Finite Elements in Analysis and Design 79 (2014) 1–8

Contents lists available at ScienceDirect

Finite Elements in Analysis and Design

journal homepage: www.elsevier.com/locate/finel

Development of an inverse finite element method with an initial guess of linear unfolding

M. Kankarani Farahani, M. Bostan Shirin, A. Assempour *

Center of Excellence in Design, Robotics and Automation, Department of Mechanical Engineering, Sharif University of Technology, Azadi Avenue, P.O. Box 11365-9567, Tehran, Iran

ARTICLE INFO

Article history:
Received 6 May 2013
Received in revised form
2 October 2013
Accepted 7 October 2013
Available online 9 November 2013

Keywords:
Sheet metal forming
Initial blank design
Finite element method
Linear unfolding

ABSTRACT

An inverse finite element method (IFEM) has been developed for estimation of the blank size and prediction of the strain distribution in sheet metal forming. In the inverse method the nodal coordinates in the final shape are known and their corresponding positions on the initial blank should be determined. The developed method deals with logarithmic large strains of membrane triangular elements, virtual work principle and a new approach for friction modeling. This method leads to a system of nonlinear equations which is highly sensitive to the initial guess. In order to avoid the converging problems, especially in the quasi-vertical walls, an appropriate initial guess is introduced. The introduced initial guess guarantees the convergence; furthermore the number of iterations in the nonlinear numerical solution is decreased and the solution speed is significantly increased. Three different problems are analyzed with the developed method and the results show good agreement to commercially available finite element software and experimental results.

1. Introduction

Sheet metal forming is a complicated process that depends on numerous parameters such as die geometry, blank shape, drawing ratio and blank holder force. An important factor for having a successful process without defects is the initial blank shape. The traditional method to simulate the sheet metal forming is the forward incremental finite element method [1–3]. In the forward method, computations start with the blank geometry which is unknown at the initial design stage. Therefore, trial and error is the nature of the forward method which is very time consuming. Due to this fact, a method is needed to predict the initial blank shape directly. So many attempts have been carried out in order to obtain the initial blank directly.

Karima [4] made use of the slip-line method to design the initial blank shape. Vogel and Lee [5] and Chen and Sowerby [6] used the characteristic of plane stress, while Blount and Stevens [7] used geometric mapping to design the initial blank shape. These methods provide good guidance to design the initial blank shape, but they neglect the height of the deformed parts, have geometric restrictions, and do not consider the deformation behavior of the materials. On the other hand, there have been several attempts to design the blank shape and estimate the distribution of the strain in a deformed part with deformation theory of plasticity. Majlessi and Lee [8–11] showed that using this theory is reasonable for rapid simulation in the first stages of design. They extended the theory of Levy et al. [11] and applied it to axisymmetric one step and multi-stage problems, obtaining good results. However, this method cannot be applied without boundary conditions implementation like friction and blank holder force. Therefore, the crash form process cannot be analyzed by this approach. Guo and Batoz [12–14] derived a formulation for field problems as an inverse method to obtain the initial blank shape and the thickness distribution in a deformed part. Due to the nonlinearities that come from considering large deformation and material properties, this method is very sensitive to initial guess and needs an efficient initial guess. Based on the work by Karima et al. [15], Assempour et al. [16–18] developed a linear inverse finite element method known as unfolding technique. Their formulation is based on the principle of potential energy minimization and linear strain–displacement relation. Thus, their formulation leads to a system of linear equations which can be solved very quickly. This method is very efficient in obtaining the initial blank; however, due to the nature of linear formulation, the strain values are less accurate compared with nonlinear formulation.

In the present study, an inverse finite element formulation has been developed which deals with the principle of virtual work, deformation theory of plasticity and logarithmic large strains of membrane triangular elements. This method leads to a system of nonlinear equations which is very sensitive to the initial guess and may diverge in numerical solution. In order to avoid the

* Corresponding author. Tel.: +98 21 6616 5529.
E-mail address: assem@sharif.edu (A. Assempour).

0168-874X/$ - see front matter © 2013 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.finel.2013.10.002

convergence problems, especially in the quasi-vertical walls, the linear unfolding method is used as an appropriate initial guess. Besides, a new approach for friction modeling is introduced which considers a non-uniform distribution of the friction force on the nodes located under the blank holder zone. Numerical simulations have been conducted and results have been compared with Abaqus Explicit and experimental results to verify its feasibility.

2. Modeling and formulations of the developed inverse algorithm

In the inverse approach only two states are considered:

(1) The initial flat blank obtained from the initial guess with a known thickness.
(2) The final work piece with a known tridimensional mid-surface discretized with flat triangular membrane elements.

2.1. Basic formulations

The geometry of final shape is discretized with linear triangular membrane elements. Positions of elements in the flat plane are obtained by an appropriate initial guess. Fig. 1 shows the schematic elements of the initial and final shapes.

By considering only the initial and final states of elements and calculating left Cauchy–Green deformation tensor $[B]$, principal stretches are obtained as below:

$$\lambda_i^{-2} = \{n_i\}^T [B]^{-1} \{n_i\} \quad (i=1,2) \tag{1}$$

where $\{n_i\}$ are eigenvectors of $[B]^{-1}$ [19]. Principal stretches which are also eigenvalues of $[B]$, can be calculated as follows:

$$\begin{cases} \lambda_1 = ((1/2)(B_{11}+B_{22})+(1/2)((B_{11}-B_{22})^2+4B_{12}^2)^{1/2})^{-1/2} \\ \lambda_2 = ((1/2)(B_{11}+B_{22})-(1/2)((B_{11}-B_{22})^2+4B_{12}^2)^{1/2})^{-1/2} \end{cases} \tag{2}$$

λ_3 is also obtained from λ_1 and λ_2 by assuming incompressibility.

