Sheet metal forming simulation-
Closing the design loop

Omar Ghouati, Xiao Chen

Ford Research & Advanced Engineering Europe, Aachen, Germany

Abstract:

Forming simulation has now reached an acceptable level of accuracy. It is now possible to predict the hardening, the thinning and required forces for sheet metal stamped parts. It is also possible to predict the geometrical defects such as springback, wrinkling and surface appearance problems. Sheet metal forming can therefore be used in a closed loop to help design parts and required tools to achieve a pre-defined geometry and mechanical performance.

The paper presents different possibilities to include sheet metal forming simulation in a closed loop design and optimization process. These possibilities are FEM based materials identification, springback compensation, die and part optimization and finally the coupled part and process optimization. Before describing these different possibilities, an example illustrating the level of accuracy that can be achieved using sheet metal forming simulation is shown.

Keywords:

Stamping, High Strength Steels, Springback, Springback compensation, Optimization, Forming history.

1 Introduction

In the automotive industry the car body structure is made of a large number of sheet metal parts. These parts, usually with rather complex geometries, make use of different materials varying from mild steels to very high strength steels or aluminum depending on the product requirements and the allowed costs. For the design of these parts, shorter development times with lower costs are available and few iteration loops can be used to come up with the optimal design both for the part as for the die set used to produce it.

The finite element simulation plays now a major role in designing sheet metal parts and is now extensively used to predict the material and geometrical properties. Various developments regarding element formulation, material modeling and the necessary numerical methods have been made. These developments have increased the accuracy and reduced the computation time leading to a higher level of confidence in the numerical predictions. The challenges facing the sheet metal forming simulation are however also evolving. As a matter of fact, recent years have shown a tremendous increase in the usage of Advanced High Strength Steels (AHSS) such as Dual Phase steels, Transformation Induced Plasticity steels and Ultra High Strength Steels (UHSS) such as USIBOR1500 for crash relevant parts. The AHSS are usually cold formed and represent therefore a challenge for the simulation regarding material behavior, press force, formability and springback prediction. The UHSS are generally hot formed and represent a challenge for the simulation in terms of the process and the material behavior. Moreover, the demands on forming simulations are increasing. The results of these simulations are not only used to get a part free of cracks and wrinkles but also to ensure that the springback can be controlled and to be able to deal with surface defects. For this purpose, several numerical tools are now available for performing the springback compensation in order to modify the die set so as to minimize the springback.

Moreover, for High Strength Steels, it is necessary to take into account forming history in order to get correct product evaluation simulations such as crash, durability or NVH simulation. It is therefore of major importance to be able to predict the formability and springback for these materials. This enables the development of optimized part designs taking into account the manufacturing and product performance requirements. The coupling of forming simulation and product performance simulation is now also possible thanks to several available tools. These tools enable transferring the forming simulation results to crash, durability or NVH models taking into account the differences in modeling.

All the elements are therefore available for a CAE based part and tool design. The difficulty now is to make use of these elements with the level of accuracy available in order to provide optimal designs in shorter times and lower costs. In order words, the goal now is to find the appropriate optimization tools that allow closing the loop of sheet metal parts design.

In this paper, we will first review the status of the sheet forming simulation by providing examples illustrating the accuracy of the results for different materials.
Then some rather new applications will be described. These applications are springback compensation and the use of forming results for product performance simulations.
Finally we will demonstrate how the forming simulation can be used to identify the behavior of complex materials and be included in an automatic optimization process.

2 Formability and springback investigations

In this chapter, a study concerning some AHSS is presented. This study has as a goal to assess the possibility using FEM simulation to reproduce the forming and the springback behavior. For this purpose two lab-samples widely used for such investigations were used: a LDR test (Limiting Drawing Ratio) for the formability and the U-shape test for the springback.

2.1 Simulation setup of the LDR test and Material properties

Different drawing ratios were tested for a target final draw depth of 55 mm. For each test, the punch force was measured. The geometry (Figure 1) of the LDR test was the following:

- Die ring with a diameter of 101.5 mm and an entrance radius of 8.0 mm;
- Punch with a diameter of 97.5 mm and a radius of 12.5mm (punch-die clearance of 2.07 mm);
- Diameter of the initial blank was varied to achieve different drawing ratios.

As during the tests it was observed that a ratio of 2.0 was the largest ratio without failure in the drawn cup, this ratio was considered for the simulation validation. This ratio corresponds to an initial blank diameter of 197.2 mm. The draw depth of the cup during the simulation was 45 mm. A drawing sheet was used on the die side. Therefore a coulomb friction coefficient of 0.05 was used on the die side and 0.1 otherwise. The blank-holder force was assumed to be 250 kN as used during the tests.
For the simulations, a radial symmetry was assumed and only a quarter of the test was modeled.

