Journal of Materials Processing Technology 169 (2005) 352–356

Journal of Materials Processing Technology

www.elsevier.com/locate/jmatprotec

Numerical and experimental evaluation of springback in a front side member

A. Andersson [a,b,*]

[a] *Volvo Cars, Body Components, Oloström, Sweden*
[b] *Division of Production and Materials Engineering, Lund University, Lund, Sweden*

Received 1 April 2004; received in revised form 1 April 2004; accepted 11 April 2005

Abstract

In this study, a part of an automotive side front section (front side member inner) was studied and a comparison both regarding material behaviour and of accuracy of the FE simulations was made. Mild steel, Rephos steel and TRIP700 were compared both experimentally and numerically. The results showed that TRIP steel has a significantly larger springback than the other materials. Furthermore, the FE simulations overestimate the twisting in this part for all materials, with the TRIP material showing the largest deviation between the experiments and the simulations. The prediction of punch forces was, however, accurate for all materials.

Keywords: Sheet-metal-forming; Simulation; Finite element method; Springback

1. Introduction

Changes in the geometry after springback are a big and costly problem in the automotive industry. The assembly process cannot handle parts with too much deviation in geometry and therefore adjusting operations are required. These extra operations cost both time and money in the production process and should be reduced to a minimum.

Today, sheet-metal-forming simulations are commonly used to assess the forming process. Reliable results of the forming process can be obtained regarding thinning, forces and fractures. The simulations have contributed to a significant decrease in the use of try-out tools [1], thus decreasing both lead time and costs. The sheet-metal-forming simulations are today widely used in the automotive industry [2,3]. However, prediction of springback is not accurate enough to be fully reliable, and therefore experimental tests are still required, to a large extent in order to evaluate the deviation in geometry due to the springback. The use of finite element (FE) simulations, decrease both the lead time and costs in the development of new parts compared to experimental tests.

Previous publications [4–9] have shown that the accuracy in springback predictions for automotive panels has increased, and that the accuracy is sufficiently good to apply die compensation based on simulation results. However, improvements are still called for. In order to test the possibility to predict springback, a test tool was analysed, namely, a front side member inner to Volvo S40. The geometry consists both of a simple U-shaped part (front) and a complex part (rear). The springback will therefore consist of both section opening (front) and twisting (caused by the complex rear part). These are problems, which occur in the production process and their study is of significant interest.

In this study, three different materials were analysed in order to investigate the correspondence between experiment and FE simulation for different qualities of material. The materials in question were mild steel, Rephos steel and TRIP steel.

TRIP steel is a relatively new material with a potential to decrease the weight or increase the strength of a part. It has a significant deformation hardening and is therefore suitable for deep-drawn parts, which have demands on both high formability and high strength. The formability has been studied by Konieczny [10] and Kamura et al. [11], who found that TRIP steel has good formability and is suitable for

* Tel.: +46 454 265 280.
E-mail address: aander58@volvocars.com.

0924-0136/$ – see front matter © 2005 Published by Elsevier B.V.
doi:10.1016/j.jmatprotec.2005.04.095

automotive applications. However, as this material is used to a small extent in the automotive industry and it therefore was of interest to include it in this study. The experiments were performed under controlled conditions. In order to verify the simulation results, these were compared to experimental results after both the blank holder closing (binder wrap) and the forming/springback. A trimming operation after forming was also included in the study.

2. Methodology

2.1. Methodology for experimental tests

The chosen geometry is well-defined, since it is the geometry of a try-out tool, for which CAD data correspond to the physical tool. The experimental test was performed in a hydraulic double-action press, which provided the possibility to perform a well-controlled process.

In order to have a well-defined process to simulate, distances of 0.1 mm between the blank holder and the die were used during the forming process. Hence, the flow of material will be controlled by the draw-beads which provide the restraining forces, and the binder will only control the tendency of the blank to wrinkle during the forming process.

In order to calibrate the draw-in in the simulation, one part was analysed after the blank holder was closed. This part was used as a reference when the draw-in was measured after the final drawing. The draw-in was then measured as the change in flange width between binder wrap and final shape at several positions. In order to have the correct position after the blank holder was closed, also the flange width was measured and compared to the simulation results. Furthermore, the punch force was registered in order to compare it with the simulation results. The trimming was done by laser cutting.

