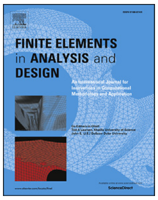
Finite Elements in Analysis and Design 138 (2017) 12–20

Contents lists available at ScienceDirect

Finite Elements in Analysis and Design

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



Finite element simulation of the dynamic behaviour of deep drawn components with accurate thickness description

F. Greco [a,*,1], E. Deckers [a,1], J. Stroobants [b], S. Van Poppel [b], K. Linck [c], W. Desmet [a,1]

[a] KU Leuven, Department of Mechanical Engineering, Division PMA, Celestijnenlaan 300B - Box 2420, B-3001, Leuven, Belgium
[b] Flanders Make, Oude Diestersebaan 133, B-3920, Lommel, Belgium
[c] OCAS NV, Pres. J.F. Kennedylaan 3, B-9060 Zelzate, Belgium

ARTICLE INFO

Keywords:
Deep-drawing
Thinning
Finite elements
Laser scan
Dynamic characterization

ABSTRACT

In this paper we consider the simulation of the dynamic behaviour of deep drawn components by means of the finite element method (FEM). The deep drawing process involves high deformations which result in an irregular thickness distribution in the manufactured component. The thickness variation therefore has to be taken into account in the dynamic FEM simulation, since a reference model based on the initial thickness of the blank material often fails to accurately represent the dynamic properties of the final component. In order to properly account for this variation, two approaches are considered in this paper: the first one consists in simulating the forming procedure with the FEM and subsequently using the resulting geometry for the dynamic analysis. The second approach consists in the generation of an FEM model for the dynamic analysis starting from laser scan measurements on the manufactured component. Both approaches are analysed in detail and are validated with experimental measurements. It is found that the framework based on surface measurements provides slightly more accurate results but the purely numerical procedure is still a useful and faster methodology to obtain engineering accuracy.

1. Introduction

The deep drawing process is nowadays a widely spread metal forming operation with numerous applications in the production of cup shaped objects, cans, shells, pressure vessels and many others. The process consists in deforming a sheet metal, referred to as blank, which is placed on a die cavity and pressed against it using a solid punch.

While the sheet metal attains the shape of the die, the so called thinning and thickening phenomenons occur, which consist in a change of the thickness values along the manufactured component with respect to the original value in the blank [1–3]. In addition, different values of plastic strain are accumulated during the forming process, which result in a residual stress distribution in the final component. These residual stresses have a non-zero resultant in terms of bending moment and cause an elastic shape change of the component known as springback [4–6]. Although this shape change can be negligible in many applications, the increased use of thin high-strength materials for the purpose of weight reduction often increases the springback effect [6], which can become relevant in many applications such as in the automotive industry [5,7,8].

For the aforementioned reasons, the performance analysis of a component may result inaccurate if only a virtual prototype is used for the numerical simulation. This is particularly true in the case of dynamic vibrational and vibro-acoustic simulations, where small geometrical details can vary significantly the dynamic behaviour of a product, especially in the mid-to high frequency range [9–11] where structural wavelengths are smaller than the characteristic dimension of a product. Besides, it should be noted that due to the increasing customer expectations and more restrictive regulations due to the impact of noise and vibration on human health [12], the vibrational and acoustic behaviour of a product has become an important design criterion in many kinds of industry (automotive, aerospace, machine design, etc). Based on these considerations, this work aims at investigating how a simulation framework can be developed that accurately accounts for the geometry of deep drawn components for vibro-acoustic analysis.

In fact, the experimental and numerical simulation of the deep drawing process have been widely studied in the literature of last years — see for instance [13–17]. However, to the authors' knowledge, only a limited amount of research has focused on the vibro-acoustic sim-

---

[*] Corresponding author.
E-mail address: francesco.greco@kuleuven.be (F. Greco).
[1] Member of Flanders Make.

https://doi.org/10.1016/j.finel.2017.10.004
Received 19 June 2017; Received in revised form 3 October 2017; Accepted 10 October 2017

ulation of the final component performance [18–20]. The purpose of this work is therefore to address to this topic by considering both a purely numerical procedure, where the FEM model for the dynamic analysis is obtained from the forming process simulation and also a reverse engineering framework, where the FEM model is obtained from surface measurements on the actual manufactured component. While the proposed methodology is applied to the simulation of the dynamic behaviour of a component, the development of an FEM model with a correct geometrical description has a crucial importance in many other applications, such as for instance stress analysis or performance evaluation simulations.

