Inverse Engineering and Preliminary Simulation of a Closed Profile Roll-forming Line

R. Perez-Santiago*, A. García, C. Sarmiento, R. Berlanga, M. Castellanos.
Metalsa S.A. de C.V.
*rogelio.perez@metalsa.com

M. Hernández, P. Zambrano, O. López.
Universidad Autónoma de Nuevo León

Abstract

Roll-forming is a key technology among Metalsa´s manufacturing capabilities. The Company's technology group is conducting research oriented to support future engineering changes and the development of new technology variants. The initial objective of the project is the development of a reliable simulation of one complete roll-forming line using LS-DYNA®. The roll-forming tools were digitized using an optical scanner, converted to 3D parts and finally assembled into the complete forming line. This geometry served as base to generate a finite element model, which was entirely set-up using LS-PrePost®. As a first trial, the elasto-plastic behavior of the sheet strip was modeled utilizing generic material properties. In order to validate the analysis procedure, the production line was halted to allow geometry measurement of cross sections at different forming stations. This article describes the whole procedure utilized along with the comparison of the numeric and physical profiles obtained from the forming process. The numerical model yielded an accurate prediction of the deformed profile. This methodology is used on all virtual validation of new designs before commissioning equipment modification or purchasing.

Introduction

In roll-forming, a sheet metal flat strip is progressively deformed to a specific geometry by the utilization of a sequence of contoured roll pairs [1]. Because of its efficiency, roll-forming is preferred over other processes to form constant section profiles used for electrical fixtures, shelving and roofing sections, racks, doors and windows frames, bicycle wheels, etc. Like other manufacturing processes, roll-forming development is slowly evolving from a base of empirical rules to a more formal engineering activity. Due to its long computational time, as of today it is difficult to implement FEM simulations in the first stages of the design process. However, simulations can be utilized to validate the process engineering before engaging in equipment fabrication or purchasing. Furthermore, the stress state at the end of the process can be utilized to validate the performance of the final part in the case of a safety application.

Metalsa, a subsidiary of Grupo Proeza, manufactures structural components for the light and commercial vehicle markets. Products include chassis frames, body structural stampings and assemblies for passenger cars and light trucks as well as side rails and cross members for Class 5-8 commercial vehicles. The Company currently has presence and operations in Europe, Asia and Australia besides North and South America[1]. Metalsa has roll-formed products for over 15 years. The Company's forming group is leading a technology project oriented to improve the correlation level of the virtual simulation of the process, in collaboration with one prestigious

---

[1] For additional information, please visit www.metalsa.com.

educational institution in Mexico, Universidad Autonoma de Nuevo Leon. The outcomes will allow both the implementation of faster engineering changes in the current production line and the development of next generation technologies, as flexible or variable thickness roll-forming. This paper briefly describes the first stage of the project, which comprised the reverse engineering and simulation of the current roll-forming line.

