Goals & Intent

FEA Information Engineering Journal is published monthly in pdf format by FEA Information Inc. To showcase 3 to 5 papers.

The focus is toward publications of papers, or other media geared for Finite Element Analysis, specifically on, and limited to, the software LS-DYNA and it’s related software LS-PrePost, LS-OPT, LS-TaSC, Dummy and Barrier models, including software that interface and/or enhance LS-DYNA in the engineering community.

Consideration is given to all aspects of technially excellent written information without limitation on length. All submissions must follow guidelines for publishing a paper, or periodical. If a paper has been previously published, permission to reprint is required with proper acknowledgment given to the publisher of the work.

Reproduction in whole, or part, without written permission of FEA Information Engineering Journal or the owner of the copyright is strictly prohibited. FEA Information Journal welcomes unsolicited topics, ideas, and articles.

FEA Information Engineering Journal will be published monthly and limited to a maximum of ten papers either re-print or new for publication by FEA Information Journal.

FEA Information Engineering Journal papers will be published on www.feapublications.com and www.feaengineeringjournal.com

For Information on publishing a paper, or recommending a paper for re-print in the FEA Information Engineering Journal contact mv@feainformation.com

Subject Line Should Be: Journal Publication
Prediction of Springback in CNC Tube Bending Process Based on Forming Parameters
Levent Sözen, Mehmet A. Guler, Recep M. Görgüüarslan, Engin M. Kaplan
- University of Economics and Technology

Process Modeling of Freeform Incremental Forming Using LS-DYNA®
Feng Ren, Zhen Cui, and Z. Cedric Zia Ford Motor Company

Numerical Simulation and Experimental Study of Electromagnetic Forming
Kianhui Shang, Larry Wilkerson, Steve Hatkevich; American Trim LLC
Pierre L'Eplattenier,, LCTC

Published in the 11th International LS-DYNA® Users Conference 2010 Proceeding.

All contents are copyright © to the publishing company, author or respective company. All rights reserved.
Prediction of Springback in CNC Tube Bending Process Based on Forming Parameters

Levent Sözen, Mehmet A. Guler, Recep M. Görgüürarslan, Engin M. Kaplan
Department of Mechanical Engineering
TOBB University of Economics and Technology
Ankara, TURKEY

Abstract

CNC tube bending machines are commonly used in several industries such as automotive, aerospace and shipping. Especially in automotive industry, usage of tube formed geometries is common because they provide weight reduction without loss of strength. Obtaining desired dimensions and geometries is a necessity for design engineers to achieve high quality end-products. One of the easiest ways of having high quality tube formed end-products without the need of welding operations is using CNC tube bending techniques. The most common problems encountered during tube bending operations are thickness reduction, ovalisation, wrinkling and springback. Especially; springback which is defined as the deviation from the predefined bend angle after the bending operation performed is an undesirable condition that causes some difficulties in the assembly process. It depends on various geometrical parameters such as thickness of the tube, bend angle and effect of mandrel type used. Occurrence of springback is also dependent on forming parameters such as friction coefficient between dies and tube, internal pressure that is applied to the tube and axial loading. In the design stage, determination of springback and various parameters affecting springback behavior by experimental methods is quite cumbersome and costly. Therefore, prediction of springback by virtual methods such as finite element method (FEM) would shorten the time and reduces the cost of the pre-determination of springback.

1. Introduction

CNC tube bending operation is a manufacturing process commonly used in various industries such as automotive, aerospace, shipping, agriculture, furniture, ornament, outdoor play systems, etc. Generally steel and aluminum tubes are bent plastically by utilizing CNC tube bending machines. Compression, draw, ram, roll and press tube bending methods are used frequently in order to bend tubes made of various materials to different angles and radii [1,2]. Rotary draw tube bending is the most common, useful and flexible bending method among the types of tube bending processes mentioned above. It provides low production cost because of low scrap rate and have variety of tooling options. High quality, accuracy, repeatability, high volume production and process automation are the most conspicuous advantages of rotary draw tube bending operations. Tight radii and thin wall bent tubes can be easily obtained by using CNC technology [1-3]. The tooling of this machine basically includes four components: a bend die, a pressure die, a clamp die and a wiper die [3] as shown in Figure 1 where a solid model of rotary draw tube bending components is demonstrated.

The fundamental steps of rotary draw tube bending operation are described by [2-4]:

- The bend die assistant, clamp die and pressure die are parallel to the feeding axes before the start of the bending operation.
- The clamp die presses the tube tightly against the bend die for the purpose of preventing the tube sliding between them during the bending process.
Figure 1. Components of rotary draw tube bending

- The bend die and clamp die then rotates together so that the bend die draws the tube along with it and against the pressure die, by the way this rotation provides plastic deformation at intrados and extrados sections of the tube.
- The pressure die pushes the tube at the outer surface of it to reduce the thinning of the tube and also assists to completing the bend by supplying additional torque during the bending operation.

In bending operations of tubes having large diameters or thin walls, the tooling can also be equipped with an internal mandrel, a wiper die and a booster [3]. The mandrel is used for preventing the cross-section of the tube from ovalising and collapsing during bending [4]. The mandrel usually remains stationary during the bend, but retracting mandrel near the end of the bending operation may be useful in order to reduce wrinkling and flattening [4]. The wiper die provides additional support to the tube just behind the tangent point of the bend and works in conjunction with the mandrel to prevent wrinkling at the intrados of the tube. The wiper die also minimizes frictional drag during bending operation [3,4]. Booster is a component that provides axial loading to the tube during bending operation [3].

