Simulation of the forming and assembling process of a sheet metal assembly

A. Govik[1], L. Nilsson[1], A. Andersson[2,3], R. Moshfegh[1, 4]
[1]Linköping University, Division of Solid Mechanics, Linköping, Sweden
[2]Volvo Car Corporation, Olofström, Sweden
[3]Chalmers University of Technology, Department of Product and Production Development, Gothenburg, Sweden
[4]Outokumpu Stainless AB, Avesta, Sweden
alexander.govik@liu.se

ABSTRACT

A sheet metal assembly must meet functional, manufacturing, and sometimes also esthetical requirements. The properties of the assembly are to a large extent affected by the manufacturing process, i.e. the forming processes of the sheet metal components and the subsequent assembling sub-processes. It is of a great industrial interest to be able to predict the properties of the assembly at an early design stage.

This paper presents a methodology, based on Finite Element simulations, which makes it possible to accurately predict the properties of a sheet metal assembly. Each forming process of the individual components is simulated, and all properties affected by the forming process are included in the subsequent simulations of the assembling process. Thus, this methodology makes it possible to optimize both the functional properties of the assembly and also its manufacturing process considering all mechanical effects introduced by the individual manufacturing processes.

A case study of a semi-industrial assembly has been conducted and the simulation results agree well to experimental data.

Keywords: Finite element simulation, Assembly, Forming, Springback, Sheet Metal

1. INTRODUCTION

The use of virtual tools, primarily the Finite Element (FE) Method, in early project phases has the potential to predict and optimize the performance and robustness of the product and its manufacturing process. Early predictions, before physical tools and parts are available, are essential for the ability to evaluate different solutions. Early decisions based on reliable data will reduce late expensive product and process changes. It also leads to a reduced project lead time, amount of cassations, and rework related to quality deviancies in projects and running production.

Today variation analysis is the dominating method to model the assembly process. Deterministic simulations of the whole manufacturing process in order to predict the geometry and other properties of a sheet metal assembly have not previously got that much attention.

Galbraith et al. [1] studied the difference in springback for the individual parts as compared to the springback of the whole assembly of an automotive hood assembly. In the study the assembly process was significantly simplified. After the forming simulation each sub-component was positioned in its assembled location and tied to each other at each spot weld position. Then a springback simulation was performed on the whole assembly, and thus neglecting the deformations due to clamping and welding.

Zhang et al. [2] presented modelling methods for manufacturing simulations of sheet metal assemblies. However deformations due to the welding process and its effect on the material properties are neglected furthermore no experimental validation of the simulations was presented.

In this study the properties of a sheet metal assembly is predicted by sequentially simulating each step of the manufacturing process. The results of each step are transferred to the next simulation step. Thus the evolution of deformations and residual stresses throughout the whole manufacturing process can be predicted and followed. Different sequences of clamping and welding will lead to different deformations and residual stresses in the assembled part. It is possible to vary these sequences in order to find the optimal one with respect to a chosen objective. However, this will be presented in a forthcoming paper.

A case study is presented where an assembly consisting of three parts made of cold rolled DP600 steel is manufactured both experimentally and virtually.

2. EXPERIMENTS

The chosen main sub-component of the assembly was originally designed as a semi-industrial part for validation of FE springback predictions [3]. In order to adapt the existing part for validation of an assembly process, strips were cut out from an original part and