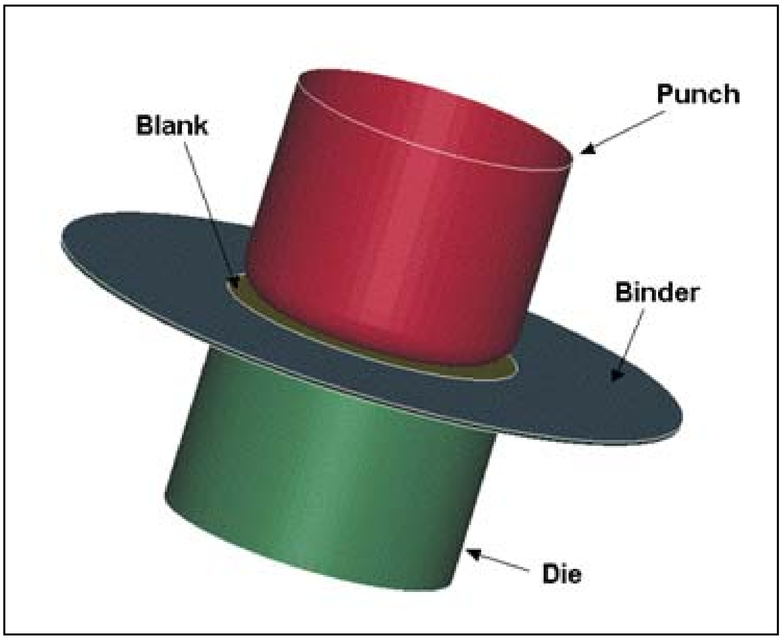


Figure 1: Description of the deep draw test

Four different materials (Table 1) were investigated in this study. The related material properties are described in Figure.3.

Material	ZStE340	DP600	DP780	TWIP
Thickness (mm)	1.50	1.60	1.50	1.90

Table 1: Material investigated and corresponding thickness

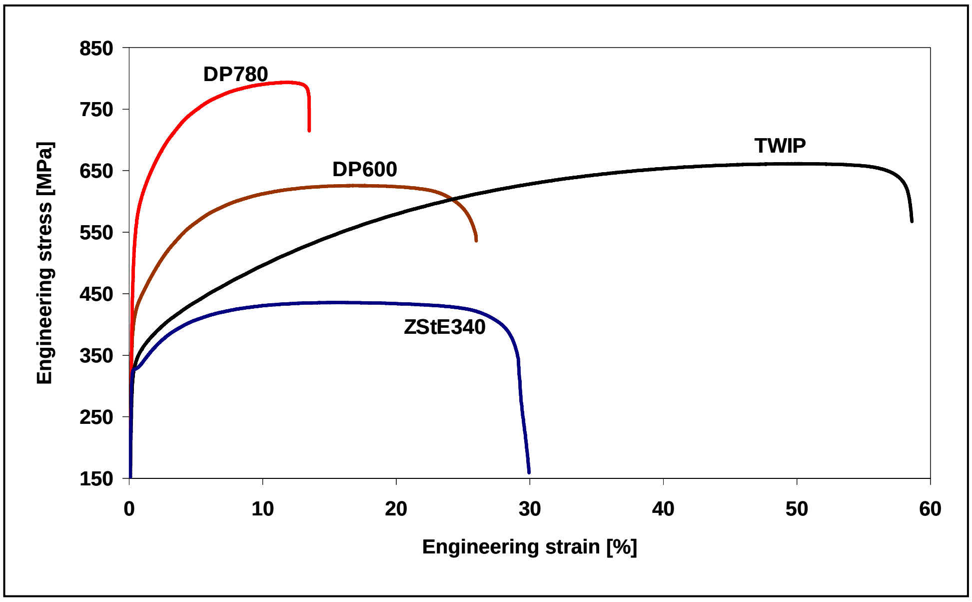
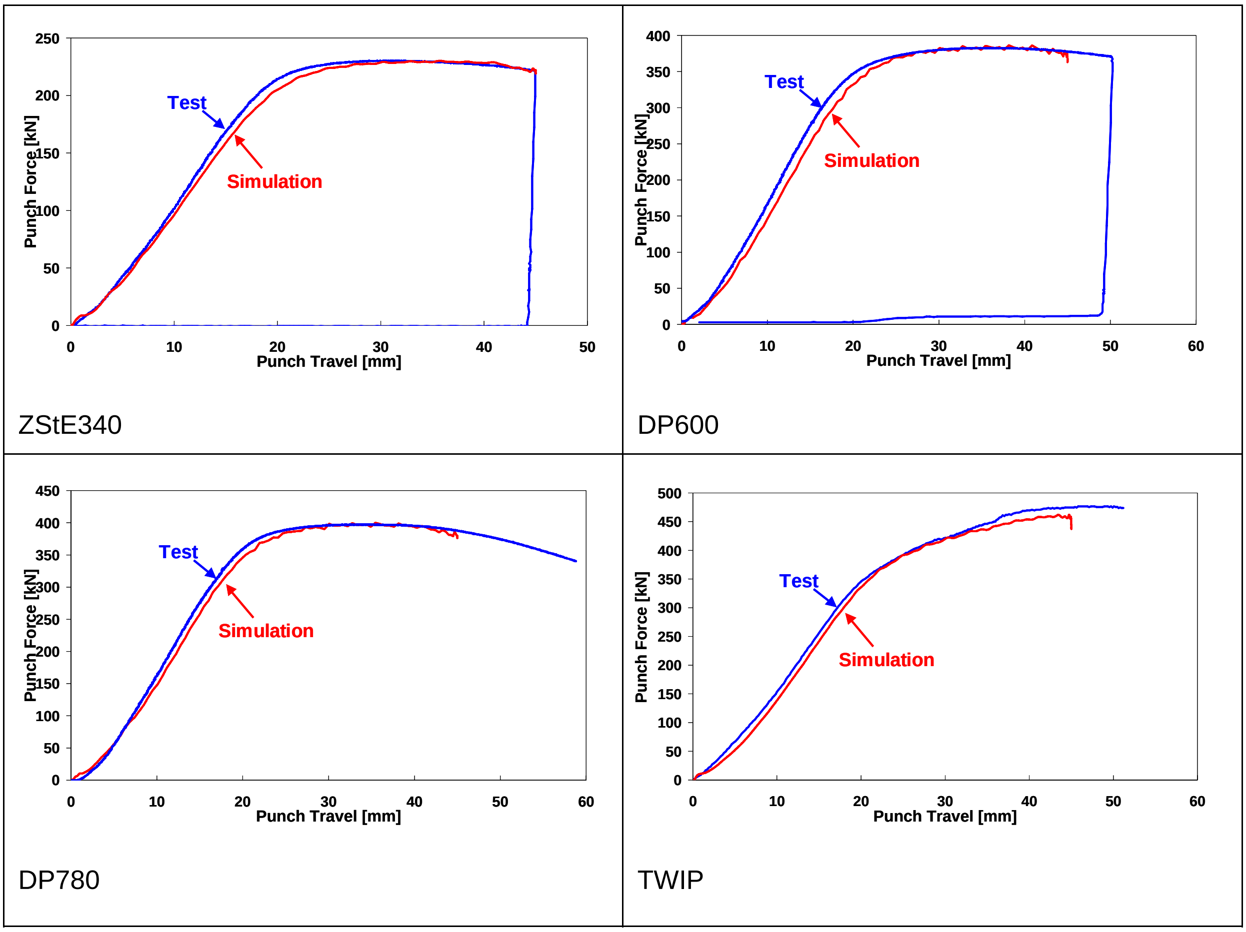