There are many papers that include experimental, numerical and analytical studies about rotary draw tube bending operations in the literature. Pan and Stelson [5] introduced an analytical study on the relation between curvature of the tube and the cross sectional deformation. In their study, plastic deformation in the bent part of the tube and elastic deformation in the pressure die area were represented as the two main reasons of the springback occurrence. A comprehensive experimental study was carried out by Khodayari [6]. In his study, rotary draw tube bending machine was used for the experiments and the effect of different tube materials such as aluminum and steel on springback, cross-section, ovality, and wall thickness change were investigated. Shr [1] performed finite element rotary draw tube bending simulations both for forming and springback processes and compared his own results with the experimental data of Khodayari [6]. Wang and Agarwal [7] also performed finite element studies and they applied axial load with internal pressure to their models in order to improve process quality. They achieved reducing wrinkling tendency and cross-sectional distortion. Gu et al [8] focused on springback occurrence for thin walled tubes at the end of the bending operation. The author
performed bending simulations using the model which includes mandrel and classified the whole tube bending process into three steps that are bending tube, retracting mandrel and springback. Da-xin et al[9] investigated the effect of tube’s geometrical parameters on the occurrence of springback by performing different bending simulations for various parameters. The effect of material properties on the springback occurrence were also analyzed and they realized the difference between mechanical properties obtained from standard bar tensile tests specimens and from specimens which were taken from metal tube.

Most of the studies in literature only analyze the forming processes; therefore they mostly consider the state of thickness variation and ovalisation problems. On the other hand, the studies investigating the springback behavior consider the effect of parameters such as material properties, wall thickness, tube diameter, bend angle and radius by using both experimental and finite element methods. In this study, in addition to parameters which are mentioned above, the effect of various parameters such as friction coefficient, mandrel type, axial loading and internal pressure on the springback was investigated and the investigation was expanded with the intention of determining dominant parameters that have a bearing on springback. The nonlinear explicit finite element code LS-DYNA® was used for the forming analysis. The results are validated by using the experimental results in literature.

2. Forming Simulation

Rotary draw tube bending operation was simulated by the using LS-DYNA. The results of the simulations were validated by the data obtained from Khodayari’s experimental study[6]. The simulations were performed using the material data and the geometrical properties given in Table 1 which are used in Khodayari’s experiments [6].

<table>
<thead>
<tr>
<th>Material</th>
<th>A573-81 65</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield Strength (MPa)</td>
<td>270</td>
</tr>
<tr>
<td>Tangent Modulus (MPa)</td>
<td>900</td>
</tr>
<tr>
<td>Young Modulus (GPa)</td>
<td>219.4</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.3</td>
</tr>
<tr>
<td>Outside Diameter of Tube (mm)</td>
<td>20</td>
</tr>
<tr>
<td>Thickness of Tube (mm)</td>
<td>1.5</td>
</tr>
<tr>
<td>Bend Radius (mm)</td>
<td>50</td>
</tr>
</tbody>
</table>

Finite element mesh is generated for the four basic components (i.e. clamp die, bend die, pressure die and wiper die) of the tube bending machine in ANSA. The element size was set to 3 mm as shown in Figure 2. The dies were modeled as rigid and the tube was modeled as piecewise linear plastic using MAT-024 from the LS-DYNA material library.

In order to simulate the contact behavior between the tube and the dies, surface to surface contact type was used with the “CONTACT_ONE WAY_SURFACE TO_SURFACE” keyword in LS-DYNA. The friction coefficient values are given in Table 2.
Bending operation starts with the rotation of bend die and clamp die which grips the tube; by the way tube had a displacement towards axes between wiper die and pressure die. In order to create the necessary moment for the bending operation, pressure die applies 50 MPa pressure to the tube. It also reduces the thinning of the tube’s outer surface. Bending simulation was terminated when bend radius reached 90°. At the end of simulation, cross sectional distortion occurred and thinning took place in extrados of the tube whereas thickening occurred at the intrados.

In Figure 3, thickness reduction contour is plotted by using LS-PREPOST. Minimum and maximum thickness values were measured as 1.378 mm at an element on the extrados of the bent tube and 1.679 mm at an element on the intrados as expected. The elements, which the thickness data were obtained, are given in Figure 3. Thickness reduction graph is plotted in Figure 4 by using the thickness data which was obtained from a cross-section of the tube near the plane of having 45° of angle with the feeding axes. The simulation results are compared with the analytical results that obtained by Eq.A.10 given in Appendix from Kervicks’s[10] and Shr’s model [1], experimental data of Khodayari [6] and simulation results from Agarwal’s study [2].

