Prediction of Negative Bulge in Two Point Incremental Forming of an Asymmetric Shape Part

To cite this article: Jaekwang Shin et al 2018 J. Phys.: Conf. Ser. 1063 012057

View the article online for updates and enhancements.
Prediction of Negative Bulge in Two Point Incremental Forming of an Asymmetric Shape Part

Jaekwang Shin\textsuperscript{1}, Ankush Bansal\textsuperscript{1}, Maya Nath\textsuperscript{2}, Randy Cheng\textsuperscript{2}, Mihaela Banu\textsuperscript{1}, Alan Taub\textsuperscript{1,2}, Brian Martinek\textsuperscript{3}

\textsuperscript{1}Department of Mechanical Engineering, \textsuperscript{2}Department of Material Science and Engineering, University of Michigan, Ann Arbor, MI, USA
\textsuperscript{3}Boeing Company, Berkley, MO, USA

E-mail: mbanu@umich.edu

Abstract. The bulging effect in incremental sheet forming (ISF) has been explained by the build-up of compressive stresses during forming resulting in the material at the center to bulge inward or outward. In this study, effect of step size on the formation of a bulge in an asymmetric shape in two point incremental forming (TPIF) is examined. Experimentally validated finite element models were developed to simulate the forming at both the macro scale, and using a finer mesh to generate the surface profile. The formation of the bulge is determined to be due to a combination of compressive stresses and an instability created by the material in the underformed area during the forming operation.

1. Introduction

Incremental sheet forming (ISF) is an agile method of sheet metal forming with applications in low volume production and prototyping. In ISF, a blank sheet is clamped between two blank holders or a die and deformed locally with a tool following a predefined tool path to reach the final shape. This process is suitable for automotive prototyping and aerospace production, where the number of parts that are manufactured is low, avoiding the high amortized tooling cost associated with conventional sheet forming technologies such as stamping as suggested by Jeswiet et al. \cite{1}.

Much of the research in ISF has been conducted on the single point incremental forming (SPIF) process. However, one of the major disadvantages of SPIF is the bending induced between the tool and the clamp leading to large geometric deviations \cite{2}. To resolve this problem, alternative methods are utilized including two point incremental forming (TPIF) which utilizes a die \cite{3} and double sided incremental forming (DSIF) which introduces a second tool.

In TPIF, the material is squeezed between the tool and the die leading to an instability in the unformed areas. The defect created by this instability is called “bulging effect” and is characterized by a spatial curvature of the surface in the areas which are supposed to be flat. Ambrogio et al. \cite{4,5} reported this effect while studying the springback during incremental forming. Hussain et al. \cite{6,7} tried to correlate forming parameters with bulge formation using response surface method. Al-Ghamid et al \cite{8} reported the bulge tendency of the material in SPIF using the finite element method (FEM) in connection with the hardening of the material, forming depth, and tool size. Isidore et al. \cite{9} tried to predict the bulging effect by changing the shape of the tool using FEM. It was shown that compressive stress is built up at the region of the tool contact inducing bulging and changing from hemispheric tool to flat headed tool.
decreases compressive stress which results in a lower bulge. Mohammadi et al. [10] concluded that depending on wall angle, the direction of radial stress could be different and that application of laser heating could improve geometrical accuracy in low angled parts.

In this research, the correlation between the step size and the bulging effect is investigated for an asymmetric shape using TPIF setup. A new approach to quantify bulge is proposed utilizing two models.

2. Experimental Method

A set of experiments was performed using TPIF to investigate formation of a negative bulge at the bottom of an asymmetric shape part, known as heart shape, as shown in Fig. 1. Experiments included forming of a 1.63mm thickness of Al 7075-O sheets. The process parameters used in the experiments are: (i) feed rate, (ii) tool diameter, (iii) step size of 0.50mm and 0.63mm and (iv) programmed squeeze factor of 10%.

![Figure 1. The heart shape part (a) cross cut planes used for measuring the dimensional accuracy, (b) cross section A-A, and (c) cross section B-B](image)

In TPIF process, programmed squeeze factor (pSF) quantifies the amount of material intended to be squeezed between forming tool and die. Assuming that sine law estimates the local sheet thickness, squeeze factor can be defined as a ratio of distance between tool contact point and die surface to local sheet thickness [11]. However, elastic deflection of forming tool and intrinsic machine compliance effectively reduces the amount of intended squeeze. Therefore, effective squeeze factor (eSF) at any instant can be represented as equation (1), where Δ is summation of tool deflection and machine compliance. The eSF will always be less than pSF and consequently less material will be squeezed between tool and die than expected. Empirical relations to estimate machine compliance and tool deflection are produced based on forces which is later employed to calculate Δ.

\[
pSF = \left(1 - \frac{d}{t \times \sin(90 - \alpha)}\right) \times 100, \quad eSF = \left(1 - \frac{d + \Delta}{t \times \sin(90 - \alpha)}\right) \times 100
\]

where \(d\) is the squeezed thickness, \(t\) is the original thickness, and \(\alpha\) is the wall angle.