Figure 2: Engineering stress-strain curves of the investigated materials

The simulations were carried out with LS-DYNA version 970_5434, where the explicit method was used for the deep drawing and the implicit one for springback. Table 2 contains the numerical parameters used for the simulations.

Parameter	**Used Value**
Punch Velocity	10 mm/s
Through-thickness Integration Points	5
Mesh Size	5 mm adapted 40 times to 3 levels
Element Type	Belytschko-Tsay (#2)
Friction Die side	0.05
Friction Punch side	0.1
Material Model	Transversely anisotropic elastic-plastic model (#37)
Contact Type	Forming one way surface to surface

Table 2: Numerical parameters used in the simulations



ZStE340

DP600

DP780

TWIP

Figure 3: Simulated punch forces compared to the experimental ones for the investigated materials.

Figure 3 represents the simulated punch forces compared to the experimental ones. A rather good agreement can be observed for all investigated materials. This good agreement was achieved because the experimental set-up was precisely controlled. A special attention was also paid to the lubrication so as to have controlled conditions. As a matter of fact, wrong values for the friction coefficients can lead to wrong results for the simulation. The material data was also precisely determined using tensile tests from the same blanks as those used to form the cups. This demonstrates that when using the same conditions as the experimental tests, the simulation provides rather good results for the punch forces. By the conditions, we mean here:

- Geometry
- Material
- Lubrication
- Other process parameters (blank-holder force if used, distance pieces, etc.)

3 Complex materials identification

Several new material models were developed in order to improve the accuracy of the forming simulation results. These material models make however use of several parameters that are difficult to identify. In order to minimize the number of necessary characterization tests, a FEM based identification procedure can be used. This allows using a complex test exhibiting different loading paths. The advantage of this approach is the fact that global quantities are used as response. These quantities are for instance punch forces, pressures, punch travel and so on. This way, there is no need to convert these quantities to local quantities such as stresses and strains. This is done via the finite element simulation and therefore there is no need for approximations that could lead to inaccurate material data [1].