B_{ij}; $i,j=1,2$ are components of $[B]^{-1}$ [19]:

$$\begin{Bmatrix} B_{11} \\ B_{22} \\ B_{12} \end{Bmatrix} = \frac{1}{(h_{2y}h_{3x}-h_{2x}h_{3y})^2} \times \begin{Bmatrix} h_{3y}^2 L_2 + h_{2y}^2 L_3 - (L_2+L_3-L_1)h_{2y}h_{3y} \\ h_{3x}^2 L_2 + h_{2x}^2 L_3 - (L_2+L_3-L_1)h_{2x}h_{3x} \\ -h_{3x}h_{3y}L_2 - h_{2x}h_{2y}L_3 + (1/2)(L_2+L_3-L_1)(h_{2y}h_{3x}+h_{2x}h_{3y}) \end{Bmatrix} \tag{3}$$

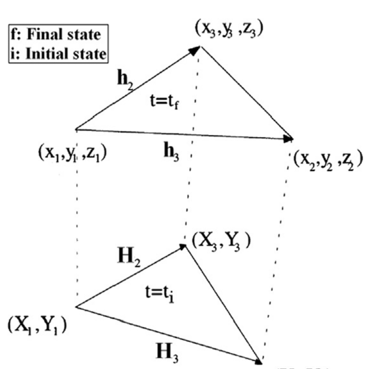

Fig. 1. Kinematics of linear triangular membrane elements between initial and final shapes.

where $L_i = \|H_i\|^2$ and h_{ix}, h_{iy} are the components of h_i (see Fig. 1) in the element local coordinates of final configuration.

Logarithmic strains in element local coordinates can be obtained from Eq. (2) as follows:

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_{xy} \end{Bmatrix} = \begin{Bmatrix} \ln\lambda_1 \cos^2\theta + \ln\lambda_2 \sin^2\theta \\ \ln\lambda_1 \sin^2\theta + \ln\lambda_2 \cos^2\theta \\ (\ln\lambda_1 - \ln\lambda_2)\sin\theta\cos\theta \end{Bmatrix} \tag{4}$$

where θ is the angle between the x axis of the local coordinate system and the principal stretch λ_1 [19]:

$$\theta = \tan^{-1}\left(\frac{\lambda_1^{-2} - B_{11}}{B_{12}}\right) \tag{5}$$

Under conditions of plane stress ($\sigma_z = \sigma_{xz} = \sigma_{yz} = 0$) and planar isotropic material, Hill's function gives

$$f(\sigma_{ij}) = \sigma_{xx}^2 + \sigma_{yy}^2 - \frac{2r}{1+r}\sigma_{xx}\sigma_{yy} + 2\frac{1+2r}{1+r}\sigma_{xy}^2 \tag{6}$$

where r is the Lankford value.

Based on deformation theory of plasticity and implementing flow rule on Hill's function, the constitutive equation is defined as

$$\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix} = \frac{2}{3}\frac{\bar{\sigma}}{\bar{\varepsilon}} \begin{bmatrix} AB & A & 0 \\ A & AC & 0 \\ 0 & 0 & AD \end{bmatrix} \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{xy} \end{Bmatrix} \tag{7}$$

where,

$$A = \frac{r(2+r)}{(1+2r)}, \quad B = \frac{(1+r)}{r}, \quad C = \frac{(1+r)}{r}, \quad D = \frac{1}{r}, \tag{8}$$

The effective strain corresponding to Eq. (6) can be expressed as

$$\bar{\varepsilon} = \sqrt{(2/3)A(B\varepsilon_{xx}^2 + 2\varepsilon_{xx}\varepsilon_{yy} + C\varepsilon_{yy}^2 + 2D\varepsilon_{xy}^2)} \tag{9}$$

The effective stress $\bar{\sigma}$ is also obtained from $\bar{\varepsilon}$ by the Holloman power law.

2.2. Solution procedure

The principle of virtual work is applied to the final work piece:

$$\sum_e W_{int}^e - \sum_e W_{ext}^e = 0 \tag{10}$$

where e is element numbers and,

$$W_{int}^e = \int_{v_e} \langle \varepsilon^* \rangle \{\sigma\} dv \tag{11}$$

$$W_{ext}^e = \int_{v_e} \langle u^* \rangle \{f\} dv \tag{12}$$

where, $\langle u^* \rangle$ and $\langle \varepsilon^* \rangle$ are the virtual displacements and their corresponding virtual membrane strains, v_e is the element volume, $\{\sigma\}$ are the Cauchy stresses, and $\{f\}$ are the external forces such as tool actions in element local coordinates.

Element internal work W_{int}^e can be written as below:

$$W_{int}^e = \langle U_n^* \rangle \{F_{int}^e\} \tag{13}$$

With $\langle U_n^* \rangle = \langle U_j^*, V_j^*, W_j^* \rangle$, $j=1,2,3$. U^*, V^*, W^* are the virtual displacements in the global coordinate system XYZ. Since the vertical displacement of each node is known, $W_j^* = 0$. $\{F_{int}^e\}$ are the element internal forces vector and are given by

$$\{F_{int}^e\} = [T]^T [\mathbf{B}]^T \{\sigma\} Ah \tag{14}$$

where $[\mathbf{B}]$ is the strain operator in the local coordinate system of each element. $[T]$ is the transformation matrix between global and local coordinate systems. A is the element area and h is the thickness of the element. h is obtained from thickness stretch which is modified at each iteration.