State of the Art

Han et al. [2] developed a numerical tool based on the finite strip method and applied it to simulate experimental results carried out by Damm [3]. In the experiments, a symmetrical U-channel was formed successfully with three sequential forming steps in a 30°, 60°, and 90° bending sequence. The utilized DINSt-37-2 strip had an initial thickness of 4mm and a width of 236mm. The rolls were considered as rigid, and the cross-section of the work-piece, represented by spline finite strip elements, was deformed by the surface of the upper rolls pressing the lower roll. Sheu [4] proposed a Design of Experiments (DOE) involving four process parameters and three factors to optimize the six stand roll-forming of a 12 mm depth U-Channel. DYNA-3D was adopted to simulate the DOE set-up. The geometry of the strip was modeled with shell elements and its plastic behavior with a power hardening law. The blank was fixed and the rolls were moved toward the blank. Dutton et al., [5] simulated a set of 22 rollers using shell elements for both the strip and rolls. The blank was held stationary while the non-rotating tools moved past it. Friction was neglected as a way to simulate the effect of the rotation. The studied material was the Dual Phase (DP) 1200 in 1.4 mm gauge. The authors claimed that the implicit time integration proved more efficient than the explicit scheme. Lindgren [6] utilized MARC/MENTAT FEM simulations to evaluate the yield strength influence on peak strain and deformation length. Different to the analytical models, the simulation results indicated a definitely influence on both outcomes. The rolls were modeled as rigid rotating surfaces, giving an initial speed to the strip. The strip was modeled with thick shell elements and the elasto-plastic behavior with the von Mises yield surface and isotropic hardening rule. Bui & Ponthot [7] also utilized the experimental forming line of Damm as validation of their numerical procedure. The in-house finite element code Metafor was used for the simulations. Swift isotropic hardening law modeled the sheet material's behavior. The rollers and strip geometry were discretized with rigid analytical surfaces and 8 node continuum elements respectively. The imposed boundary conditions were displacement and symmetry in the mid-plane of the strip. Contact elements, based on the Coulomb friction model, were used to simulate the sheet-roll interface (μ=0.20). The validated methodology was utilized to explore the influence of friction, material properties and some technological parameters like inter-stand distance and velocity in the final geometry and stress/strain distributions. Similarly, Paralikas et al. [8] validated their model using the results of Damm before employing the same procedure to study the influence of the process design in the forming of three different profiles with DP780, 1.2 mm thick, sheet material. In this case the forming results were obtained with LS-DYNA and later transferred to ANSYS for spring-back prediction. The strip material was modeled with integrated shell elements and a mixed (isotropic-kinematic) hardening rule coupled with Cowper-Symonds strain-rate dependent model.

The previous review points out the diversity of numerical choices utilized in the simulation of the roll-forming process. Among the numerical procedures, there is a clear predominance of FEM, based on both implicit and explicit time integration schemes combined with shell and continuum elements. In most of the models, the elasto-plastic behaviour was modeled with a combination of isotropic yield surface and power law hardening rule. Typical boundary conditions are velocity applied to the moving strip with fixed rollers or the opposite approach. In all the cases, symmetry simplification was imposed in combination with frictional or friction-less contact elements among the rollers and sheet.

Aimed to fundamental studies, most works in the literature focused in a relatively low number of stands (4-6). Furthermore, due to the high cost of roll-forming equipment, there is scarcity of experimental data to validate the simulations. As a consequence, the study of Damm [3] has become a validation benchmark. On the other hand, this work was developed in an industrial environment, which demands the simulation of a complete forming line as well as the detailed comparison of experimental and numerical shapes at several forming stations. Preliminary results presented herein include only 15 stands.

Methods

Part Geometry and Material

Figure 1 depicts the studied component. The part is a structural automotive component fabricated with a material similar to the HSLA50. The forming process comprises several rolling stands before passing to the soldering and cutoff stations. Based on the part specifications, the engineering and fabrication of the roll-forming mill was commissioned to a specialized OEM.

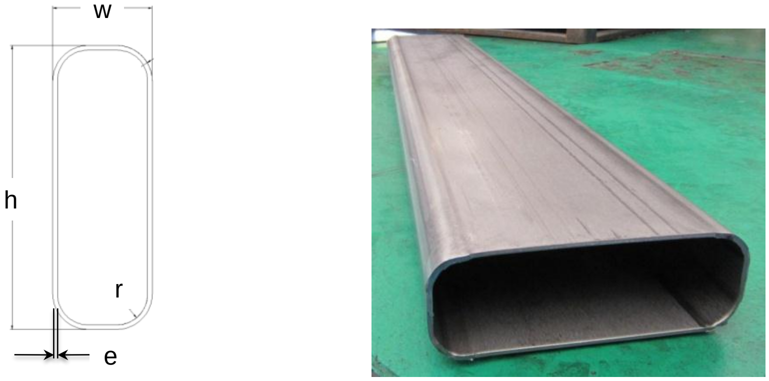

Figure 1. (left) Drawing, not to scale, of the cross section, and (right) picture of the final part.

Reverse Engineering

Like in any FEM simulation, correct representation of the geometry is a key input for the accuracy of the results. Unfortunately, the drawings of the roll-forming mill were not available and could not be obtained from the manufacturer, resulting in the need of conducting a reverse engineering phase. To do this, a rolling stand was disassembled (Figure 2a) and the components digitized using the Konica RANGE 7 (Figure 2b) high accuracy, non-contact 3D digitizer.