3. Finite element Model for Simulation of TPIF

3.1 TPIF Finite Element Model

To model the TPIF for the heart shape part, Abaqus 6.14-1 is used using the dynamic explicit integration scheme. A 400 × 400mm blank sheet is clamped on the contour and formed using a finger-tip hemispheric tool. The tool, die and the top clamp were modelled with discrete rigid bodies using a 4-node 3-D bilinear rigid quadrilateral element. The mesh size of tool and the die is approximately 0.2mm to avoid any distortion due to the roughness induced by rigid tool or the die. The blank was meshed with a 8-node linear brick element with reduced integration and hourglass control. A blankholding force of 300kN is applied on the contour of the blank, to ensure no material is drawn in during the forming
process. The computation was conducted using 120 cores of 2.8 GHz Intel Xeon E5-2670v2 processors provided by supercomputing center, ARC-TS of University of Michigan.

A mass scaling factor of $10^5$ and no time scaling factor were set-up. The speed of the tool was set up as in the experiment. The energy ratio of the kinetic energy to the total energy was monitored during simulation to ensure that it does not exceed 1% and inertia effect is within acceptable limits.

The tool path was generated using a tool path generation software specialized for incremental forming, created by AMPL research group at Northwestern University. A contour tool path strategy was used with step sizes of 0.50mm and 0.63mm. The eSF was calculated for each step size as 1.74% and 1.21% respectively.

3.2 Material Properties

The material properties of the AA7075-O alloy were determined from tensile tests following the ASTM standard for aluminium sheets. The elastic properties are Young Modulus, $E = 70$ GPa and Poisson ratio, $\nu = 0.33$. The 0.2% offset method is applied for the stress–strain curve obtained from tensile tests, to determine the plastic part of the curve. The material model is isotropic hardening with Hill’s 48 quadratic yield surface. Voece law, equation (2) was used to describe the isotropic hardening behaviour,

$$\sigma = k_0 + Q(1 - e^{-\beta\varepsilon}),$$

where $k_0$, $Q$, and $\beta$ are the tensile yield strength, saturation stress and material constant, respectively. These values were identified by fitting the stress strain curve obtained from tensile tests. The values are: $k_0 = 91.3025$ MPa, $Q = 149.34$ MPa, $\beta = 26.71$. To identify the anisotropy of the sheet, tensile tests were performed for 0°, 45°, and 90° with respect to the rolling direction. The Lankford coefficients $r$ are $r_0 = 0.7396$, $r_{45} = 0.4474$, and $r_{90} = 0.8865$. This anisotropy is introduced in the model through six yield stress ratios. Due to the planar anisotropy, three of these yield stress ratios ($R_{11}$, $R_{13}$, $R_{23}$) are 1, and the other three ($R_{12}$, $R_{22}$, $R_{33}$) are calculated based on Lankford coefficients [12]: $R_{12} = 1.2254$, $R_{22} = 1.0513$, $R_{33} = 0.9739$. Also tension and compression tests has been conducted on the same material [13]. The difference between the two curves was less than 15MPa, which is less than 10% of the forming stress. This leads to the conclusion that tension curves can be used in the model, which covers tension-compression behavior with insignificant errors.

4. Results and Discussions

4.1. Finite Element Model Validation

To validate the TPIF finite element model for the heart shape part, the thickness variation along the wall of the part profile and the forming force were compared between simulation and experiment using

![Figure 2](image-url). Forming force (a) and thickness (b) comparison between experiment and simulation.
0.50mm step size and 0% squeeze factor. In simulation, this profile was obtained by calculation of the thickness for each top and bottom element across the thickness, and plotted along the same profile as the experimental measurement. The results show a very good correlation between experiment and model for both thickness and forming forces, as shown in Fig. 2.

To understand the origin of the bulge formation, principal stress components were compared for the two step sizes (Fig. 3a,b) as well as the dimensions of the bulge predicted through the TPIF model and measured from the experiments (4.5mm versus 9mm).