Effective plastic strain values were measured from the elements which were detected from the extrados and intrados segments of the bent tube by using LS-PREPOST. These data were compared with the analytical results obtained from the Eq.A.3 from the study of Shr[1]. Maximum effective strain value at the extrados segment was calculated to be 0.185 and it is considerably close to the maximum effective strain value 0.187 measured from the model which was bent to 90°. Comparative plots are shown in Figure 5.
Figure 3. Thickness reduction contour

Figure 4. Thickness reduction
At the intrados segment of the tube, maximum effective plastic strain value was measured as 0.19 from the finite element model. Simulation results were also compared with the maximum effective strain calculated by analytical model and comparative plots are given in Figure 6.
3. Springback Simulation

Springback prediction by analytical methods may not give satisfactory results due to the several parameters involved such as geometrical, mechanical and forming parameters. It is therefore necessary to use the finite element method to predict the springback angle. In this section rotary draw bending simulations were performed for different geometrical parameters such as tube thickness, bend angle, internal mandrel type respectively. Moreover simulations were repeated for forming parameters such as coefficient of friction, internal pressure and axial pull. First of all, rotary bending simulation was achieved by LS-DYNA and dynain file was generated at the end of the simulation. By using dynain file, springback simulations were performed for the mechanical and geometrical properties which are given in Table 1. Springback values for different bend angles were compared with the experimental results of Khodayari’s study as shown in Figure 7. In order to determine the effect of internal pressure and axial loading; 10 MPa internal pressure (P_I) and 7.2 kN axial loading (F_a) were applied to the model in separate simulations and the results are also compared in Figure 7.

![Springback values for different bend angles](image)

**Figure 7.** Springback values for different bend angles (D = 20 mm)

Thickness and friction coefficients are also the parameters that affect the occurrence of springback. Springback angles for various values of thickness and friction coefficients between tube and dies are given in Table 3 and Table 4 respectively. Springback angle were measured at the end of the 90° bending simulation.

**Table 3.** Springback values for different tube thickness

<table>
<thead>
<tr>
<th>Thickness of the tube (mm)</th>
<th>Springback angle for 90° bend angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
<td>1.9°</td>
</tr>
<tr>
<td>1.5</td>
<td>1.3°</td>
</tr>
<tr>
<td>2.0</td>
<td>1.3°</td>
</tr>
</tbody>
</table>
Table 4. Springback values for different friction coefficients

<table>
<thead>
<tr>
<th>Coefficient of friction between tube and dies</th>
<th>Springback angle for 90° bend angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.05</td>
<td>1.6°</td>
</tr>
<tr>
<td>0.10</td>
<td>1.6°</td>
</tr>
<tr>
<td>0.15</td>
<td>1.5°</td>
</tr>
<tr>
<td>0.20</td>
<td>1.4°</td>
</tr>
</tbody>
</table>

Springback behavior is also investigated for different geometrical and mechanical properties which are given in Table 5. This model was agreed with Shr’s study[1] and springback was also calculated for different bend angles. Material model of the dies were rigid and the tube material was modeled as power law plasticity using MAT-018 from the LS-DYNA material library. In Figure 8; variation of springback angles with respect to bend angles are shown for 10 MPa and no internal pressure. The data are also compared with the analytical results of Shr[1], which were obtained from Eq.A.16.

Table 5. Mechanical and geometrical properties [1]

<table>
<thead>
<tr>
<th>Material</th>
<th>SS 304</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield Strength (MPa)</td>
<td>215</td>
</tr>
<tr>
<td>Young Modulus (GPa)</td>
<td>210</td>
</tr>
<tr>
<td>Strain hardening coefficient (MPa)</td>
<td>1451</td>
</tr>
<tr>
<td>Strain hardening exponent</td>
<td>0.6</td>
</tr>
<tr>
<td>Poisson’s Ratio</td>
<td>0.3</td>
</tr>
<tr>
<td>Outside Diameter of The Tube (mm)</td>
<td>57.15</td>
</tr>
<tr>
<td>Thickness of The Tube (mm)</td>
<td>2.858</td>
</tr>
<tr>
<td>Bend Radius (mm)</td>
<td>171.45</td>
</tr>
</tbody>
</table>

Figure 8. Springback values for different bend angles (D = 57.15 mm)
Figure 9. Springback values for different bend angles (D = 60 mm)

Figure 10. Mandrels with different ball numbers

Effect of mandrel type on the springback occurrence was also investigated in this study. Springback angles were measured for 30°, 60° and 90° bending simulations for models having different number of mandrel balls in Figure 9. Finite Element model of the mandrels, which have different number of balls are given in Figure 10. For the models equipped with internal mandrels,
mechanical and geometrical properties were the same as in Table 5, except outer diameter of the tube was taken to be 60 mm and clearance between mandrel and tube were 0.286 mm.

Springback results are given for the models which had 1 ball, 2 balls and 3 balls internal mandrel in Table 6 respectively.

Table 6. Springback angles for different mandrel model

<table>
<thead>
<tr>
<th>Ball number for internal mandrel</th>
<th>Springback angle for 90° bend angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 ball</td>
<td>1.75°</td>
</tr>
<tr>
<td>2 balls</td>
<td>1.78°</td>
</tr>
<tr>
<td>3 balls</td>
<td>1.76°</td>
</tr>
</tbody>
</table>

4. Results

In this study, both forming and springback simulations were performed for the rotary draw tube bending operations and the results were compared with the experimental, numerical and analytical studies in literature. The following conclusions can be drawn from this study:

- The thickness distribution obtained after the forming simulations were compared with the experimental, numerical and analytical studies in the literature and very good agreement was observed. The effective strain distribution at the intrados and extrados were compared with the analytical study in the literature and again good agreement was observed.
- Springback angles were obtained from simulations for various bend angles and it was observed that as the bend angles were increased the amount of springback was also increased.
- Axial pull and internal pressure were applied and their effect on the springback angles was determined. As the axial pull was applied, the springback angles decreased on most of the bend angles, the internal pressure were determined to lower the springback.
- As the thickness of the tube was enhanced the springback angle reduced.
- When the friction coefficient between the tube and the dies was increased, the springback angle decreased.
- To see the effect of the number of mandrel balls used, models having one, two and three balls have been prepared and the springback angles were measured. It was concluded that there was not much effect of number of mandrel balls used on the springback angle.