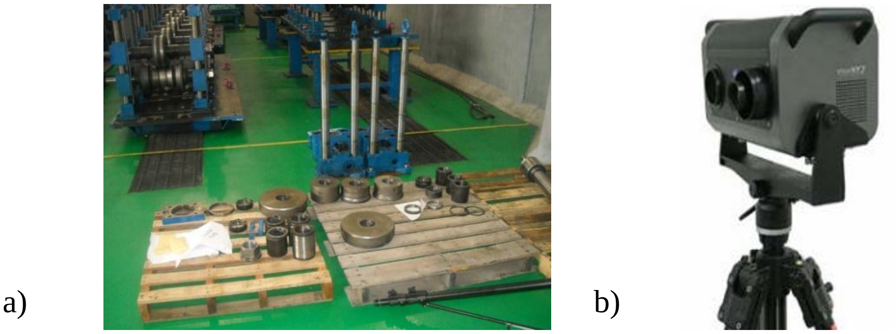

Figure 2. (a) Disassembled stand (b) and the Konica RANGE 7 equipment utilized for the optical measurements.

Generation of the CAD Model for the Basic Stand and Forming Line

With the Konica equipment, each component of one roll forming stand and every roller of the different stations were digitized. The cloud of points obtained with the laser scanner was processed with the software Range Viewer to create the CAD surfaces. The surfaces were then utilized as constructing base to generate solid models in the Solid Works software. Figure 3 presents a bracket component and the solid model generated using the process afore described. The components were then assembled into one complete stand (Figure 4). Finally, the software CATIA was utilized to assemble each set of rollers with the stand instance in order to complete the whole forming line (Figure 5).

Figure 3. Real bracket compared with its digitized 3D model.

Simulation Methodology

LS-DYNA 971, running on a 4 core Windows PC, was utilized to simulate the deformation of the sheet metal in the roll forming line. For the moment, the simulation was carried over just for fifteen stands. The material strip was generated in CATIA®, assembled to the forming line and positioned before the first stand. The sheet was sized to 1500 mm as a way to enforce the contact with three forming stations. The geometry of the whole forming system was imported in the LS-PrePost software and then split up in order to take advantage of symmetry. The forming tools were discretized with default formulation shell elements using a rigid material. An additional mass element was generated in each roller's center of gravity (Fig. 6) and coupled to the shell elements. The mid-surface of the strip material was extracted and modeled using a regular mesh of fully integrated shell elements (*SECTION_SHELL, ELFORM=16) with 5 integration points thickness-wise. The total number of elements and nodes was 56955 and 60257 respectively.

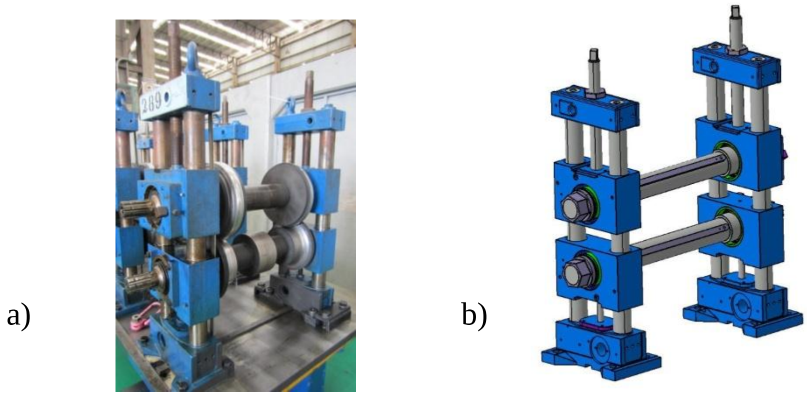

Figure 4. (a) A typical stand and (b) its 3D model after assembling the digitized components.