The measurement of the springback was based on a 3–2–1 system. This means that all six rigid body modes are suppressed by support in five points. These points are located at three points in z, two points in y and one in x (see Fig. 1). The parts were measured in a CMM machine and the deviation from the CAD model was evaluated in two different

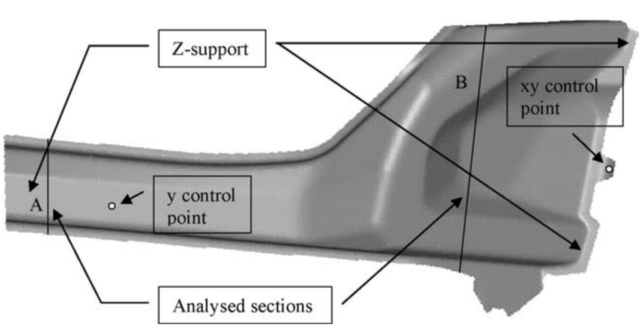

Fig. 1. Support points and evaluated sections.

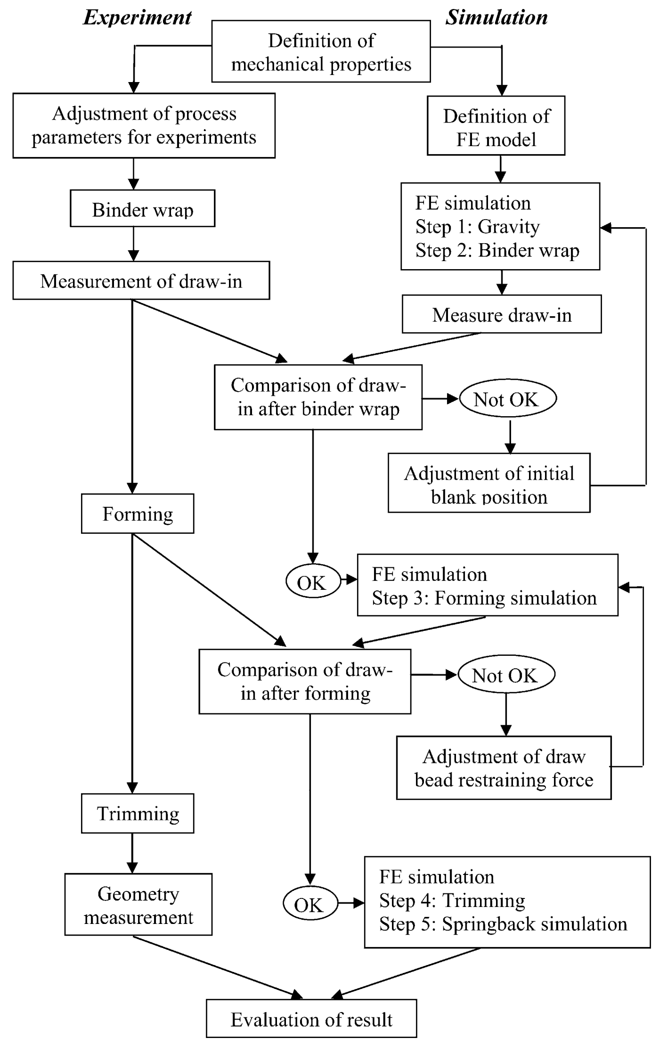

Fig. 2. Methodology for simulations and experiments.

sections, which correspond to the two different cases mentioned in Section 1 (see Fig. 1): a U-profile which generates 2D-opening (section A) and a more complex section (section B) which also generates twisting.

Three parts were evaluated for each material in order to find the scatter in the experimental results.

The methodology for the experimental analysis is described in Fig. 2 together with the methodology for the simulation.

2.2. Simulation methodology

The simulation methodology is described in Fig. 2. First the mechanical properties of the material were obtained from a tensile test.

Based on the CAD geometry, a FE model was created. The simulation was divided into five steps:

1. gravity simulation;
2. binder wrap simulation;