- Principal stress components in the elements near the bottom of the walls were analyzed to confirm whether compressive stresses are able to produce the bulge, as proposed by Isidore et al. [9]. Fig. 3 shows the compressive stress component, \( \sigma_{22} \), along the wall, in the local coordinate system. The plot of the evolution of \( \sigma_{22} \) in the top, middle and bottom elements against the distance from the edge of the sheet in lateral direction shows that in the bulge area compressive states are mostly induced in the bottom element while the top element is in stretching.

![Stress components](image.png)

**Figure 3. Stress components the wall calculated from the TPIF simulation**

The difference between the stresses which develop in the global model is insignificant between the 0.50mm and 0.63mm step size contradicting the result of the experiment where 0.63mm step size shows a lower bulge. It can be concluded that the mechanism of bulge development cannot be fully explained by the effect of the compressive stresses along the wall.

- An in-depth analysis of the predicted bulge height (as defined in section B-B, Fig. 1) and the measured one shows that the TPIF model provided two times bulge values compared with the experiment (Fig. 4). One potential source is that the step size in the tool path (less than 1mm) is smaller than the mesh size (1mm) resulting in the inability of the model to capture geometrical changes smaller than 1 mm. To overcome this challenge, it is proposed a high-resolution finite element sub-model that is able to simulate material flow during caused by squeezing during forming.

4.2. High-Resolution Simulation Using a the Finite Element Sub-Model

To better understand the origin of the effect of the step size on the bulge formation, a computation-efficient finite element sub-model was built, which is a segmentation of the TPIF model in the flat walled section in an effort to predict the surface profile during forming (Fig. 4). Thus, the sheet size was modeled to 10\( \times \)12\( \times \)1.63mm and the mesh size of 50\( \times \)50\( \mu \)m with six solid elements across the thickness.

Two step size values, 0.50mm and 0.63mm, were used for modeling. Only seven passes of the tool were considered identical to the original tool path used in the full TPIF model. Boundary conditions were imposed on the edges of the sheet to simulate actual conditions from TPIF. The blank holder side edge was fixed while the opposing side was constrained in x and y while allowing movement in z. The edges perpendicular to the tool path were constrained in the direction of the tool path.

The sub-model was able to simulate the surface profiles with repetitive half sine waves with the period corresponding to each step size as seen in Fig. 5. The surface profiles were experimentally identified on the incremental formed parts [14]. A single profile was selected at the mid-point of the
simulation to minimize any end effect of the boundary condition. The profile was then rotated to align with the coordinate axis to allow easier post processing. The bulge height in the section $A - A$ was analyzed. While the 3-dimensional profile of the bottom of the part could be different depending on the surrounding wall geometry conditions, it can be assumed that bulge profile in section $A - A$ would only be influenced by the amount of material that is being pushed in from the wall into the bottom of profile. Thus, using equations (3) and (4), the amount of the material that has been moved in the bulge area has been calculated from simulation and compared with the experimental values.

$$\Delta l_{\text{exp}} = l_{\text{bulge}} - l_0$$
$$\Delta l_{\text{model}} = \frac{2n_s (\sum A_m - \sum A_r)}{t},$$

where $l_{\text{bulge}}$ and $l_0$ are the measured bulge and reference length in experimental part, $A_m$ and $A_r$ are the area under one period of the surface profile for modeled surface and sine law profile respectively, $n_s$ is the number of cycles that the tool did to reach the bottom of the part and $t$ is the thickness of the sheet. The results of the calculation versus experiments, for 0.5 step size, are $\Delta l_{\text{model}} = 1.9533\text{mm}$, $\Delta l_{\text{exp}} = 1.9772\text{mm}$ and for 0.63 step size $\Delta l_{\text{model}} = 0.7883\text{mm}$, $\Delta l_{\text{exp}} = 0.6996\text{mm}$.

Based on this calculation, a surface profile consisting of average of 0.83mm period between peak to peak was created with the distance of 16$\mu$m from peak to valley for 0.50mm step size and 1.0436mm between peak to peak and distance of 20$\mu$m between peak and valley for 0.63mm step size. It can be observed that the area difference between the surface profile of 0.50mm step size and the sine law curve is greater compared to that of the area difference between 0.63mm step size and sine law (Fig. 5). It can be concluded that more material is being pushed in to the bulge area for 0.50mm step size compared to 0.63mm step size although peak to peak period and peak to valley amplitude is larger than that of 0.50mm by 4$\mu$m. In 2D, this calculated area of the material results in compressive state for the undeformed thickness leading to a buckling phenomenon. Thus, buckling analysis is used to convert lateral length change to vertical displacement (bulge height). The bottom section of the heart where addition of the material results in the buckling of the part can be considered simply supported element.

