Tool Design for a High Strength Steel Side Impact Beam with Springback Compensation

Authors:

Trevor Dutton, Dutton Simulation Ltd
Richard Edwards, Wagon Automotive Ltd
Andy Blowey, Wagon Automotive Ltd

Correspondence:

Trevor Dutton

Telephone +44 1926 732147
Fax +44 1926 732147
Email trevor@duttonsimulation.com

Keywords:
Springback prediction and compensation, high strength steel

ABSTRACT

Prediction of formability for sheet metal pressings has advanced to a high state of confidence in recent years. The major challenge is now to predict springback and, moreover, to assist in the design of tooling to correctly compensate for springback. This is particularly the case for materials now being routinely considered for automotive production, such as aluminium and ultra high strength steels, which are prone to greater degrees of springback than traditional mild steels.

This paper presents a case study based on the tool design for an ultra high strength steel side impact beam. The forming and springback simulations, carried out using eta/DYNAFORM (based on the LS-DYNA solver), are reported and compared to measurements from the prototype panels. The analysis parameters used in the simulation are presented, and the sensitivity of the results to variation in physical properties is also reviewed. The process of compensating the tools based on the analysis prediction is described; finally, an automated springback compensation method is also applied and the results compared with the final tool design.

INTRODUCTION

In 2004, Wagon Automotive were appointed by Honda of the UK Manufacturing to develop a new concept for a side impact beam for use in the front door of a new vehicle. A concept of a single stamping was developed; this has the benefit of simplified welding and assembly operations in comparison to other concepts involving a sub-assembly of e.g., a constant section roll-form, extrusion or tube attached to endplates for connection to the door.

CAE performance predictions established that the design would need to have a W profile and the material would have to be ultra high strength steel (yield strength in excess of 1000MPa), in order to achieve the required force vs. deflection performance within the available packaging space. The initial concept is shown in Figure 1; the part is 990mm long, 100mm wide and 35mm in section depth.

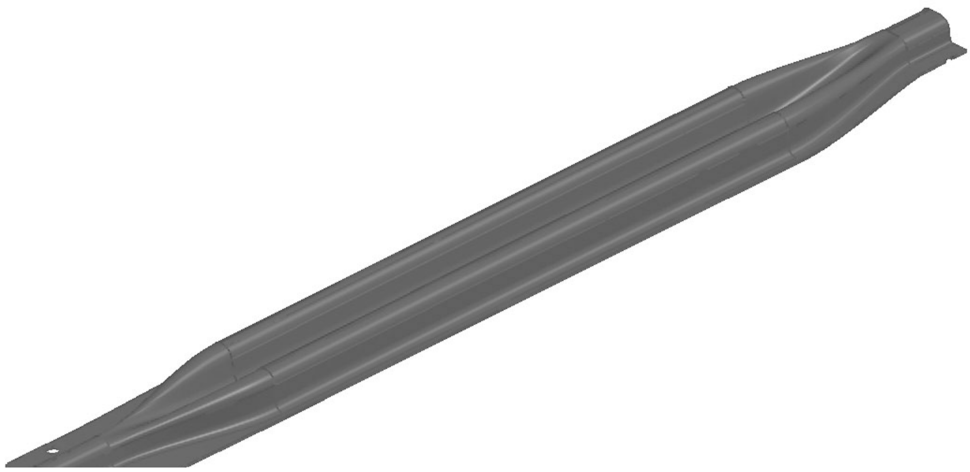

Figure 1: Side Impact Beam concept design

It was quickly appreciated that stamping even such a gently curved profile in ultra high strength steel would not be simple – and in particular, problems with springback were anticipated. Dutton Simulation was asked to assist in the prediction of formability and springback, to assist the appointed tooling engineer (Brierley Ltd) in developing the tool design.

SPRINGBACK PREDICTION

LS-DYNA has been used to predict formability of stamped panels since the 1980s and most practitioners are now highly confident in their analysis results. Hundreds of thousands of dollars and many weeks of program time are saved every year when CAE is used to test forming processes and tool designs prior to build.