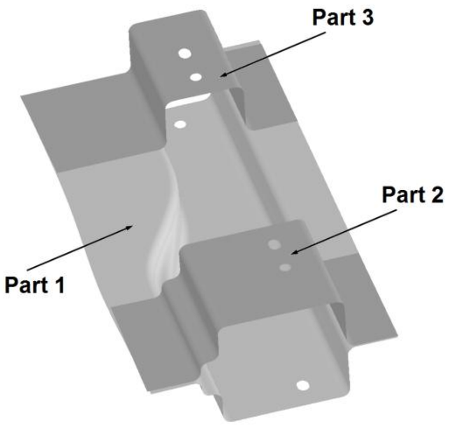


Fig. 1: Assembly

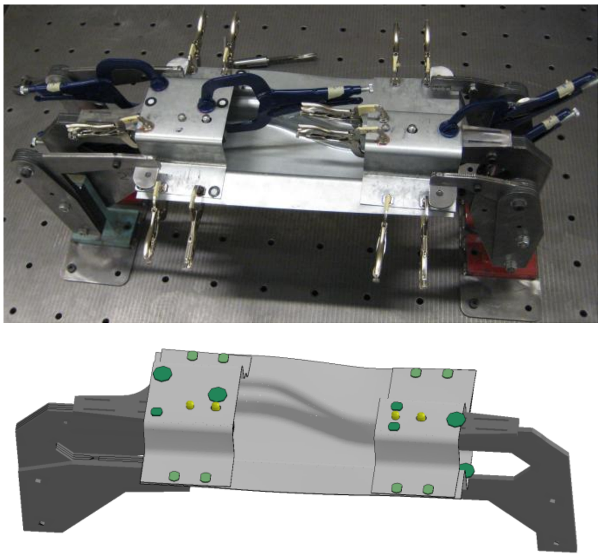


Fig. 2: Components fully clamped in the assembly fixture. Experimental and FE model

extra reference holes for positioning of the strips were included. The strips were then turned 180 degrees and positioned on top of an original part so that the flanges can be welded together, see Fig. 1.

2.1 Stamping procedure

The components are made from 1.4 mm DP600 steel with an initial blank size of 500 x 350 mm with the rolling direction along its short side.

The stamping tool is mounted in a hydraulic single action press and the draw depth is set to 60 mm with a progressive blank-holder force starting at 375 kN and ending at 510 kN. Lubricant is manually distributed over the blank surface. In the tool a device is incorporated that marks positions on the blank in the bottom position of the punch motion. After the part has been removed, a hole or a slot is made at the marked positions.

2.2 Assembly procedure

The assembling process is performed manually in accordance to the PCFR-cycle: Positioning, Clamping, Fastening and Releasing.

The components are positioned in an assembly fixture based on the 3-2-1 locator scheme, i.e. each part is positioned by three vertical supports and the hole and slot are mated with corresponding reference pin. Then the components are clamped to the fixture at each vertical support by C-clamp locking pliers, hence called “clamps”. Thereafter the flanges are clamped to each other with two clamps uniformly positioned on each flange of the parts 2 and 3 according to Fig. 2. It should be noted that the components used in this semi-industrial case study demonstrate much larger springback and clamping forces than normally would have been accepted in a real production process.

Three spot welds per flange are made using a handheld weld gun. Finally the clamps are released, first the clamps on the flanges then the clamps on the fixture.

2.3 Measurement procedure

All measurements are conducted using a coordinate measuring machine (CMM). Points along a number of lines in the width direction are measured to find the cross-section geometries, see Fig. 3b,c.

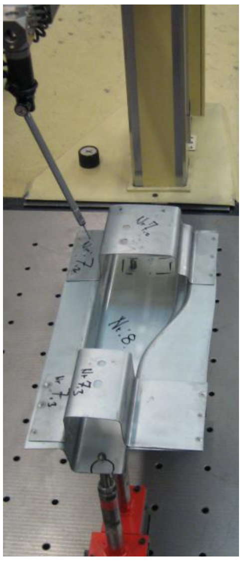
The first measurements are performed on each of the sub-components after they have been positioned and clamped to the fixture. The final geometry of the assembly is measured using a different fixture where the assembly is not clamped, see Fig. 3a.

3. SIMULATIONS

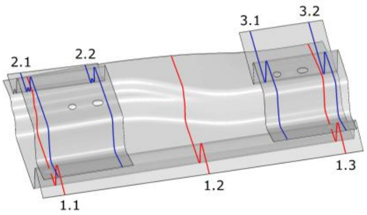
The numerical analyses are performed using the FE code LS-DYNA [4]. After each simulation of a manufacturing step a file is created that contains all essential data for the next simulation step, i.e. geometry data and the stress and strain fields in the components.