Acknowledgments

The authors wish to acknowledge Ulrich Stelzmann of CADFEM and Mehmet F. Aycan for their contributions to this study. Financial support for this project was provided by Turkish Ministry of Industry and Trade and Doganer Makine A.S. within the scope of the SAN-TEZ project 00370.STZ.2009–1.
References


Appendix

A.1. Normal Strain Distribution on the Cross-Section of the Bent Tube [1, 10]

The coordinate system of a bent tube is shown in Figure 11. Let R be the radius of curvature measured at the mid-plane of the tube and let h be the distance of an element from the mid-plane. Moreover, let L be the arc length at h and let $L_0$ be the arc length at the mid-plane. $L_0$ is constant during bending operation and can be written as below.

$$ L_0 = R \cdot \theta $$

(A.1)

At h, the arc length L can similarly written as below.

$$ L = \theta \cdot (R + h) $$

(A.2)

In Eq. A.2, h can be expressed as $h = r \cdot \sin \phi$ where r is the radius of the tube and $\phi$ is the angle on the cross-section. By using these equations the nominal strain distribution in the axial direction is written as below.

$$ \varepsilon_z = \frac{L - L_0}{L_0} = \frac{h}{R} = \frac{r \cdot \sin \phi}{R} $$

(A.3)

Also, the plastic strain value on the cross-section, $\varepsilon_z$, can be written as:

$$ \varepsilon_z = \ln \frac{L}{L_0} = \ln (1 + \frac{h}{R}) = \ln (1 + \frac{r \cdot \sin \phi}{R}) $$

(A.4)

The plastic strain in tube axial direction, $\varepsilon_z$, is defined as negative inside of the neutral plane (compression) and positive outside of the plane (tension). The strain is linearly distributed with the distance from any point on the tube to the neutral plane.
A.2. Thickness Distribution on the Cross-section [1, 10]

Volume of the tube material is considered as constant and the material is considered as homogeneous; in order to obtain the thickness distribution on the cross-section easily. The length of the bent tube at any point on the cross-section is obtained with the following equation by using the geometrical parameters of the bent tube.

\[ L = L_0 (1 + \frac{r \sin \phi}{R}) \quad (A.5) \]

The total strain in every direction at an infinitesimal element must be zero if the volume is assumed to be constant.

\[ \varepsilon_z + \varepsilon_r + \varepsilon_\theta = 0 \quad (A.6) \]

It is assumed that pure bending moment subjects to the tube, so any fiber of the tube is subjected to normal stress only in axial direction. Therefore, the strain in the radial direction becomes as follows.

\[ \varepsilon_r = \varepsilon_\theta = -\frac{1}{2} \varepsilon_z \quad (A.7) \]

If plastic strain (Eq. A.4) is substituted into Eq. A.7;

\[ \ln \left( \frac{t}{t_0} \right) = -\frac{1}{2} \ln \left( \frac{L}{L_0} \right) \quad (A.8) \]

By substituting Eq. A.5 into Eq. A.8, the thickness distribution on the cross-section expressed as the following equation.

\[ t = \frac{1}{\sqrt{1 + k \sin \phi}} t_0 \quad (A.9) \]

In this equation, \( k = r/R \) is the ratio of the tube radius to the bending radius. Therefore, the thinning distribution on the cross-section becomes:

\[ \text{Thinning(\%)} = \frac{t - t_0}{t_0} = \left( \frac{1}{\sqrt{1 + k \sin \phi}} - 1 \right) \cdot 100\% \quad (A.10) \]
A.3 Springback Angle [1, 10]

While the machine load released from the bent tube, the total bend angle decreases and the radius of bent increases simultaneously due to the springback. In analytical calculations, it is assumed that the total bending length in the deformation zone remains the same at all time. Let $\theta_L$ and $R_L$ be the bend angle and bend radius in loaded condition; let $\theta_U$ and $R_U$ be the parameters for unloaded one. Total bending length $L$ can be written as:

$$L = \theta_L \cdot R_L = \theta_U \cdot R_U$$  \hspace{1cm} (A.11)

The springback angle $\Delta \theta$ is defined as:

$$\Delta \theta = \theta_L - \theta_U$$  \hspace{1cm} (A.12)

and the curvature $(K=1/R)$ changes due to the radial growth is defined as:

$$\Delta K = K_L - K_U$$  \hspace{1cm} (A.13)

The curvature changes due to the springback can be calculated in terms of bending moment $M_L$ as follows.

$$\Delta K = \frac{M_L}{(dM/dK)}$$  \hspace{1cm} (A.14)

where $dM/dK$ is the slope of the moment-curvature relationship in the elastic region that is given by the following equation.

$$\frac{dM}{dK} = E \cdot I$$  \hspace{1cm} (A.15)

and $I$ is the moment of inertia of the tube $(I = \pi^3 t_0^3)$. By substituting Eq. A.13 and Eq.A.14 into Eq.A.15, the final bend angle $\theta_U$ after the occurrence of springback is calculated with the equation below.

$$\theta_U = \theta_L \left(1 - \frac{R_L M_L}{EI}\right)$$  \hspace{1cm} (A.16)
Process Modeling of Freeform Incremental Forming Using LS-DYNA®