![Figure 4](image-url)  
**Figure 4.** Modeled result of the TPIF model and the sub-model

![Figure 5](image-url)  
**Figure 5.** Surface profile for a determined segment on the wall

![Figure 6](image-url)  
**Figure 6.** Maximum bulge height measurement
The vertical displacement then follows second order differential equation (5) if approximated to a beam:

$$EI \frac{d^2w}{dx^2} + Pw = 0,$$

where \( w \) and \( x \) being the vertical and lateral displacement, \( E, I \), and \( P \) being elastic modulus, moment of inertia and compressive load respectively. Using simple support boundary condition, the solution of then becomes equation (6):

$$w = A \sin \frac{n \pi x}{l_0},$$

where \( A \) is the amplitude , \( n \) the number of half sine wave and \( l_0 \) the base length, with \( n=1 \).

The results show that calculated maximum displacement is 4.3mm for 0.50mm step size and 3mm for 0.63mm step size. These values obtained using the sub-model were added to the plot in Fig. 6. Thus, using the sub-model results, the bulge formation can be explain as a combination of compressive stresses and a buckling phenomenon due to the accumulation of the material during forming which is dependent on combination of step size and squeeze factor.

6. Conclusions

The correlation between step size and bulging was investigated using two step size values, e.g. 0.50mm, and 0.63mm for an asymmetric part, known as heart shape. Two finite element models were built for simulation the TPIF process and the surface profile generated during forming. The analysis of the material movement during forming revealed the following conclusions: i) The mechanism of the bulge is due to material being pushed forward due to the combination of the squeeze factor and step size. The amount of the material being transfers was calculated per cycle and computed to predict total material that is being added to the bottom of the heart which was then compared with experimental measurement; ii) The maximum bulge height of the part was predicted using buckling analysis where the displacement of the buckled element was back calculated from the geometry of the bulge, which is based on the amount of material accumulated in front of the tool during forming of the walls and iii) To minimize bulge, surface profile has to be considered which is influenced by both squeeze factor and step size.

Acknowledgments

The research was funded by DOD-ONR N00014-14-2-0002 - LIFT 0007A-4 project through the American Lightweight Materials Manufacturing Innovation Institute – LIFT. The authors grateful thank to their partners Boeing Company, for allowing to use the heart shape part geometry, and Northwestern University for providing the tool path generator and compression test data.

Reference

[1] Jeswiet J, Micari F, Hirt G, Bramley A, Duflou J and Allwood J 2005 CIRP Annals 54 88-114
[2] Malhotra R, Cao J, Ren F, Kirieda V, et al. 2011 J. of Man. Sci. & Eng. 133 061019
[3] Silva M and Martins P 2012 J. of Mat. Eng. and Performance 22 1018-1027
[4] Ambrogio G, Filice L, De Napoli L and Muzzupappa M 2005 Proceedings of the Institution of Mechanical Engineers, Part B: J. of Eng. Man. 219 823-830
[5] Ambrogio G, Cozza V, Filice L and Micari F 2007 J. of Mat. Proc. Tech. 191 92-95
[6] Hussain G, Gao L and Hayat N 2011 Mat. and Man. Proc. 26 1546-1553
[7] Hussain G, Al-Ghamdi K, Khalatbari H, Iqbal A and Hashemipour M 2014 Mat. and Man. Proc. 29 454-460
[8] Al-Ghamdi K and Hussain G 2014 Proc. of the Inst. of Mech. Eng., J. of Eng. Man. 229 744-753
[9] Isidore B, Hussain G, Shamchi S and Khan W 2016 J. of Mech. Sci. and Tech. 30 2151-2161
[10] Mohammadi A, Vanhoove H, Van Bael A and Duflou J 2014 Int. J. of Mat. Forming 9 339-351
[11] Ren H, Lia F, Moser N, Leem D, Li T, Ehmann K and Cao J 2018 CIRP Annals
[12] Dassault Systems, 2011, “Abaqus Analysis User’s Manual (v.6.11),” Providence, RI, USA
[13] DOD-ONR N00014-14-2-0002 - LIFT 0007A-4 Project weekly report
[14] Nath M, Shin J, Odykirk T, Banu M and Taub A 2017 Proc. of Mat. Sci. & Tech. Conf. MS&T.