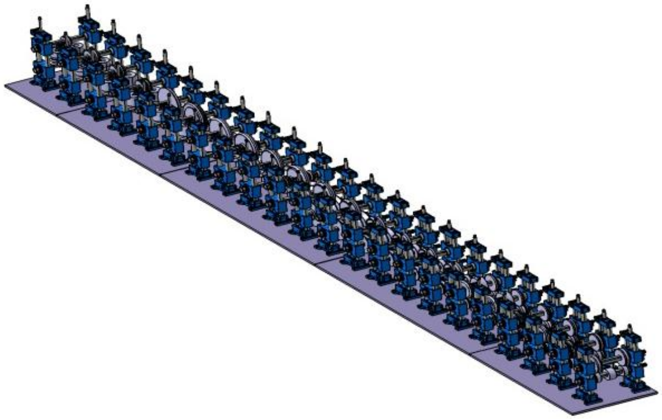

Figure 5. Schematic view of a typical set of roll-forming stands.

The profile is manufactured using an automotive specification steel similar to the HSLA50. Material characterization tests will be conducted in the future. For the moment, the simulation relied on nominal elastic properties of steel combined with the Hollomon's law parameters provided in a Numisheet 2005 benchmark [9]. Therefore, the material properties summarized in Table 1 were utilized within the *MAT_POWER_LAW_PLASTICITY model.

In order to support the blank before entering the first rolling stand, a rigid wall was specified with *RIGIDWALL_PLANAR_FINITE. Symmetry boundary conditions were applied in the nodes at half-width of the strip. Using *PRESCRIBED_MOTION_SET, the velocity utilized in the forming line was imposed on the same nodes. All degrees of freedom of the rollers, excepting the rotation around their axis, were constrained. Frictional contact (μ=0.1) between the tools and the sheet was defined with *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE. As a consequence, the free tools rotated as the sheet passed through them. Finally, mass density was scaled as an aid to decrease the computation time (*CONTROL_TIMESTEP, DT2MS=0.005).

Table 1. Properties of the HSLA50 material model

Young Modulus (GPa)	Poisson's ratio	Density (kg/mm^3)	K (GPa)	n
207.000	0.300	7.850E-6	0.733	0.182

Figure 6. Meshed strip before entering the first forming station. Note the mass element at the center of each roll.

Results

The elapsed solution time of the simulation was 104 hours. Before looking at the forming results, the energy equilibrium of the model was verified. The reference Kinetic Energy (KE) can be computed with Equation 1, where *m* and *V* are respectively the mass and the constant velocity imposed on the modelled strip segment.

$$KE = \frac{1}{2}(mV^2) = 0.694\ [kN - mm] \qquad (1)$$

Figure 7a indicates that the initial numerical KE (0.681 kN-mm) is in agreement with the theoretical value. During the simulation, the KE is not constant but artificially adjusted as a consequence of the mass scaling procedure. The maximum additional mass is around 30% and, as expected (Figure 7b), the ratio of kinetic energy over total energy progressively decreases along the process.

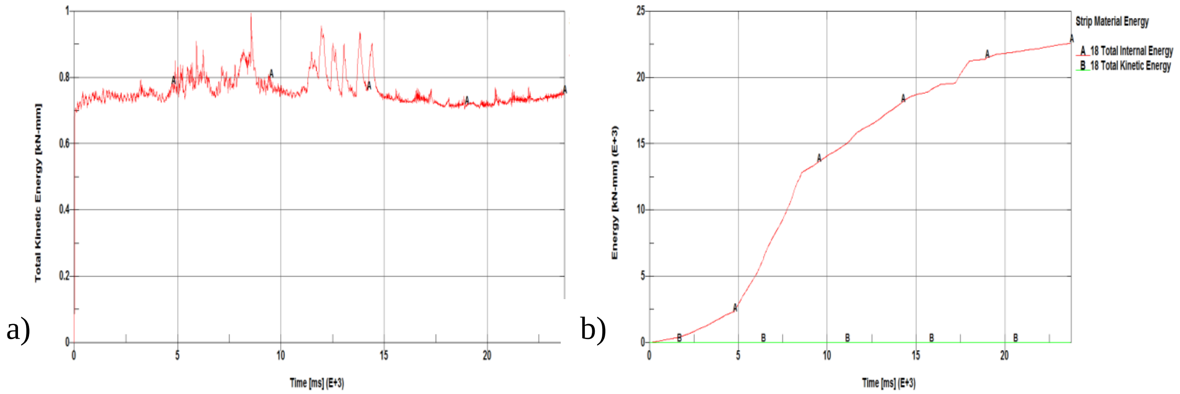

Figure 7. (a) Evolution of the kinetic energy in the strip material, and (b) comparison of kinetic vs. internal energy.