More recently, the focus has been on improving springback prediction in order to minimise rework in die tryout when tools have to be re-cut late in the program to achieve a component within specified tolerances. Numerous papers presented in the past ten years show that, using the implicit solver in LS-DYNA (especially in Version 970), accurate springback prediction is now attainable. Guidelines on best practice have been published by Maker & Zhu (Reference 1). The triennial NUMISHEET conferences have set a series of benchmarks for springback prediction; at the last conference in 2002, LS-DYNA showed consistently good correlation (Reference 2).

In the early 2000s, the Springback Predictability Project sponsored by NIST (Reference 3) examined the ability of a range of finite element simulation software to predict springback in automotive panels. This work showed the predictions from LS-DYNA to be consistently the best and led to further work on compensation methods.

These considerations suggested that springback prediction for the side impact beam was achievable and so, working closely with the appointed tooling engineer, a series of simulations was carried out to support tool design.

FORMING SIMULATION FOR SIDE IMPACT BEAM

Initial tool designs were prepared by the tooling engineer based on the proposed CAD model. After considering a number of options in parallel with development of the component design, eventually a two-stage forming process was established. This involved an initial draw from a rectangular blank creating the full “W” profile, followed by a restrike or setting operation to restore any areas that moved excessively. A final trim and pierce completed the process.

Simulation of the two-stage forming operation was carried out using LS-DYNA (Reference 4). The model was set up using eta/DYNAFORM (Reference 5). Shell element formulation type 16 (fast fully integrated) was used throughout with nine integration points to aid in springback accuracy. Forming simulations used the single precision explicit solver while springback used double precision implicit.

The ultra high strength steel, JSC 1180, was modelled using material type 036 (Barlat & Lian, 1989). This material has a minimum tensile strength of 1180MPa. A stress vs. strain curve was employed directly – the equivalent work hardening exponent (n) was 0.12. Anisotropy parameters were given as r_0 = 0.73, r_{45} = 1.1 and r_{90} = 0.83. Thickness was 1.6mm. Forming limit data was also provided.

TOOL MODIFICATION FOR OBSERVED SPRINGBACK

Given the uncertainty over springback, simulation of the initial prototype tool was carried out prior to tool build. Formability results were acceptable; however, predicted springback was considerable, as shown in Figure 2.