The principle of the FEM based material identification is represented in Figure 4. The FEM is used to evaluate the response in a chosen test using a certain set of material parameters. These material parameters will be adjusted until this calculated response matches the measured one.

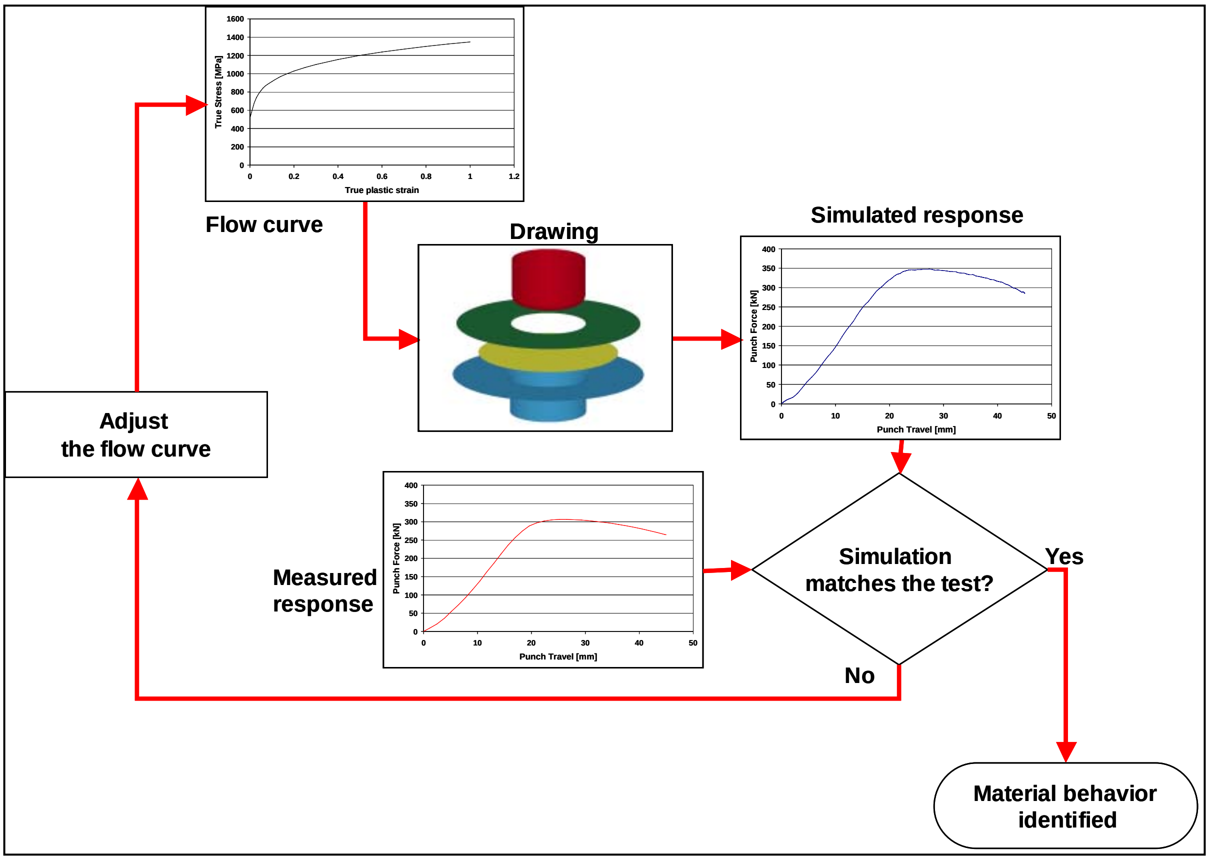


Figure 4: FEM based complex material behavior identification.

4 Springback simulation and springback compensation

Springback simulation has taken advantage from the many developments in the forming simulation tools. This combined with an experienced user provide an acceptable accuracy for springback predictions. Furthermore, several developments lead to the implementation of strategies for the springback compensation. This enables the design of a die set closer to the optimal die leading to a part geometry satisfying specified tolerances.