Table 1: Mechanical properties of the DP600 steel

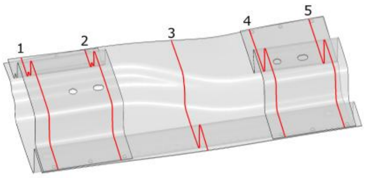
σ_0 (MPa)	σ_{45} (MPa)	σ_{90} (MPa)	σ_b (MPa)	r_0	r_{45}	r_{90}	r_b	Thickness (mm)
386.1	399.7	414.8	405.8	0.76	0.93	0.95	0.99	1.4



(a) Measuring of the assembly



(b) Cross-sections of the components



(c) Cross-sections of the assembly

Fig. 3: Measuring procedures

3.1 Material modelling

In order to achieve accurate results from forming and springback simulations, in particular for components made from AHSS materials, advanced material models are mandatory. In this study a material model based on the Barlat YLD2000 anisotropic yield criterion, [5], together with a mixed isotropic/kinematic hardening law are used. The material parameters needed for this model are obtained from uniaxial tensile tests in three different material directions and from a bulge test, see Table 1. Apart from the equibiaxial yield stress and the equibiaxial r-value, the bulge test also gives the flow curve for high plastic strain levels of the material. This together with the flow curve from the tensile test is utilized to form the stress-strain relation of the material, see Fig. 5, in a similar manner as presented by Sigvant et al. [6].

Furthermore, the degradation of the elastic stiffness due to plastic straining is taken into account in the springback simulation. The importance of this degradation has been shown by Morestin et al. [7].

3.2 Procedure of the simulation process

The procedure of the simulation process is outlined in Fig. 4. It starts with a simulation of the forming process using an explicit methodology and is followed by a springback analysis, including mesh coarsening, using implicit methodology. From one of the formed components, one strip at each end is cut out to form the parts 2 and 3. These cut parts are subjected to further springback, which is handled by an additional springback analysis. The simulation process then proceeds with the assembling phase. The individual components are positioned in the assembly fixture by

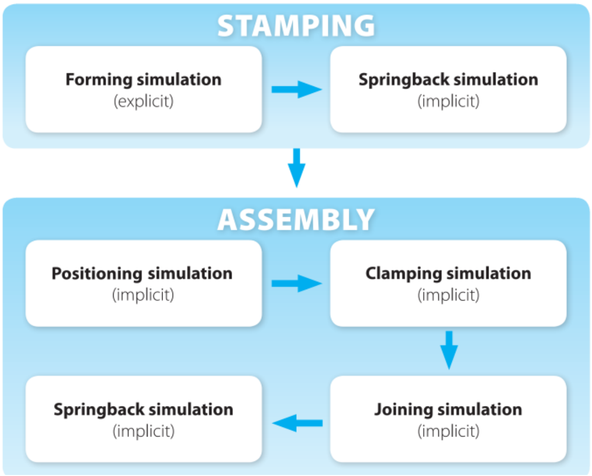


Fig. 4: Procedure of the simulation process

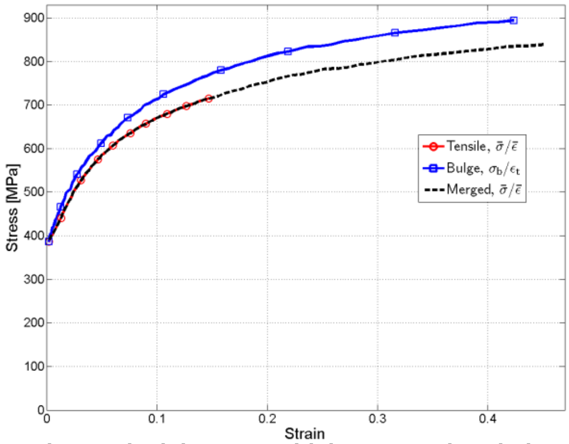
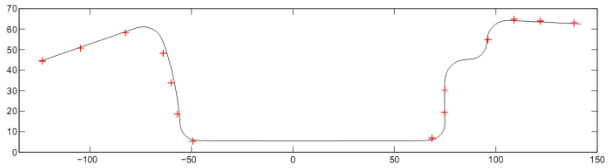
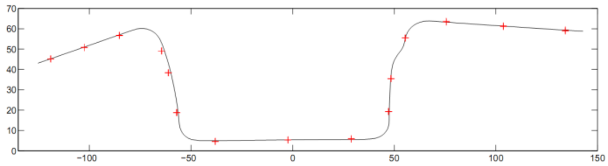