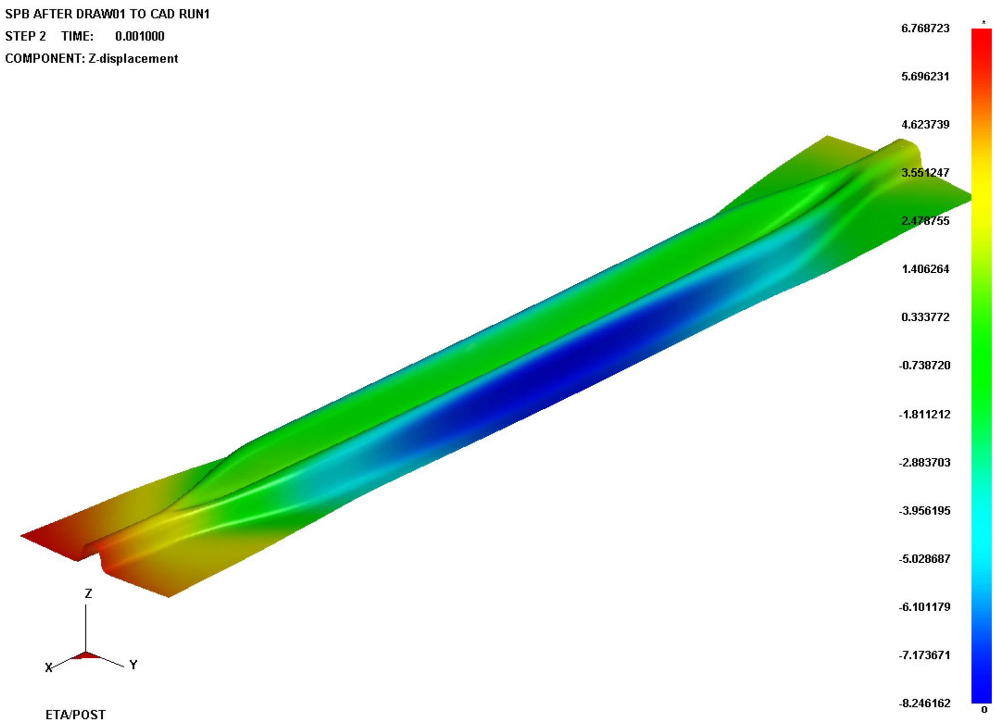

Figure 2: Springback prediction (prior to trim) from initial tool design

The following significant distortions were observed: the form sagged from end to end with an overall vertical movement of more than 15mm; the centre of the "W" reduced in height relative to the outer parts by 6mm; and the "W" spread out by up to 5mm on each side. The flanges also drooped downwards.

This result was used to manually compensate the tooling CAD model before initial tool build. A new design was created with the inverse of these distortions, to the best of the engineer's judgement. Simulation of this design indicated that the results from the compensated tool would be much improved compared with the original so the prototype tool was built to the compensated design and trial parts produced.

SPRINGBACK RESULTS FROM MANUALLY COMPENSATED TOOL

The prototype tool with "manual" springback compensation, based on the initial simulation, was found to produce a panel that was reasonably close to the required geometry after the restrike and trimming operations. However, the section depth and width were still somewhat less than desired. A certain amount of reverse engineering was employed at this point. Even though the panel off the compensated tooling did not conform exactly to the original design, a panel was tested for compliance with the Force vs. Deflection criterion and found to meet requirements. It is believed that the work hardening due to forming may have increased the strength of the part sufficiently to pass the test, despite the loss of section height.

The panel from the tool with compensation exhibited a number of formability issues due to the changes in tool geometry. The increase in depth led to a “risk of crack” area appearing at the transition to the narrow end; and the non-flat blankholder allowed a certain amount of wrinkling to develop in two places on either side of the form. Nonetheless, the result was felt to be acceptable.

These results were confirmed by the simulation of the compensated design. Figure 3 shows the formability results for the trimmed panel. The small marginal patch at the transition (lower left) corresponds exactly with the location observed on the panel, and the wrinkling patches also match very well.