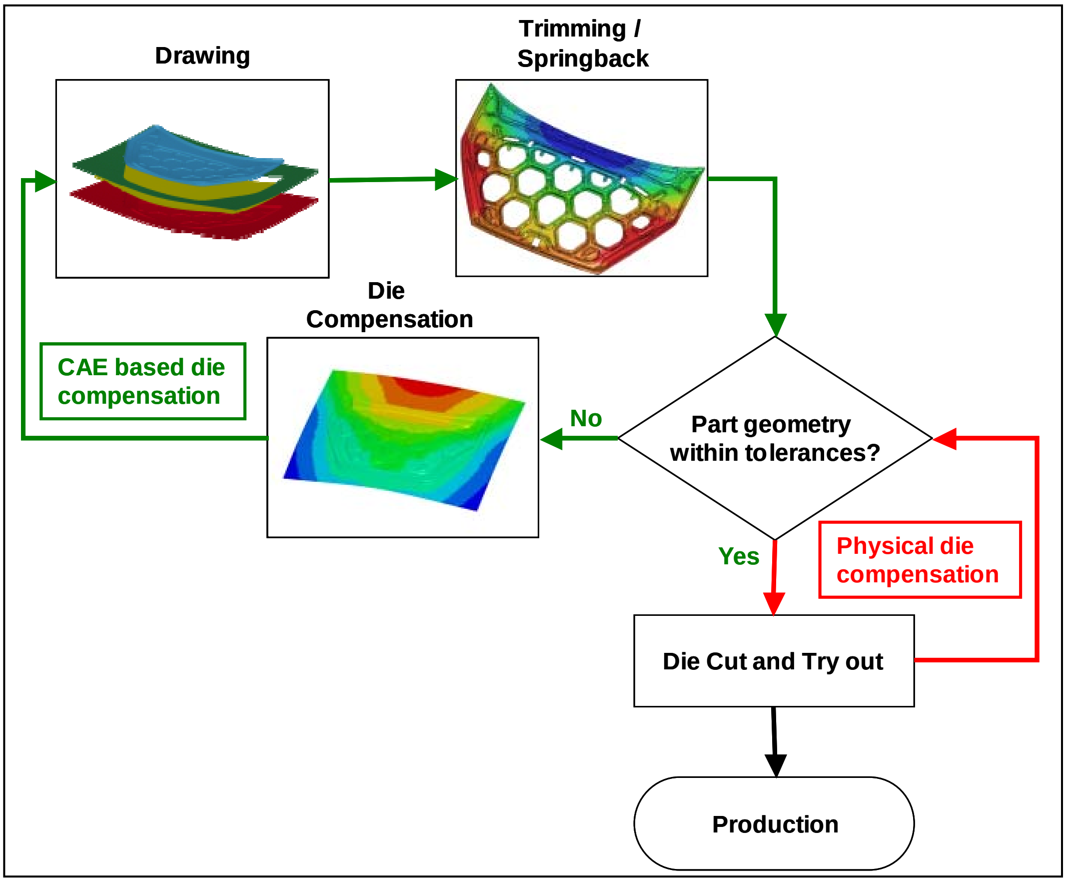


Figure 5: Principle of the implementation of CAE based springback compensation in die design.

Figure 5 represents the principle of the implementation of CAE based springback compensation in die design. The goal is to run as much iterations as necessary within the CAE based compensation loop in order to get closer to the optimal design. As a consequence the number of iterations in the physical die compensation loop will be reduced to a minimum leading to a considerable time and cost saving during the try-out phase.
To illustrate this approach, the example of a hood inner will be presented. Figure 6 represents the result of the springback after trimming using the initial die design.

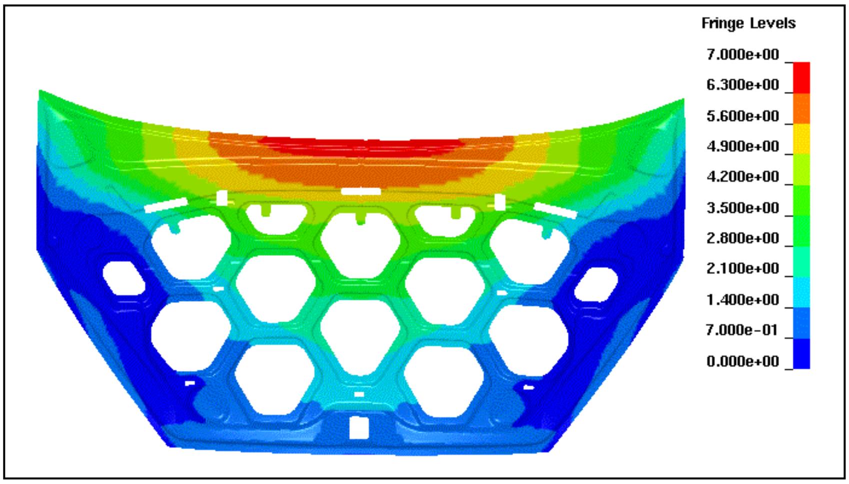


Figure 6: Distance to the part design geometry using the initial die design.

Using the springback compensation tool developed by Cedric Xia at the Ford Research and Advanced Engineering N.A. a new die was determined (Figure 7.

