Finite Elements in Analysis and Design 46 (2010) 992–1000

Contents lists available at ScienceDirect

Finite Elements in Analysis and Design

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

Stamping process design using FEA in conjunction with orthogonal regression

Mehmet Firat*, Osman H. Mete, Umit Kocabicak, Murat Ozsoy

The University of Sakarya, Department of Mechanical Engineering, 54187 Adapazari, Turkey

ARTICLE INFO

Article history:
Received 27 July 2008
Received in revised form
7 July 2010
Accepted 8 July 2010
Available online 4 August 2010

Keywords:
Sheet metal forming
Stamping
Springback
Formability
Response surface

ABSTRACT

A process design technique is presented for the formability assessment of sheet metal stamping parts and feasibility analysis of process conditions. The proposed approach is based on numerical simulation of stamping processes by using explicit – incremental and implicit – iterative finite element techniques. The influence of the numeric model parameters are investigated with factor analysis and described with response surfaces obtained by multi-linear regression. A forming process leading to springback-critic channel geometry is selected for the application of the proposed methodology. The effects of modeling parameters are determined by evaluating influences of the punch velocity and the element size, in order to obtain a numerically calibrated simulation model. Then the sensitivity of the springback deformations to the contact interface friction and the blankholder force is predicted, and a set of response surfaces is generated. Comparisons with the experimental data indicated the suitability of the proposed approach in springback predictions. The proposed technique is employed in the stamping analysis of an engine suspension bracket made of high-strength steel. The process conditions are investigated in terms of drawbead penetration and blankholder force setting, and the predicted part shapes are compared with CMM measurements of the manufactured parts. An evaluation of computed springback distortions shows a good correlation with experiment results and confirms the use of process parameters estimated with the proposed design.

1. Introduction

Continuously reducing lead-times and development expenses of automotive sheet metal components put more rigorous restrictions on the design practice and manufacturing costs of stamping tools and dies. The use of computer-aided design and analysis tools is known to help reaching objectives more feasibly, and computer simulations of forming processes based on finite element analysis (FEA) have found wide-spread use in the stamping industry [1]. At present, besides calculating the formability of the sheet material part, methods engineers are now expected to estimate the part geometry more accurately enforcing the progress in the accuracy of finite element (FE) simulations both qualitatively and quantitatively particularly for complex stamping geometries [2,3]. The growing requirements for cost-effective process design practices, especially for expensive lightweight materials such as advanced high-strength steels, necessitate the precise calculation of the sheet metal deformations, including springback in order to improve the industrial process [4].

It is known that the numerical analyses of forming processes based on the FE procedures produce approximate solutions that show dependence on the modeling techniques and discretization algorithms in time and space [4–6]. Considering the nonlinear material behavior during a forming process, the stress and strain state at a material point is dependent on both the instant of forming process and the history of the deformation, and therefore, is strongly a function of the time frame characterizing process [6]. Accordingly, in a sheet metal forming analysis including the springback step, the description and the time variation of the tooling loads are important modeling decisions that may have a considerable influence on the final simulation outcome [4–7]. The blank–tool interaction defined by the contact–friction constitutive model is also subject to the modeling effects, due to the mesh density and layout used for both the sheet blank and the tool [5,6]. Considering the contact modeling based on the penalty-factor method, the interface friction is mostly described with the bilinear Coulomb model that uses the static and dynamic coefficient of friction as well as other factors such as the damping ratio and the stiffness factor. In view of an explicit FEM, previous studies have indicated the importance of the interaction between the element size and mesh layout of deformable blank and the rigid contact segments [8–10]. Therefore, the description of tool forces and modeling approach followed in the FE mesh

* Corresponding author: Tel.: +90 264 295 5451; fax: +90 264 295 5450.
E-mail addresses: firat@sakarya.edu.tr, testfem@yahoo.com (M. Firat).

0168-874X/$ - see front matter © 2010 Elsevier B.V. All rights reserved.
doi:10.1016/j.finel.2010.07.005

discretization for both the deformable blank and rigid tooling surfaces may influence the simulation results in terms of the sheet formability and the springback shape.

In the last decade, die-makers have adapted to the process simulation early in the stamping design and methods engineering phase of sheet metal parts. There are, however, several design variables to be considered in the forming process and part feasibility studies and therefore, many computer iterations are usually required to reach an optimum forming process conditions, while satisfying the necessary stamping quality [1,5,8]. This trial-and-evaluate approach may be time-consuming and considerably costly, and reaching a successful forming process still depends on the experience of the methods engineer, especially for complicated part shapes difficult to stamping. In order to reduce the design and analysis costs, FE analyses are now employed in conjunction with response surface methods (RSM) [11–17]. In this approach, design variables related to tool geometry or process parameters are firstly chosen and their allowable variations are also prescribed. A design space is thereby generated and a set of data points are sampled from this design space by using the design of experiment (DOE) methods, such as Latin hypercube design, Taguchi methods and orthogonal arrays [18]. Furthermore, the design objectives are defined according to the process requirements, and the minimum blank thickness or amount of springback is generally chosen in the assessment of design points [12,14,15,17]. Then FE analyses are conducted for all parameter combinations, and the objectives are determined for each data point in the design space. Finally, a response surface for each design objective is constructed by fitting a polynomial function from the set of FEA results. Consequently, a surrogate model is obtained and the need for an expensive FEA for each process conditions is eliminated efficiently.