Feng Ren, Zhen Cui, and Z. Cedric Xia
Ford Motor Company

Todd Slavik, Li Zhang and Xinhai Zhu
Livermore Software Technology Corporation

Abstract

Incremental Sheet Forming (ISF) is a manufacturing process for sheet metal prototyping where the blank is incrementally deformed into a desired shape by one or more stylus tools traveling along a prescribed path. Conventional ISF can be categorized into two types, Single-Point Incremental Forming (SPIF) where the sheet metal is formed from one side by a single stylus tool; and Double-Point Incremental Forming (DPIF) where a die positioned underneath a stylus tool pushes the sheet metal to wrap around the die. More recently a Freeform Incremental Forming (FIF) is developed at Ford Motor Company where two stylus tools synchronized in motion and deform the sheet metal from opposite sides as they are traveling to form a product shape. The new technology provides significant advantages for sheet metal fabrication process in terms of cost and flexibility because forming dies are completely eliminated and complex geometries can be formed. However the uniqueness of the process also brings significant challenges to its process design. This paper presents new capabilities developed in LS-DYNA for simulating Freeform Incremental Forming (FIF). The rigid stylus tools can move arbitrarily in both translational and rotational Degrees-of-Freedom (DOF). Challenges for numerical simulations and their modeling techniques are addressed in the paper. Numerical and experimental examples of Freeform Incremental forming processes are presented. It is demonstrated that the simulation results correlates very well with laboratory measurements.

Introduction

Incremental Sheet Forming (ISF) is a manufacturing process for sheet metal prototyping and small volume production where the blank is incrementally deformed into a desired shape by one or more stylus tools traveling along a prescribed path (see [1] for an excellent review). Conventional ISF can be categorized into two types, Single-Point Incremental Forming (SPIF) where the sheet metal is formed from one side by a single stylus tool; and Double-Point Incremental Forming (DPIF) where a die positioned underneath a stylus tool pushes the sheet metal to wrap around the die. More recently a Freeform Incremental Forming (FIF) is developed at Ford Motor Company where two stylus tools are synchronized in motion and deform the sheet metal from opposite sides as they are traveling to form a product shape. The new technology provides significant advantages for sheet metal fabrication process in terms of cost and flexibility because forming dies are completely eliminated and complex geometries can be formed. In general small hemispherical stylus tools are used to locally deform the sheet instead of the conventional punch. The sheet metal is clamped along its periphery on a platform to prevent any metal slippage or draw-in. A pre-determined tool path program feeds into a Numerically-Controlled (NC) machine or other robotic arms which controls the movement of stylus tools. Due to the process's flexibility, low-cost and short lead-time, it has been the focus of extensive
research over the years. Major research interests included the understanding and development of incremental forming mechanics; failure prediction and formability assessment; and the development of more efficient manufacturing process.

More recently, finite element simulations have been used to gain understanding of the process and guide further technology development [2-8]. However in the incremental forming, the plastic deformation is highly localized to the tool/sheet metal contact area; and the total length of the prescribed tool paths can be as long as several kilometers, depending on the size and complexity of the part, the required step size to achieved desired surface quality, and whether multi-stage forming is needed or not. These particular challenges demand sophisticated modeling techniques and numerical treatments, and LS-DYNA® is well suited for such tasks.

This paper presents some of the new features developed in LS-DYNA for the process modeling of Freeform Incremental Forming (FIF). In particular, the development of tool motions capable of all six Degrees-of-Freedom (DOF) is a critical enabler for the simulation of FIF process. Simulation results are also presented for the investigation of path synchronizations in FIF and use it to guide further process development.

**LS-DYNA Implementation**

A new keyword *BOUNDARY_PRESCRIBED_ORIENTATION_RIGID_VECTOR is implemented in LS-DYNA to describe the rigid body rotations of the stylus tools in Freeform Incremental Forming. The parameters are the rigid part ID and its motion curves as specified by the three components of its orientation vector

\[ \vec{V} = (V_x, V_y, V_z) \]  

(1)

The motion curves are defined through *DEFINE_CURVE. Three separate curves have to be defined for \( V_x, V_y, V_z \) individually.

It's customary to normalize \( \vec{V} \) as a unit vector with

\[ |\vec{V}| = \sqrt{V_x^2 + V_y^2 + V_z^2} = 1 \]  

(2)

LS-DYNA will internally normalize it if it is not already done so in the specified curves.

The keyword *BOUNDARY_PRESCRIBED_ORIENTATION_RIGID_VECTOR must be used in conjunction with the keyword *PART_INERTIA where a nodal point or the X-Y-Z coordinate of a point has to be specified when defining a rigid body. It serves as the reference point for the specified rigid body rotations.

At first thought, it would seem that implicit solution techniques are ideal for the modeling of incremental forming. The physical process usually takes half an hour to a couple of hours for a reasonably-sized panel with moderately complex geometry, and the deformation is essentially quasi-static. However upon closer examination, it is recognized that the required time increment is also dictated by the geometric features to be formed, and the tool paths are highly non-linear when geometric features are present. A larger time step would not be able to capture the exact paths the tools. Consequently the metal would not deform in the same way as in physical process. This consideration greatly diminishes the advantages of implicit methods where a relatively larger time increment can be adopted and thus speed up computational time.
Both implicit and explicit solution techniques are investigated during the course of the work. It is concluded that explicit solutions are more effective for the modeling of Freeform Incremental Forming. The results presented in this paper are all conducted with explicit method.