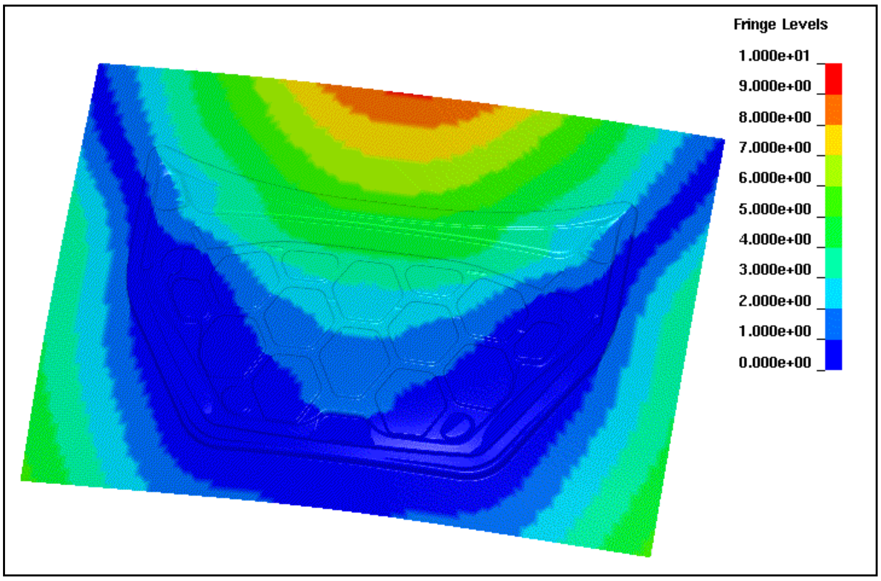


Figure 7 Compensated die obtained after the first iteration.

The geometry of the part after trimming and springback using this die geometry is represented in Figure 8 and shows already a rather big improvement. After 3 iterations, it was possible to obtain the geometry within the required tolerances.

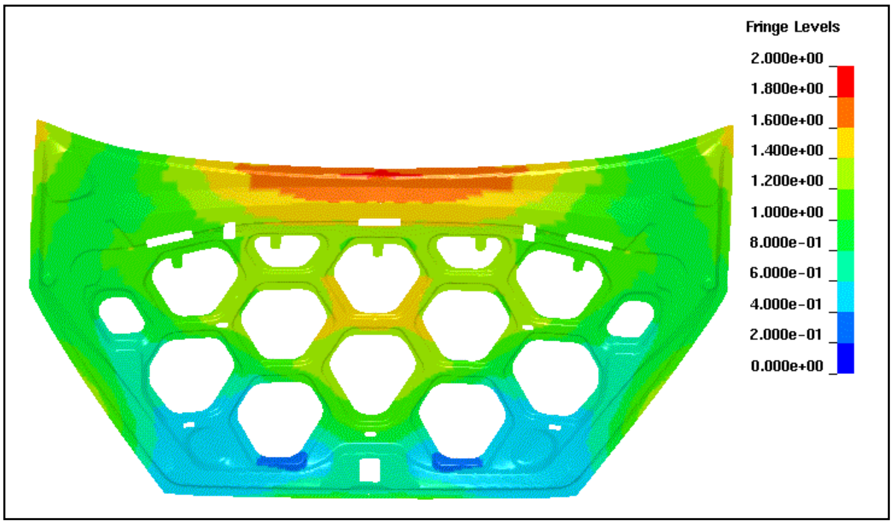


Figure 8: Distance to the part design geometry using the compensated die design.

5 Automatic part and process design

The development times for sheet metal parts are getting shorter and shorter which requires an efficient die design with fewer physical die rework. In this context, an automatic CAE based die and process optimization allows considerable time and cost savings. This has mainly taken benefit from the development in computation capabilities and simulation tools, which reduced the necessary time for a single simulation. This has also taken advantage from the availability of efficient optimization tools that allow not only to deal with the mathematical problem of optimization but also to integrate the FEM tool used for sheet metal forming simulation.

Figure 9 represents the principle of die and process optimization. Starting with the part geometry, an initial die is designed. Using this die the forming simulation will be performed. As long as the part contains cracks, wrinkles or any unacceptable defect, the part, die and process will be modified. The possible modifications has to be pre-defined by the user and will be automatically performed by the numerical optimizer following a user-defined strategy [2].