Fig. 5: Biaxial stress - thickness strain relation for the bulge test and effective stress- effective strain relations for the uniaxial tensile test together with the merged curve

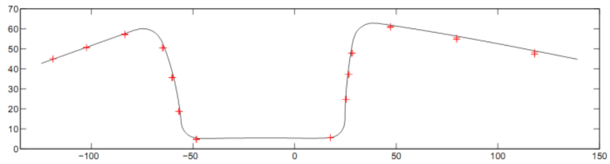
applying gravity loading. The components are first clamped to the fixture, and then their flanges are clamped to each other. The simulation of the clamping process is performed using implicit methodology with contact defined between all parts and with the clamps modelled as rigid discs, see Fig. 2. The clamps on the flanges are applied in the same sequence as was done in the experiments. The simulation of the joining process, in this case spot welding, neglects all effects from heating associated with the process. I.e. the effect from heating and its influence on the material properties around a spot weld is here assumed to be local, but will be investigated in a future study. However, the mechanical aspect of the process is modelled, i.e. the electrode forces are applied on rigid electrodes to ensure that there is contact between the parts at the locations of the spot welds. The spot welds are then modelled using standard features in LS-DYNA and its associated pre and post processor LS-PrePost.



(a) Section 1.1

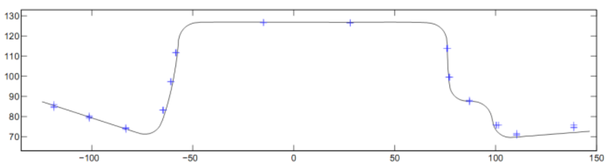


(b) Section 1.2

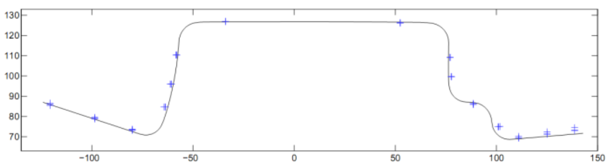


(c) Section 1.3

Fig. 6: Cross-sections of part 1. The measured data from experiments are marked by crosses and the black line denotes simulation results.

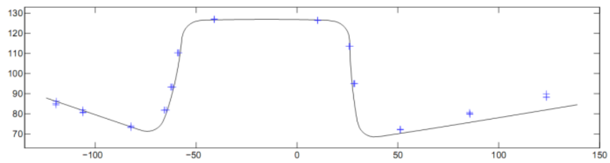


(a) Section 2.1

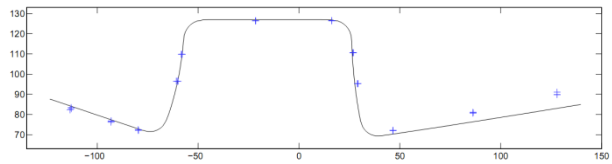


(b) Section 2.2

Fig. 7: Cross-sections of part 2. The measured data from experiments are marked by crosses and the black line denotes simulation results.