Based on the FEM simulation of the deep drawing process, it is possible to predict the theoretical thickness distribution of the manufactured component and to account for it in the subsequent dynamic numerical analysis. This is particularly important since, by applying a constant thickness (coming from the original blank material), in practice the mass of the object is always overestimated, which is not desirable for the dynamic analysis. At the same time, other factors such as the springback and the intrinsic variability of the material forming procedure may generate a dynamic behaviour which is slightly different with respect to the one predicted with the nominal model. Therefore, for a more accurate simulation from the geometrical point of view, the exact shape of the component is also considered in this work and is obtained from laser measurements on the real, manufactured geometry. Coordinate measuring machines (CMMs) are nowadays a useful tool for reverse engineering and quality inspection applications [21–23]. In particular, laser CMMs can obtain large amount of point data without entering in contact with the scanned surface in a short time period and are widely adopted for many applications in the industry. Starting from the CMM measurements, an FEM product model is then generated and used for the analysis.

In this study, the purely numerical simulation framework and the one based on surface measurements on the manufactured component are validated based on an experimental test of the dynamic behaviour of the component. The process considered is the deep drawing of DP 600 steel cups. DP 600 is a dual phase steel with a high press-formability and it is widely used in the automotive industry [24]. Given the closed and axis-symmetrical shape of the cups, the springback phenomenon is less pronounced with respect to other sheet metal deep drawn components and is not accounted for in the purely numerical simulation. On the other hand, their final shape is properly captured in the model based on surface measurements.

The outline of the paper is the following: after a brief description of the component given in Section 1, the two simulation frameworks are described in Sections 3 and 4. Section 5 describes the test setup and the dynamic measurements. Experimental and simulations results are compared in Section 6. Concluding remarks follow in Section 7.

2. Component description

The study in this paper considers an industrially relevant case, yet with a simple geometry. A deep drawn cup is studied, as often considered in studies specifically related to the deep drawing manufacturing process, see e.g. Refs. [2,25]. The cup is produced in the DP600 steel with a nominal thickness of 0.97 mm, while applying a blank-holder force of 45 kN and a drawing depth of 24 mm. The parameters are chosen in a way to avoid rupture and defects such as tearing, earing, necking and wrinkling [1,2].

A blank with a diameter of 100 mm is considered. Before starting the deep drawing process, the blank is electrochemically etched using a regular raster of dots with a diameter of 1 mm and a spacing of 2 mm. A Nitto protection foil type SPV224P (t = 75 μm) is applied after rastering, to minimise friction and friction differences. The cup is produced by deep drawing at OCAS on a Erichsen 145-60 press. The punching speed is 50 mm/min and the punch diameter is 50 mm. The die opening is 52.88 mm and the die entry radius is 3 mm. Fig. 1 shows the produced cup. After the deep drawing process, an optical measurement is performed with an ARGUS GOM system, resulting in a 3D point cloud with the coordinates of the part and the corresponding strain.

Fig. 1. The studied component.

3. Development of a product model starting from a FE process simulation

An FE product model is used to calculate product attributes, such as mass, stiffness, strength and normal modes analysis. This section shows how a product model with an accurate thickness distribution can be obtained through an FEM process simulation.

3.1. *Forming simulation*

The deep drawing process is simulated using the commercial software Autoform with the Incremental module, which uses shell elements with an implicit time integration scheme [1,26]. The parts constituting the FEM process model are outlined in Fig. 2. The geometry of the die is imported in the software using a CAD model and the ones of the punch, the blank-holder and the blank are directly derived from it.

The tools and the blank are initially meshed with a regular mesh and during the calculation the mesh of the blank is automatically refined when it is needed, for instance in zones with curved geometry or when cracks occur. In this way the memory use and the computational times are also optimized. The material properties used in the simulation are the following:

- for the hardening curve, the Ludwik law [27] is used. Required parameters are the strain hardening exponent n, the yield strength R_e and the ultimate tensile strength R_m;
- for the yield surface, the Hill criterion [28] is chosen and the normal anisotropy coefficients r_0, r_{45} and r_{90} have to be specified;
- for the forming limit curve (FLD), the Arcelor V9 model [29] is chosen.

The mechanical characteristics of the material used in the aforementioned laws have been determined by tensile tests performed at OCAS and are reported in Table 1.

Finally, the blank-holder force value of 45 kN is also specified in the simulation and a friction coefficient $\mu = 0.12$ is considered. Based on measurements performed during the manufacturing process and on the experience developed by OCAS in this type of forming simulation, a friction coefficient $\mu = 0.15$ is normally employed in Autoform for the simulations involving DP600 steel. However, taking into account the