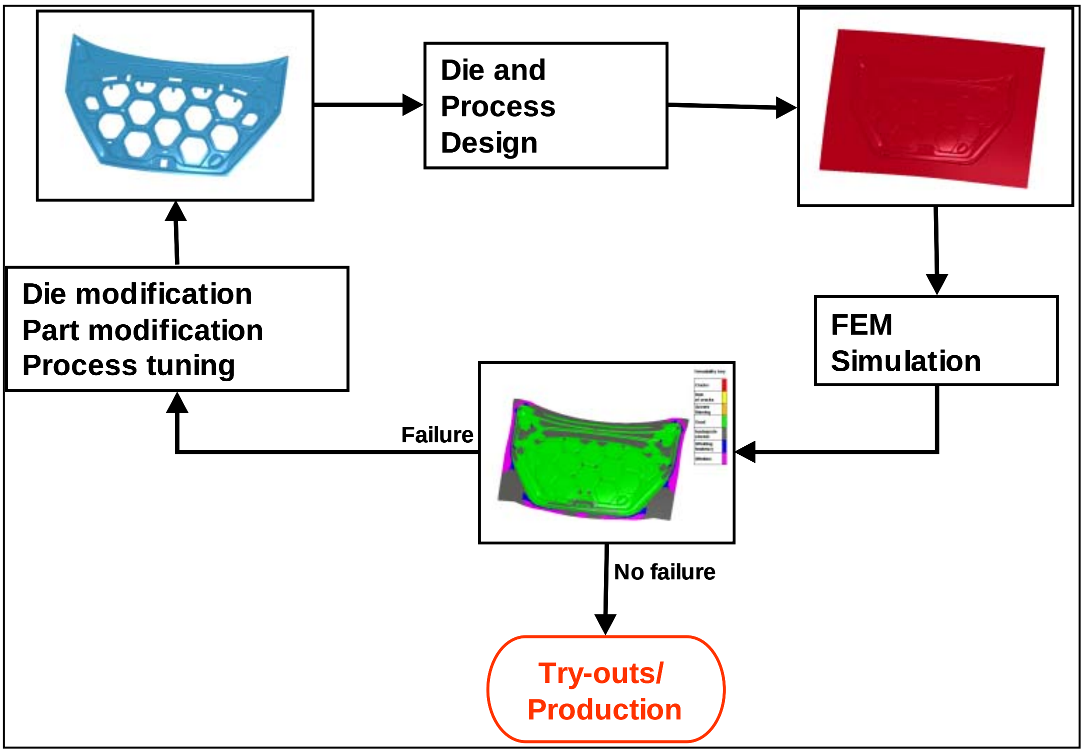


Figure 9: FEM based part and die design

6 Linking forming simulation and product performance simulations

FEM simulations to assess product performance (crash, durability, NVH…) are nowadays well established in product development. On the other hand, the tendency is to reduce the number of physical prototypes until a late stage of development. More accuracy is therefore demanded from the FEM simulations predictions. Parallel to this, high strength steels (HSS) are increasingly used in order to achieve a better efficiency. These materials show a rather high hardening during stamping ending up with material properties completely different from the virgin material. It became then obvious that in order to reproduce the actual behavior of the material during a product performance simulation, it is necessary to take into account the history of deformation. As a matter of fact, during the forming operations, the material experiences several behavioral changes, mainly:

- change in the thickness distribution,
- change in the material properties, mainly the current yield strength is changed due to the hardening,
- residual stresses in the part.

First attempts to improve product performance simulations were applied to NVH by simply taking into account the thickness distribution after stamping. These studies showed that the results were improved compared to simulations were the initial blank thickness was considered. Similar studies were conducted to improve crash, dent-ability or durability simulations. In this case, not only the thickness distribution but also the hardening was updated so as to take into account the forming history. The first studies achieved this update by defining different material properties for several areas of the component in order to reflect the effect of forming [3-17]. It is now possible to fully couple sheet metal forming simulation and product performance simulation tools. The principle of this coupling is represented in Figure 10. The coupling tool has to be able to transfer the forming results of several parts from the forming simulation to the product simulation taking into account the differences in modeling between the two simulations and without lost in accuracy. This coupling is the first step in the coupled process and product optimization leading to an optimal design covering product requirements and process conditions.