In this paper, a finite element technique is presented for the formability assessment of sheet metal stamping parts and feasibility analysis of process conditions. The proposed approach is based on an FE simulation of stamping processes by using explicit – incremental and implicit – iterative techniques. The influence of the FE model parameters are investigated by means of a factor analysis. The effects of process conditions on the springback distortions of stamping parts are described with RSM obtained by multi-linear regression analysis. Firstly, the part shape variations due to the FE modeling approach are investigated using a channel forming process characterized by simultaneous drawing and bending type deformations. The channel geometry is described by three geometric parameters that indicate the shape distortions due to the springback for this particular stamping form. The effects of the numerical parameters are determined based on the predicted shape parameters at three different levels of punch velocity and element size. The sensitivity of the springback deformations to the contact interface friction and the blankholder force is predicted using the previously calibrated FE model. A set of response surfaces is generated based on a multi-dimensional regression technique describing the interaction between process parameters. Finally, the proposed approach is employed in the process evaluation and springback prediction of an industrial stamping part. The process conditions are investigated and the predicted part profiles are compared with CMM measurements of the manufactured parts. An evaluation of the results showed the applicability of the proposed methodology in the prediction of the shape distortions of stamping part due to springback.

2. Channel forming process simulation

The 3-D FE simulations are performed for the Numisheet'93 channel forming process using explicit and implicit analysis capability of Ls-Dyna software [19]. The channel forming process investigated in this section is a single-step forming process, in which an initially flat strip of sheet metal is deformed into the shape of a symmetric U-channel (Fig. 1) [20]. The main reasons for the selection of this particular forming application are twofold. Firstly, the U-channel profile geometry is common feature of deep-sectioned stamping parts typically found in many springback-critical automotive structural members. Secondly, the forming process induce a combined bending and stretching mode of sheet metal deformations that may be changed in a controlled manner by the blank restraint force for a consistent springback control.

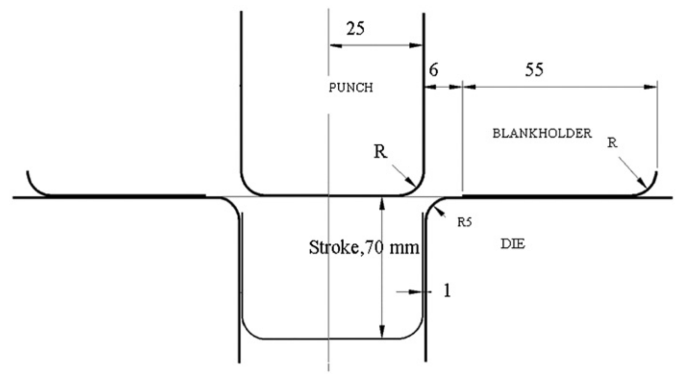

Fig. 1. Numisheet`93 U-channel tooling (mm).

2.1. FE simulation modeling

The blank material used in the channel forming process is an IF mild steel of 0.78 mm nominal thickness [12]. Due to the symmetry of the tooling, loading conditions and the particular placement of the blank, the halves of the tooling and the blank are included in the 3-D FE models together with the appropriate symmetry boundary conditions applied. Additionally only the surface geometries of the tooling and the blank are modeled assuming a rigid die construction and the applicability of the shell type FE element based on plane stress deformation hypothesis. The blank geometry is sheet metal strip of size $300 \times 35\ mm^2$. The material elasticity properties are assumed to be isotropic, and the Barlat–Lian yield function is used to describe the orthotropy of the sheet metal deformations. The elastic modulus and Poisson's constant are given as 207 GPa and 0.3, respectively. The rolling direction is taken as the reference axis and the equivalent stress–plastic strain curve is described with Swift Law.

$$\overline{\sigma}_o(\overline{\varepsilon}^p) = 654(0.00823+\overline{\varepsilon}^p)^{0.17} \tag{1}$$

In the U-channel forming process, the blank is initially clamped between the binder and the die by the application of a constant blankholder force of 19.6 kN. Then the punch moves in the die cavity for a prescribed amount of stroke 70 mm. The applied blankholder force and the punch stroke are the main process parameters in Numisheet'93 U-channel forming experiments [20]. Two sets of FE models are generated for each process condition of the U-channel, including the springback step. Firstly, a process model composed of the blank and the tooling meshes is prepared and an explicit-dynamic incremental FE analysis is conducted for a given set of process parameters. Following the forming analysis, a static implicit incremental-iterative springback analysis is done using the blank mesh only, and the final part geometry and thickness and residual stress distributions are determined. The blank mesh in these FE models is composed of full-integration corotational quadrilateral shell elements with