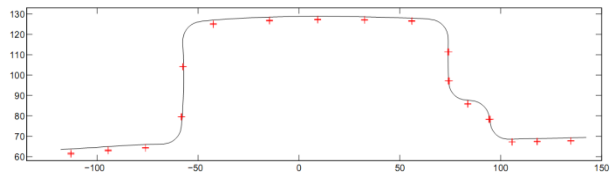


(a) Section 3.1

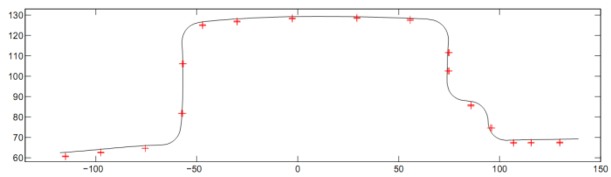


(b) Section 3.2

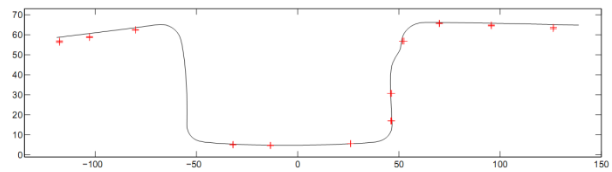
Fig. 8: Cross-sections of part 3. The measured data from experiments are marked by crosses and the black line denotes simulation results.



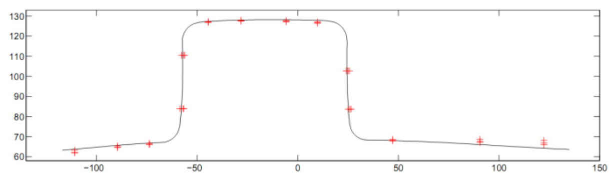
(a) Section 1



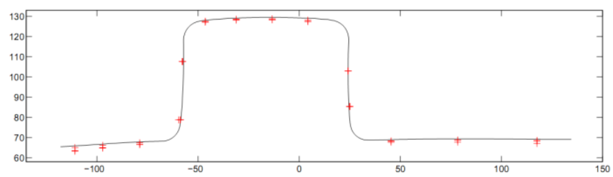
(b) Section 2



(c) Section 3



(d) Section 4



(e) Section 5

Fig. 9: Cross-sections of the assembly. The measured data from experiments are marked by crosses and the black line denotes simulation results.

Table 2: Angles of the components after clamping to the fixture.

Section	Flange		θ	β	α
1.1	Right	Exp.	90.4	87	3.4
		Sim.	89.4	86.6	2.7
	Left	Exp.	103	84.4	18.5
		Sim.	101.6	82.7	18.9
1.3	Right	Exp.	97.2	86.4	10.8
		Sim.	96.2	86	10.2
	Left	Exp.	104.2	84.8	19.3
		Sim.	103.5	83.5	20
2.1	Right	Exp.	94.1	86.7	7.4
		Sim.	92.9	88.3	4.6
	Left	Exp.	102.4	85.2	17.1
		Sim.	101.4	83.3	18.1
3.2	Right	Exp.	96.3	83.8	12.5
		Sim.	93.7	84.5	9.2
	Left	Exp.	102.2	84.6	17.6
		Sim.	102.6	83.8	18.9

Finally all clamps are removed and an analysis of the springback is performed on the assembled structure in order to estimate the final geometry and mechanical properties of the assembled part. For a detailed description of the simulation methods, see [8].

4. RESULTS

In this section the results from the proposed simulation procedure will be validated against experiments. Three assemblies have been produced to give some information about the scatter in the physical processes.

The cross-sections of part 1 after clamping to the fixture, see Fig. 6 and Table 2, show an excellent agreement with experimental data. The simulation results lie within the scatter of the experiments. For parts 2 and 3 the agreement is generally very good but for the right hand side flanges, see Figs. 7, 8 and Table 2. This disagreement may be caused by deformations due to the manual cutting of the experimental parts

The angles in Tables 2 and 3 are extracted in accordance to Fig. 10. The angles are calculated between the outer most experimental data points on each flange/wall using the median value at each point.

