wall angle of 65°

On the diagram presented below are showed the forces and displacements obtained from the numerical simulation of single point incremental forming process. The red curve represents the variance of force according to displacement for the truncated con with a wall angle of 65°, whereas the blue curve represents the same parameters for the cone with a wall angle of 55°. From this diagram it can be predicted the moment of material failure in relation to wall angle of the truncated cone.
The program used for running these simulations, ABAQUS, provides output data from which can be observed the material thinning of the sheet metal during the forming right before material failure. Figure 6 shows the thinning of the material at the initiation moment of material failure and this value is approximately 44% for both simulated models for the truncated cone.

3. Conclusions
The incremental sheet forming process is complex procedure so the use of finite element method analysis is beneficial because it helps save time and materials used to produce parts. Knowing the initiation moment when the material failure appears is very important in the plastic deformation in the
process because it allows to be able to choose the right material to use in order to obtain the desired precision and shape of a part, this being impossible without the use of numerical simulations.

The use of explicit dynamic analysis for the studied process has the advantage that it is possible to determine the moment when material failure appears, as well as the depth of the truncated part in the moment of failure.

In the case of truncated parts, as expected and seen in figure 4, the material failure comes earlier at parts with a higher wall angle i.e. at 65° and depth of 17.5mm, whereas at parts with a wall angle of 55° the failure takes place at 23.6mm.

References
[1] Nasulea D and Oancea G 2018 Stroj Vestn-J Mech E 64 643-651
[2] Martins P A F, Bay N, Skjoedt M and Silva M B 2008 CIRP Ann Manuf Technol 57 247-252
[3] Yanamundra K K, Karthikeyan R and Naranje V 2017 International conference on recent advances in Materials & Manufacturing Technologies 346 1-12
[4] Blaga A and Oleksik V 2013 Adv Mater Sci Eng, 2013, 1-10
[5] Oleksik V 2014 Proced. Eng. 81 2280-2285
[6] Kim Y H and Park J J 2002 J Mater Process Technol 130-131, 42-46
[7] Mugendiran V and Gnanavelbabu A 2014 Comparison of FLD and thickness distribution on AA5052 aluminium alloy formed parts by incremental forming process 12th Global Congress on Manufacturing and Management
[8] Young D and Jeswiet J 2004 J Eng Manuf 2008 1453-1459
[9] Li J, Geng P and Shen J Int J Adv Manuf Technol 68 2637–2644
[10] Jin W, Lihua L, Pen Z, Xu W and Shufeng S 2019 J Adv Manuf Technol 101 225-234
[11] Ben H R, Thibaud S, Richard F, Hapsari G and Malecot P 2017 Numerical prediction of the forming limit diagrams of thin sheet metal using SPIF tests 17th International Conference on Sheet Metal