Figure 8a shows the outcome of the 15 stand forming simulation. In order to validate the fitness of the results, actual measurements of the formed profile were taken in the forming line using a FARO® measuring arm. Up to 20 points on the sheet were digitized at different stages of the forming process (Figure 8b).

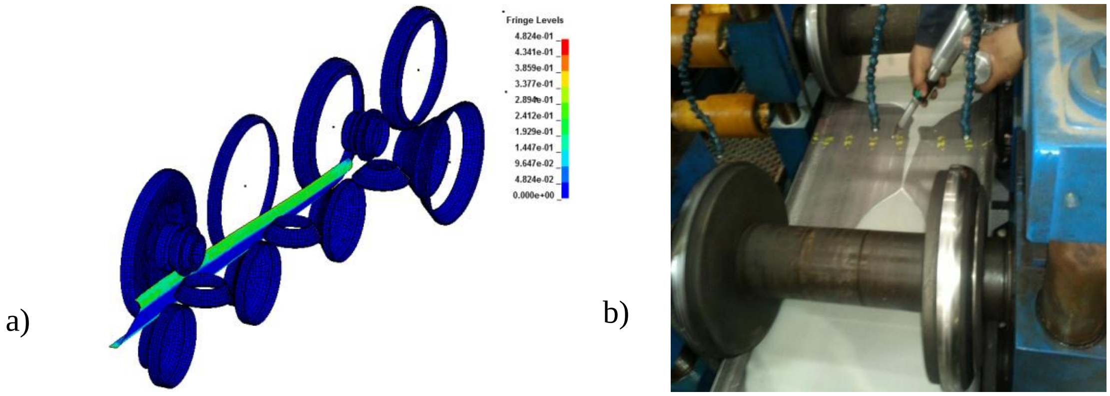

Figure 8. (a) Plastic strain contour of the part at the end of the simulation. (b) Measurement of points at the third stand.

Figure 9 compares the predicted profile to the experimental measurements. Because of space limitations, only stands 7, 9 and 11 are presented. The important offset observed in the stand 7 suggests that the first rollers were not correctly positioned. However, this issue was corrected at the following forming stations.