The cross-sections of the final assembled structure correlate well to the experimental data, see Fig. 9 and Table 3. One can note that even though the flange angle on the right hand side of part 3, see Section 3.2 in Table 2, lacks in accuracy, the corresponding flange angle on the assembly, see Section 5 in Table 3, is accurate. This can be explained by the fact that part 1 is stiffer than part 3 and thus will govern the deformation behaviour.

Table 3: Angles of the flanges for the final assembly.

Section	Flange		α
1	Right	Exp.	1.1
		Sim.	1.3
	Left	Exp.	4.1
		Sim.	3.4
5	Right	Exp.	0.2
		Sim.	0.2
	Left	Exp.	4.3
		Sim.	3.7

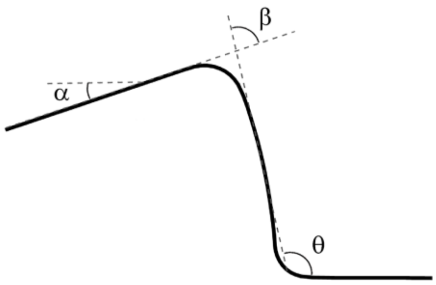


Fig. 10: Measures of springback, where θ is the sidewall angle, β is the flange angle and α is the deviation from nominal geometry.

The angle of the flange on the right hand side in Figs. 9d and 9e is in the first case over predicted and in the later case under predicted, consequently there exist an angle of the flange in the length-direction of the structure which is seen in the simulation result that is not present in the experimental results. This discrepancy could stem from the neglecting of heat effects during welding in the simulation model.

Other possible explanations for the small discrepancies seen can be component and assembly process variations.

5. CONCLUSIONS

In this study the geometry of a sheet metal assembly has been predicted by sequentially simulating each step of the manufacturing process. Evaluation against experiments shows good agreement. However, further studies are needed to analyze the effects of the clamping and spot welding sequences and to optimize the process parameters. Furthermore, variation is a vital part of the production process. The process variation can drastically change the results and must therefore be considered. Further studies are also needed in order to foresee the effect of variations in the involved processes. This would be of great value when choosing the correct process parameters and tolerance settings for the manufacturing of the product and its constituent components.

6. ACKNOWLEDGEMENT

The material characterization and stamping of the components were conducted by Per Thilderkvist and Jörgen Hertzman at the Industrial Development Center in Olofström, Sweden. The work presented in this paper has been carried out with financial support from the Swedish foundation for strategic research, the ProViking programme.

7. REFERENCES

[1] Galbraith, C., Thomas, D. Finn, M.,(2003), Manufacturing simulation of an automotive hood assembly, *4th European LS-DYNA Users Conference,* Ulm, Germany, pp. 37-44.

[2] Zhang, L., Zhu, X., Xu, S., Wang, C.T., (2009), Advanced simulation technology using LS-DYNA for automotive body manufacturing process: From stamping to assembly. *SAE World Congress & Exhibition,* Detroit, USA.

[3] Andersson, A., (2007), Numerical and experimental evaluation of springback in advanced high strength steel, *Journal of Materials Engineering and Performance* **16**, pp. 301-307.

[4] Hallquist, J., (2009), *LS-DYNA Theory Manual*, Livermore Software Technology Corporation, Livermore

[5] Barlat, F., Brem, J.C., Yoon, J.W., Chung, K., Dick, R.E., Lege, D.J., Pourboghrat, F., Choi, S.H., Chu, E., (2003), Plane stress yield function for aluminum alloy sheets - Part 1: Theory*, International Journal of Plas*ticity, **19**, 1-23

[6] Sigvant, M., Mattiasson, K., Vegter, H., Thilderkvist, P., (2009), A viscous pressure bulge test for the determination of a plastic hardening curve and equibiaxial material data, *International Journal of Material Forming*, pp 1-8.

[7] Morestin, F., Boivin, M., (1996), On the necessity of taking into account the variation in the Young modulus with plastic strain in elastic-plastic software, *Nuclear Engineering and Design,* **162 (1)***,* pp. 107-116

[8] Govik, A., Nilsson, L., Moshfegh, R. (2011), paper in preparation.