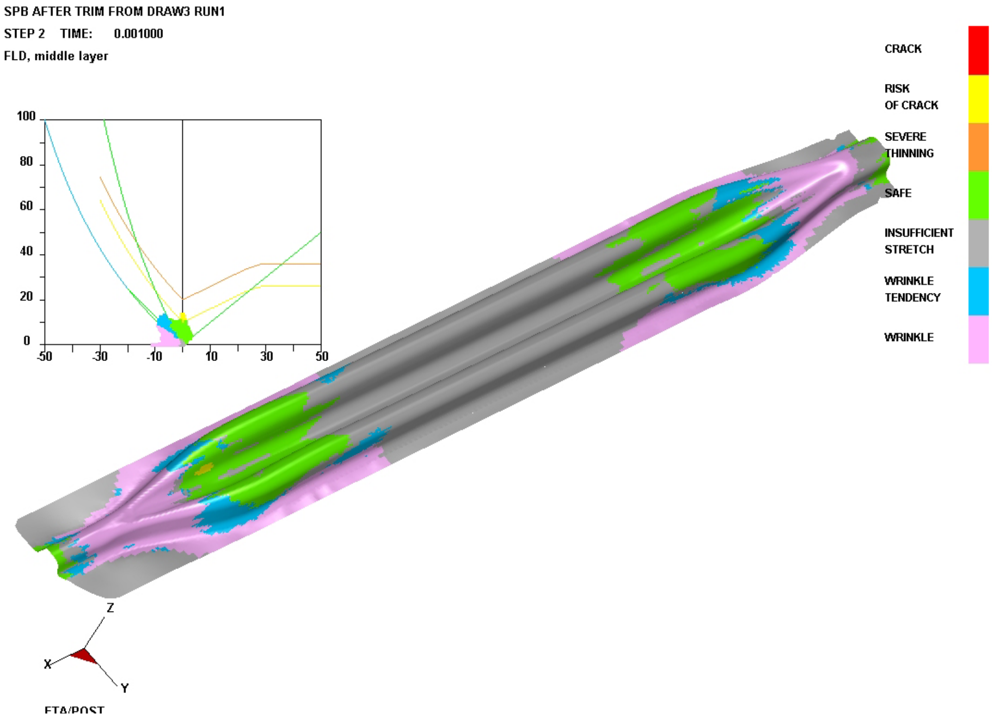

Figure 3: Forming simulation results for modified tooling

Several sections were digitised from the finished panel which allowed comparison with the springback predictions from the compensated model. Figure 4 compares the results for nine sections.

The correlation between the scanned data from the panel (black lines) and the sections cut from the simulation model (grey lines) is generally very close; the sections at either end are almost line-on-line, while the four sections through the mid-region show slightly less springback in the simulation than in reality – the simulation results have spread out less and the flanges have stayed slightly flatter. Overall, the maximum difference here never exceeds 2.0mm.

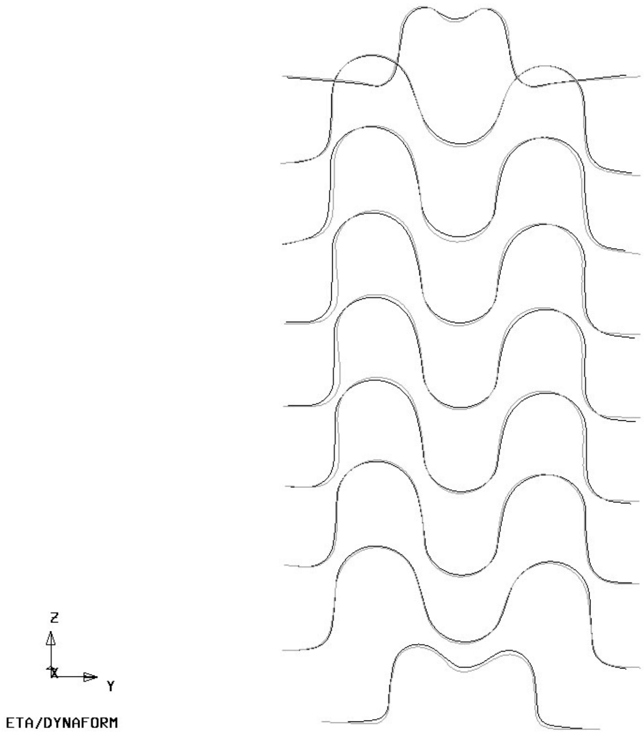

Figure 4: Comparison of sections from panel (black) and simulation (grey)

EFFECT OF VARIATION IN YIELD STRENGTH ON SPRINGBACK RESULTS

One concern with ultra high strength steel is the potential variability in the mechanical properties of the delivered material. The simulations were repeated with yield stress varied by ±15%. In fact, the difference in predicted springback was not as great as the variation in yield; overall springback displacement was reduced by 8% with the lower yield material and increased by 7% with higher yield.

The trend in the results was that the higher strength material slightly increased the reduction in section height and increase in section width (making the results correlate a little better with the scanned sections). This is an area meriting further investigation – predictions based on actual material data measured from different batches should be used to confirm that the final component will remain within specified tolerance.

AUTOMATIC SPRINGBACK COMPENSATION METHOD

The manual compensation approach described above was quite successful in achieving an acceptable panel. However, it would be better still to be able to employ an automatic method that produced a modified tool. This is exactly the goal of the Springback Compensation Project, sponsored by US Government and with participation from DaimlerChrysler, Ford, GM, Alcoa, US Steel and LSTC (References 6, 7). The outcome of the project is a software tool that is able to generate a modified set of tooling to compensate for springback.