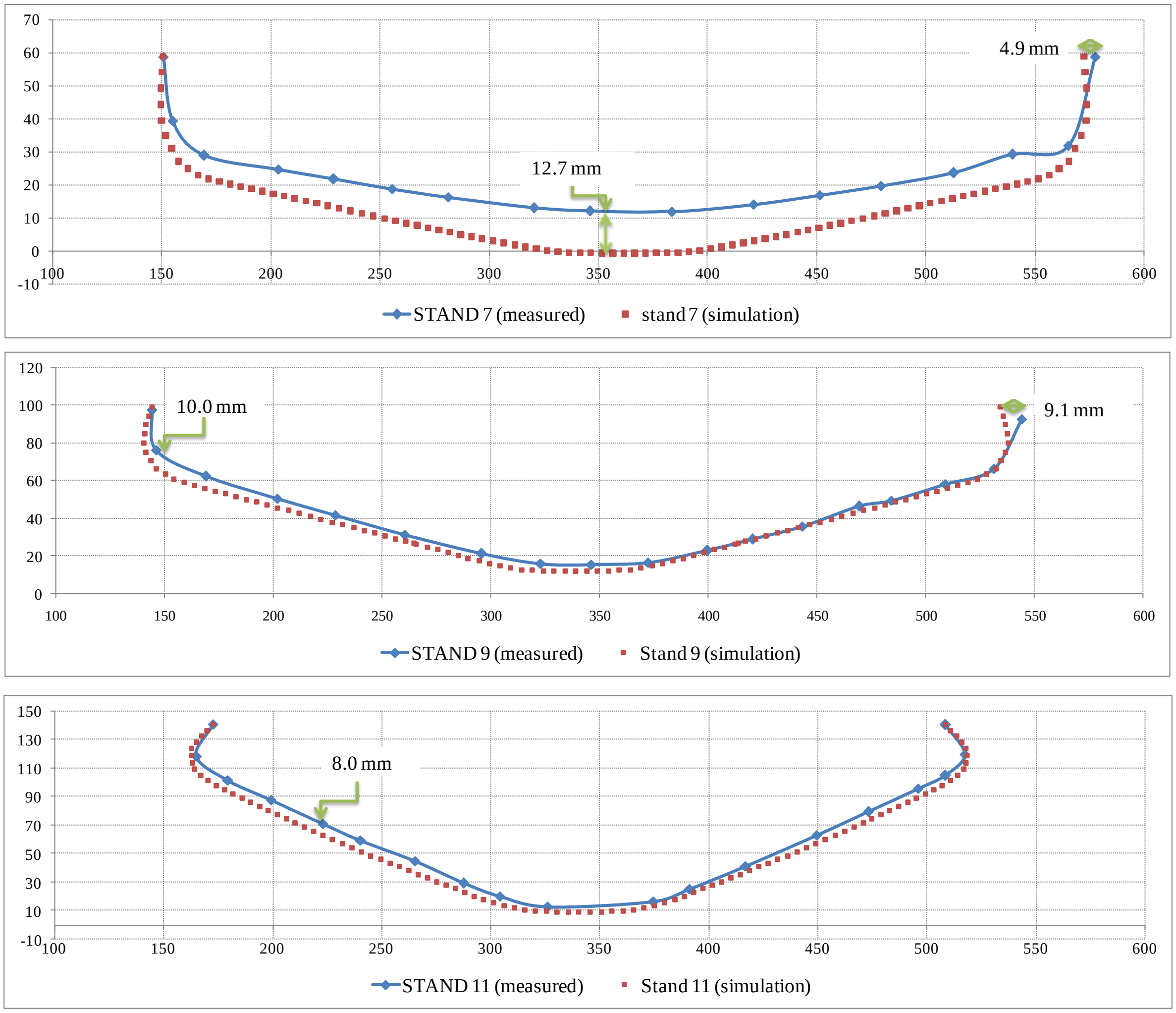

Figure 9. Comparison of numerical (dotted red) vs. experimental (continuous blue) curves of Stands 7, 9 and 11.

The preliminary results indicate that the presented numerical procedure is able to predict the strip deformation along the process. Besides adding the rest of the forming stations, further efforts will be oriented to the: (i) Correct positioning of the rollers, especially in the first stage, (ii) implementation of properties obtained from tensile tests of the actual material, (iii) validation of numerical strains using analytical models and (iv) comparison of thickness distribution in the final part.

References

[1] Halmos, G., 2006. *Roll forming handbook*. Boca Raton: Taylor & Francis.

[2] Han, Z., Liu, C., Lu, W., Ren, Q., 2002. Simulation of a multi-stand roll-forming process for thick channel section. Journal of Materials Processing Technology, 127, pp. 382-387.

[3] Damm, K., 1989. Determination of longitudinal strains in roll forming of standard sections in a multi-stand machine. Dissertation. Institute for Production Technology, University of Sarmstadt, Germany.

[4] Sheu, J., 2004. Simulation and optimization of the cold-roll forming process. AIP Conference Proceedings, 712 (1), p. 452.

[5] Dutton T., Richardson, P., Tomlin, M., Harrison, T., 2006. Simulating the complete forming sequence for a rolled formed automotive bumper beam. 6th European LS-DYNA Users Conference.

[6] Lindgren, M., 2007. Cold roll-forming of a U-channel made of high strength steel. Journal of Materials Processing Technology, 186, pp. 77-81.

[7] Bui, Q., Ponthot, J., 2008. Numerical simulation of cold roll-forming processes. Journal of Materials Processing Technology, 202, pp. 275-282.

[8] Paralikas J., Salonitis, K., Chryssolouris, G., 2011. Investigation of the effect of roll forming pass design on main redundant deformations on profiles from AHSS. Int. J. of Adv. Manuf. Technol, 56, pp. 475-491.

[9] GM Research Labs, 2005. Benchmark 3. NUMISHEET 2005: Proceedings of the 6th International Conference and Workshop on Numerical Simulation of 3D Sheet Metal Forming Process. AIP Conference Proceedings.
http://www.numisheet2005.org/channel_draw.htm