Both adaptive meshing and uniform fine meshing are investigated. No major differences are found as long as the final mesh quality is comparable. However it is noticed that, in the case of stylus tools with smaller diameters, the adaptive meshing techniques might not be able to refine the meshes well ahead in time for contact deformation. Some adjustments have to be made for the parameters in the keyword *CONTROL_ADAPTIVE.

Selective Mass Scaling technique is used to reduce computational time while minimizing overall dynamic effects.

The contact between the rigid stylus tools and the sheet metal is defined by the keyword *CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE. The sheet metal is modeled with Hughes-Liu shell element (Element Type 16) and the material modeled as elasto-plastic with in-plane transverse isotropy. This is consistent with conventional sheet metal forming modeling practices, and is defined by MAT37 in LS-DYNA through *MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC.

The periphery of the sheet metal is constrained on a frame in order to prevent any slippage or draw-in. It is thus not unreasonable to fix the nodal movement of these nodes to zero through the keyword *BOUNDARY_SPC_NODE.

Since the deformation is highly localized between tool and sheet metal contact area, and sometimes tools with smaller diameters such as 6mm are used, it is prudent to conduct a study where the sheet metal is modeled as solid elements and the simulation results compared to those obtained from shell element models. For solid elements, the material card is defined with *MAT_ANISOTROPIC_PLASTIC.

**Simulations and Results**

The simulations of using four different methods to incrementally form a truncated axisymmetric cone are investigated in this paper. The design intent of the truncated cone has a base radius of 40mm, with a wall inclination angle of 50°. It is truncated at a height of 40mm. The four forming methods include:

(a). Single Point Incremental Forming (SPIF). This is the baseline for the study;

(b). Freeform Incremental Forming – Strategy 1 (FIF-1), where two stylus tools both with semi-spherical tips incrementally form the part from opposite sides of the sheet metal;

(c). Freeform Incremental Forming – Strategy 2 (FIF-2), where two stylus tools both with semi-spherical tips, with one incrementally forming the sheet metal and the other supporting the sheet metal along the top;

(d). Freeform Incremental Forming – Strategy 3 (FIF-3), where two stylus tools, one with a semi-spherical tip as the forming tool, and the other with a flat tip as the supporting tool.

A sketch of the four processes are illustrated as inserts in Figure 1.
The emphasis of this study is to compare the geometric accuracies using different incremental forming methods. In particular, three Freeform Incremental Forming strategies are simulated here (FIF-1, FIF-2, and FIF-3). The final geometries from all four methods are presented in Fig. 1. It is evident from the simulation results that significant geometric deviations from the design intent are at the top of the cone where the flat sheet metal transitions to the cone geometry. Since a flat sheet metal has less stiffness than a part with more shape in it, the transitioning area is relatively weak. As a consequence, the SPIF and FIF-1 processes where the formed part was not supported at the area with the other tool tended to give the biggest deviations, while FIF-2 and FIF-3 maintained the design intent as expected.

The simulation results for the four forming strategies have been confirmed by experimental studies. They provided a useful tool to guide process design and strategies for making incrementally formed parts.

Discussions

Numerical simulations of incremental forming processes have seen great progress over the past a couple of years. Among commercially available software tools, LS-DYNA has been proven to be capable and effective. New capabilities are also being incorporated as needed. However, some issues still remain and new development and improvement are required. In particular,

(1). The computational resources required are very intensive for the simulations of incremental forming process. This is partly intrinsic to the process itself since the deformation is highly localized, and the tool travel paths are often very long. Recent research on a local-global approach of simulations should be an effective technique and is promising [9].

(2). There are questions as whether the use of shell element in the simulation is adequate or not if the tool radius is relatively small. Preliminary studies by the authors indicate that it might not be too serious an issue for the prediction of general deformation or springback. However it might not be sufficient for forming limit prediction since the failure usually caused by fracture, not localized necking in conventional sheet metal forming [10-13]. As a result, the stress triaxiality becomes very important, and can only be adequately captured by solid element simulations.

(3). Freeform Incremental Forming is still at its infant stage. If the process simulations are to be applied for full production panel sizes such as hoods, doors and decklids, the total tool path lengths could be in tens or even hundreds of kilometers. This requires tremendous computing
powers and might not be able to meet production requirements for sheet metal rapid prototyping. The development of new simulation technologies is essential for future production applications.

References
Numerical Simulation and Experimental Study of Electromagnetic Forming

Jianhui Shang\textsuperscript{1}, Pierre L'Eplattenier\textsuperscript{2}, Larry Wilkerson\textsuperscript{1}, Steve Hatkevich\textsuperscript{1}

\textsuperscript{1} American Trim LLC, Lima, OH 45901 USA
\textsuperscript{2} Livermore Software Technology Corporation, Livermore, CA 94551 USA

Abstract

Compared to traditional sheet metal forming, electromagnetic forming (EMF) has several advantages, such as increased formability, cost savings and improved flexibility. There are many EMF applications in sheet metal forming, especially for aluminum alloys, because aluminum alloys have relatively low formability and high conductivity when compared to steel. The EMF process uses magnetic field generated by a conductive actuator upon large capacitor discharge to accelerate workpiece to high velocity. It is a complex coupled mechanical-thermal-electromagnetic phenomenon, which makes it difficult to numerically simulate. However, to save time and cost, numerical simulation is needed to accurately predict results of EMF.