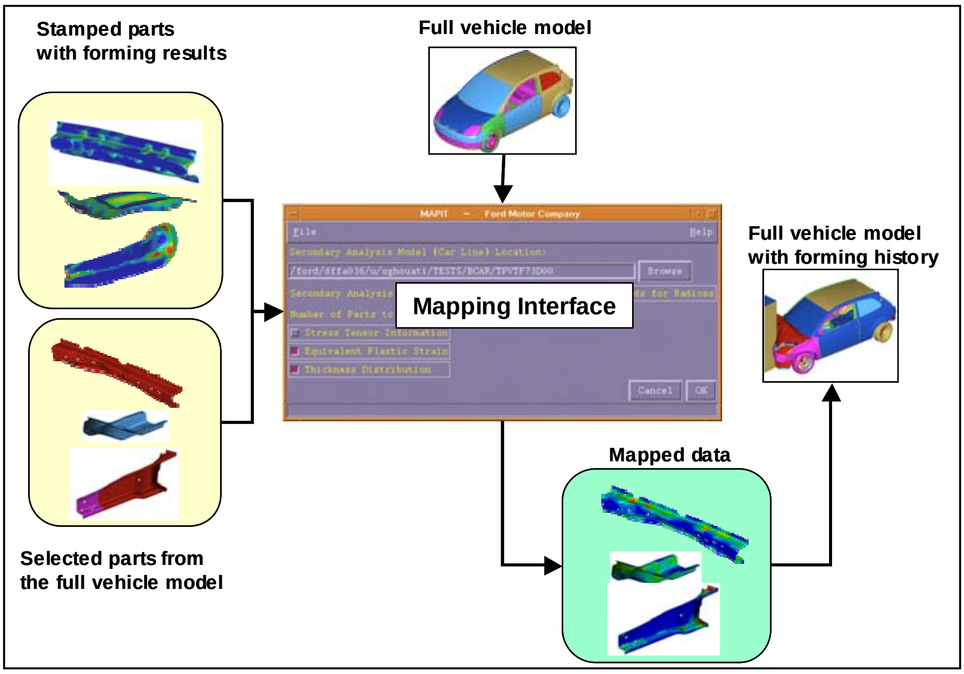


Figure 10: Linking forming and product performance simulations

7 Conclusion

The paper presented the different possibilities to integrate the sheet metal forming simulation in a closed loop design optimization. These possibilities include complex materials identification, springback compensation, part and die design and coupled product and process optimization. This became now possible thanks to the many improvements made to sheet metal forming simulation. As a result of these improvements, the accuracy has reached now an acceptable level making of it a reliable tool to predict both material and geometrical properties of stamped parts.

8 Bibliography

[1] O. Ghouati, J.C. Gelin, "A finite element-based identification method for complex metallic material behaviours", Computational Materials Science, Volume 21, Issue 1, May 2001, Pages 57-68.

[2] P. Picart, O. Ghouati and J.C. Gelin, "Optimization of metal forming process parameters with damage minimization", Journal of Materials Processing Technology, Volumes 80-81, 1 August 1998, Pages 597-601.

.

[3] H. Lanzerath, R. Schilling and O. Ghouati, "Crashsimulation von Umformdaten", CADFEM User's Meeting, Friedrichshafen, September 2000. (In German).

[4] R. Schilling, H. Lanzerath, M. Paas and J. Wesemann, "Numerische Analysen zum Einsatzpotential neuer Werkstoffe in der passiven Sicherheit", VDI Berichte Nr. 1559, Würzburg, September 2000. (In German).

[5] X.M. Chen, J.J. Droulin, D.R. Coopmans, R.D. Dell'Osso and P.J. Belanger, "Stamping and crush performance of Dual Phase steel", (2001), SAE Technical Paper 2001-01-3074.

[6] G. Chiandussi, G. Belingardi, M. Sperati and M. Fariello, "Analysis of the effects of the manufacturing process in the mechanical behaviour of an automotive bumper", (2001), In Proceedings of the 9th International Conference on Sheet Metal, Leuven Belgium.

[7] T. Duton, R. Sturt, P. Richardson and A. Knight, “The effect of forming on automotive crash results”, (2001), SAE Technical Paper 2001-01-3050.

[8] H. Lanzerath, R. Schilling and O. Ghouati, “Crashsimulation in der Fahrzeugentwicklung-Einfluss des Fertigungsprozesses auf die Crasheigenschaften am Beispiel des Pkw-Frontalaufpralls“, Werkstof-Kongress, Januar 2001. (In German).

[9] H. Lanzerath, R. Schilling and O. Ghouati, “Crashsimulation mit Umformdaten”, 10. Fachtagung ”Prozesskette Karosserie“, Fellbach, September 2001. (In German).

[10] J. Shaw and K. Watanabe, “Steel strength and processing effects on impact deformation for a crash energy management component”, (2001), SAE Technical Paper 2001-01-1053.

[11] R. Schilling, H. Lanzerath and O. Ghouati, “Crashsimulation unter Berücksichtigung der Ergebnisse vorangegangener Umformsimulationen“, Tagung “Crashsimulation“, Haus der Technik, März 2001. (In German).

[12] S. Simunovic, J. Shaw and G.A. Aramayo, “Steel processing effects on impact deformation of Ultralight steel auto body”, (2001), SAE Technical Paper 2001-01-1056.

[13] B. Yan, P. Belanger and K. Citrin, “Effect of forming strain on fatigue performance of a mild automotive steel”, (2001), SAE Technical Paper 2001-01-0083

[14] H. Yoshida, A. Uenishi, Y. Kuriyama and M. Takahashi, “Crashworthiness improvement of the side crash by hardening of pre-strained High Strength Steel”, (2001), SAE Technical Paper 2001-01-3112.

[15] D. Zeng, S.D. Liu, V. Makam, S. Shetty, L. Zhang and F. Zweng, “Specifying steel properties and incorporating forming effects in full vehicle impact simulation“, SAE Technical Paper 2002-01-0639.

[16] H. Lanzerath, O. Ghouati, A Bach, “Crash simulation with forming data“, SAE Technical Paper 2006-01-0318.

[17] R. Gao, L. Pan, T Tyan, K Mahadevan, O. Ghouati, H. Lanzerath and M Kessen , “Impact Simulation of Hydro-formed Front End Vehicle Structure“, SAE Technical Paper 2006-01-0312.