The input to the software is as follows:

- Reference geometry. i.e., the target shape

-
- The current blank shape prior to springback (trimmed or untrimmed)
- The current blank shape after springback
- The current tooling geometry

Generally, two to three iterations are needed to get to an acceptable result. A scale factor is applied to the compensation process as it has been observed that "full" compensation (i.e., a scale factor of 1.0) leads to over adjustment. Best values appear to be in the range 0.5 to 1.0 but this is geometry and process dependent.

A smoothing method and factor must also be chosen; these are again case dependent. The purpose is to ensure that the resulting tool is not overly distorted by any wrinkling in the sprungback shape, and that there are no glitches in the geometry in areas of transition and extrapolation. Any undercut generated in the process is identified. An algorithm is included to avoid undercut but is not suitable for all cases; further development is intended to eliminate undercut.

The output is a new set of tooling (finite element mesh) and a reference file which can be used if further iterations are required. Once an acceptable result is achieved, the compensated tooling mesh can be mapped to the original surface data using eta/DYNAFORM.

RESULTS FROM AUTOMATIC SPRINGBACK COMPENSATION

With assistance from LSTC, the springback compensation software was retrospectively applied to the side impact beam problem. The starting point for the process was the original tooling design to the initial CAD model. A range of scale factors was examined but the best results were found using a factor of 0.8.

Figure 5 shows five sections cut through the middle of the component (only half the part is shown). The black line with square symbols is the target geometry; this is the from the original component CAD model. The line labelled "Initial" (diamond symbols) is the springback shape from the tooling based on CAD. The remaining three sections show the sequence of convergence with the tooling generated from three iterations of the springback compensation; iteration 1 (crosses) has moved close to target over the "W" region but the flange still has some way to go; iteration 2 (circles) is very close and has closed the distance to the flange; iteration 3 (stars) is line-on-line with the target for most of the "W" and is within 1.0mm of the flange. Other locations along the part show similar or better convergence.

Figure 6 compares the original CAD model for the tool with no compensation (square symbols) with the tool developed by traditional, manual methods (diamonds) and the tool created from three iterations of automatic springback compensation (stars). The sections are aligned at the part centreline. The trend in the two compensation methods is extremely similar; the "W" itself is made both deeper and narrower and the flange has been angled up (to a greater degree in the manual method). Compensation in the automatic result is slightly less; this is in line with the comparison between panel and simulation earlier where the simulation slightly under-predicted springback. Although the tools have not (yet) been re-cut to the automatic result, it would appear that the likelihood is that the springback correction would be confirmed.

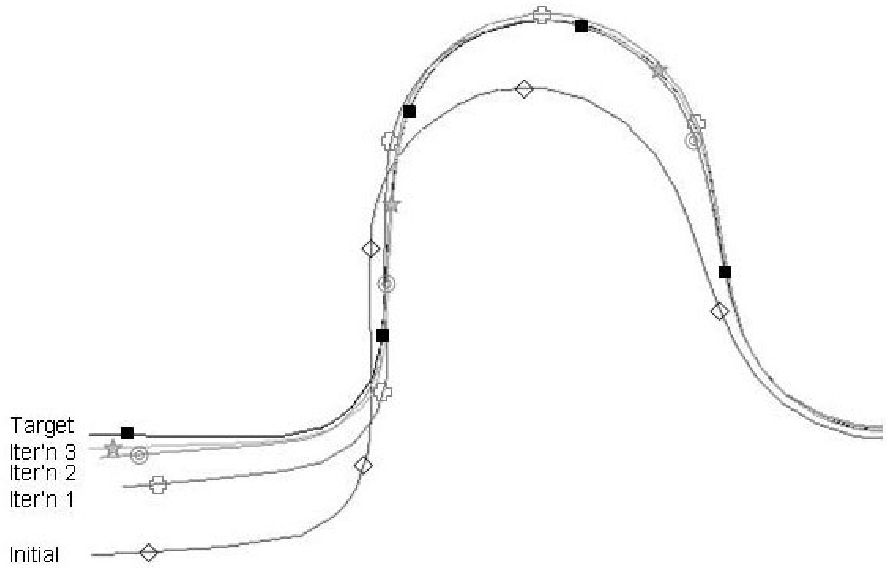

Figure 5: Results of Automated Springback Compensation; sections show trend from initial springback (diamonds) through three iterations (1=crosses, 2=circles and 3=stars), compared with the target geometry (squares)