The Electromagnetism (EM) module of LS-DYNA\textsuperscript{®} has been developed by LSTC, which can be used for numerical simulation of EMF. American Trim has applied this module to assist in its EMF designs. In this paper, to access the capability of EM module, the numerical and experimental results of sheet metal formed with EMF were compared. The experiment was to apply EMF for straight-edge flanging of Al 6061-T6 sheet. Then this flanging process was modeled using both SMP and MPP version of LS-DYNA EM module. The comparison between the final shapes of flanged samples and the numerical simulation showed the good correlation between experimental and numerical results, which indicates the good predictive ability of the LS-DYNA EM module for EMF.

Introduction

Electromagnetic forming (EMF) is characterized by very high forming velocity and deformation strain rates, which makes it fundamentally different from traditional sheet metal forming. At sufficiently high velocities and/or strain rates, stretching limits are not bounded by the restrictions of a traditional Forming Limit Diagram (FLD). Instead, ductility far beyond typical quasi-static forming limits can be achieved [1-3]. Therefore, EMF has the potential to improve material formability, and thus also to expand the range of candidate materials for a particular application. In addition, only single-sided tooling is required in EMF, as opposed to a precisely machined matched punch and die set in conventional stamping. Single-sided tooling reduces tooling cost and complexity of forming systems, and also eliminates tedious alignment requirement in conventional stamping.

Above advantages of EMF makes it attractive to form metal sheets, especially aluminum alloys due to their low formability and high electrical conductivity. But it is difficult to design and predict EMF process due to its complexity. During EMF, a large electric current pulse passes through a conductive coil by discharging a capacitor bank. The current pulse produces a transient magnetic field around the coil that induces eddy currents in a nearby metal workpiece. The mutually repulsive force between the stationary coil and the metal workpiece causes the metal workpiece to be accelerated toward, and to impact upon, a nearby die surface at very high
velocity. Figure 1 is a schematic diagram of the basic EMF. It is impossible to accurately predict this complex process by analytical calculations due to the transient electromagnetic forces, large deformation and high strain rate.

To save time and cost, numerical simulation is highly needed to accurately predict the results of EMF and help the design of the process. EMF is a coupled mechanical-thermal-electromagnetic phenomenon, which means the electromagnetic, thermal and mechanical fields should be solved simultaneously. This is difficult to achieve for the commercially available codes.

Recently Livermore Software Technology Corporation (LSTC) has developed LS-DYNA® electromagnetism module, which can be applied to solve the 3D coupled mechanical-thermal-electromagnetic process. To evaluate the capability of this electromagnetism module, a simple experiment was carried out to use EMF to flange Al 6061-T6 sheet. And the LS-DYNA electromagnetism module was applied to simulate the flanging process. This paper will present the experimental procedure, numerical simulation and the comparison between the experimental and numerical simulation results.

**Experimental Procedure and Results**

Flanging is a forming operation to bend a narrow strip at the edge of a metal sheet along a straight or curve line. It is relatively simple to carry out and simulate, comparing to other sheet metal forming processes. And the coupled mechanical-thermal-electromagnetic phenomenon still occurs in this process, when EMF is applied to bend metal sheets. In this paper, a straight-edge flanging of aluminum alloy sheet was carried out for experimental and numerical simulation study.

The capacitor bank used in this study was 18 kJ Elmag machine. And the single-turn coil was made of Cu and connected to the capacitor bank to generate electromagnetic forces for flanging, shown in Figure 2. The coil was covered with Kapton tape for insulation purpose. The material...
used was Al 6061-T6 with 0.8mm thickness. The size of the Al samples was 50mm x 50mm. During the flanging, the Al sheet was positioned between the coil and a solid block made of G10 Garolite. The G10 block was clamped by a hydraulic press to hold the rest area of the Al sheet. The flanging height was 10mm. Figure 3 shows the experiment setup.

![Figure 2. Photo of the single-turn coil used in experiments](image)

![Figure 3. Photo of experiment setup](image)

In this study, different energy inputs were applied to bend the Al sheets. It was found that the Al sheet was bent to 90 degree by 3.6 kJ energy input and 64.5 degree by 2.7 kJ energy input. In the 2.7 kJ case, a Rogowski coil was used to measure the electric current in the single-turn coil. Figure 4 shows the flanged Al sheet in 2.7 kJ case and Figure 5 shows the measured current trace in the coil.
The numerical simulation was performed using the LS-DYNA electromagnetism module available in the “beta” 980 version. In this module, Finite Element Method (FEM) is coupled with Boundary Element Method (BEM) to compute magnetic field, electric field and induced current by solving Maxwell equations in eddy-current approximation. FEM is applied to solve Maxwell equations for the solid conductors and BEM is used for the surrounding air. The detailed introduction of this module can be found in other paper [4].
The 3D simulation model of the flanging was built for LS-DYNA, shown in Figure 6. There are three parts: the Cu single-turn coil, the Al 6061-T6 sheet and the G10 holder. The coil and the Al sheet were modeled using eight node hexagonal solid elements, which are required for the solid conductors in electromagnetism module. The area of the Al sheet under the G10 holder had the coarser meshing than the flanging area, since this area didn’t have plastic deformation at all during the flanging and coarser meshing could save computing time. The G10 holder was modeled as rigid body with shell elements since G10 Garolite didn’t have plastic deformation during the flanging and is nonconductive material.