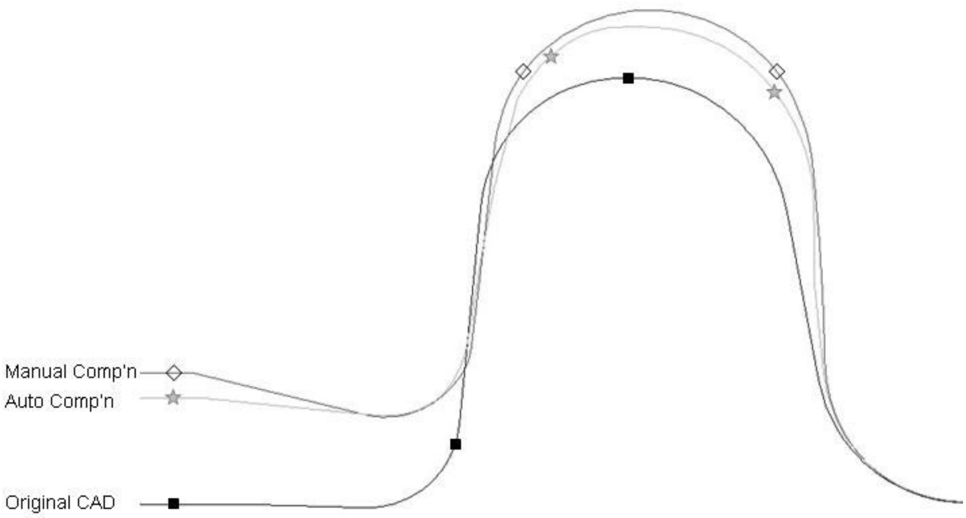

Figure 6: Tool sections comparing the original CAD model (squares), the tool developed by manual compensation (diamonds) and automatic compensation (stars)

CONCLUSIONS

The conclusions of this study were as follows:

- eta/DYNAFORM and LS-DYNA can be effectively used to predict formability and springback in ultra high strength steel components;
- manual adjustment of tooling can be supported by simulation results to achieve an acceptable result;
- automatic compensation using new software developed by LSTC appears to offer a powerful method to develop tooling with compensation for springback built in.

ACKNOWLEDGMENTS

The authors would like to acknowledge the assistance of Julian Tebbutt, Honda of the UK Manufacturing Ltd; Andy Gray, Brierley Ltd; and Xinhai Zhu, LSTC

REFERENCES

1. Input Parameters For Springback Simulation Using Ls-Dyna. Bradley N. Maker, Xinhai Zhu. 3rd European LS-DYNA Users' Conference, Gothenburg. June 2001.

2. NUMISHEET 2002, Proceedings of the 5th International Conference and Workshop on Numerical Simulation of 3D Sheet Forming Processes, Jeju Island, Korea, 21-25 October 2002. Dong-Yol Yang (ed.), Soo Ik Oh (ed.), Hoon Huh (ed.), Yong Hwan Kim (ed.). 2002.

3. Materials Science and Engineering Laboratory. FY2002 Programs and Accomplishments. Metallurgy Division. Carol A. Handwerker, Frank W Gayle, National Institute of Standards and Technology, U.S. Department of Commerce. NISTIR 6907, p 37. Sept 2002.

4. LS-DYNA Keyword User's Manual, Version 970. Livermore Software Technology Corporation. April 2003.

5. eta/DYNAFORM User's Manual, Version 5.2. Engineering Technology Associates Inc. 2004.

6. A New Concept on Stamping Die Surface Compensation. Li Zhang, Jin Wu, Dajun Zhou, Xinhai Zhu. 8th International LS-DYNA Users' Conference, Dearborn, Michigan. May 2004.

7. Springback Compensation with LS-DYNA. Xinhai Zhu, LSTC. Presentation at DYNAmore seminar, December 2004.