Since this flanging process involves high strain rate and large deformation, Cu and Al 6061-T6 were modeled using Johnson-cook strength model, which has the following form [5]:

$$\sigma = (A + B\varepsilon^n)(1 + C \ln \dot{\varepsilon})[1 - \left(\frac{T - T_{room}}{T_m - T_{room}}\right)^m]$$

Table 1 shows the Johnson-cook strength model parameters for Al 6061-T6 and Cu. The G10 Garolite was modeled as elastic material. Table 2 lists the material properties of Al 6061-T6 and Cu. The measured current trace in Figure 5 was set as the input for the simulation. Due to the long computing time, the ending time of the simulation was set as 186µs to save time.

Table 1. Johnson-cook strength model parameters [6, 7]

<table>
<thead>
<tr>
<th>Material</th>
<th>A (MPa)</th>
<th>B (MPa)</th>
<th>C</th>
<th>n</th>
<th>m</th>
<th>Tm (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Al 6061-T6</td>
<td>324</td>
<td>114</td>
<td>0.002</td>
<td>0.42</td>
<td>1.34</td>
<td>925</td>
</tr>
<tr>
<td>Cu</td>
<td>90</td>
<td>292</td>
<td>0.025</td>
<td>0.31</td>
<td>1.09</td>
<td>1356</td>
</tr>
</tbody>
</table>
Table 2. Material properties of Al 6061-T6 and Cu [8]

<table>
<thead>
<tr>
<th>Property</th>
<th>Al 6061-T6</th>
<th>Cu</th>
</tr>
</thead>
<tbody>
<tr>
<td>Electrical Conductivity ((10^6 / \Omega \text{ mm}))</td>
<td>0.025</td>
<td>0.058</td>
</tr>
<tr>
<td>Mass Density (g/mm(^3))</td>
<td>0.0027</td>
<td>0.00894</td>
</tr>
<tr>
<td>Young’s Modulus (GPa)</td>
<td>68.9</td>
<td>115</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.33</td>
<td>0.31</td>
</tr>
<tr>
<td>Heat Capacity (J/g °C)</td>
<td>0.896</td>
<td>0.385</td>
</tr>
<tr>
<td>Thermal Conductivity (W/m K)</td>
<td>167</td>
<td>391</td>
</tr>
</tbody>
</table>

**Results and Discussion**

The simulation results were interpreted by LS-PREPOST. Figure 7 shows the comparison between the final shapes of the flanged sample and the simulation result in the 2.7 kJ case. For the flanged sample, the Al sheet was bent to 64.5 degree. The simulation result showed that the Al sheet was bent to 69.4 degree. There is 7.6% difference between the simulation and the experiment, which indicates the good correlation between experimental and numerical simulation results.

Figure 7. Comparison of the experimental and numerical simulation results for 2.7 kJ case (Left: simulation—bent angle: 69.4 degree; Right: experiment—bent angle: 64.5 degree)

Figure 8 shows the left edge of the Al sheet at the different time steps in the simulation, which indicates that the straight strip of the Al sheet was not accelerated to move at the same time. At the early stage, the middle area of the straight strip moved first and the two edges followed the middle area. But at the late stage, the middle area rebounded back and the two edges continued to move forward. Because of the continuousness of the metal strip, the middle area and the two edges rebounded back and forth several times and then settled down at last. This phenomenon makes sense because the induced current densities were different at different areas of the Al sheet and then the resulted repulsive Lorenz forces were also different. Figure 9 is the vector of the induced current density in the Al sheet at the time of 23µs. Figure 10 is the vector of the Lorenz force in the Al sheet at the time of 23µs. These two figures clearly agree with the above statement.
Figure 8. Left edge image of the Al sheet at the different time steps

- t = 0 µs
- t = 37 µs
- t = 74 µs
- t = 111 µs
- t = 148 µs
- t = 186 µs

Figure 9. Vector of induced current density in Al sheet at the time of 23µs (top view)
SMP and MPP Version

The above simulation was carried out using the Shared Memory Parallelism (SMP) version of LS-DYNA electromagnetism module. Recently, a Massively Parallel Processing (MPP) version of the electromagnetism module was developed, allowing sharing the CPU and memory between different processors and thus faster computations on larger problems. To test the capability of the MPP version, the same simulation was run again using the MPP version. Both of the simulations were performed using a computer with two quad-core Intel Xeon processors (3.00 Ghz/1333MHz). And the computer has 16GB of RAM. The simulation using SMP version was carried out on a single CPU and the simulation using MPP version was performed on four CPUs. The simulation results show that the MPP version dramatically reduced the elapsed time by 63%, comparing to the SMP version. Table 3 is the comparison between these two versions for above simulation.

<table>
<thead>
<tr>
<th>Version</th>
<th>CPU number</th>
<th>Elapsed time (second)</th>
</tr>
</thead>
<tbody>
<tr>
<td>SMP</td>
<td>1</td>
<td>367774</td>
</tr>
<tr>
<td>MPP</td>
<td>4</td>
<td>136040</td>
</tr>
</tbody>
</table>

Conclusion

The straight-edge flanging of Al 6061-T6 sheet by EMF was studied by experiments and LS-DYNA numerical simulations in this paper. The comparison between the final shapes of the flanged sample and the numerical simulation shows the good prediction capability of the LS-DYNA electromagnetism module for EMF. It is a good tool to understand and design the EMF process.
Both SMP and MPP version of LS-DYNA electromagnetism module predicted EMF process reasonably well. And MPP version can reduce the elapsed time dramatically.

Acknowledgement

The authors thank LSTC for collaboration and software support.

References


