FEA Simulation of Free-Bending – a Preforming Step in the Hydroforming Process Chain

N Beulich\textsuperscript{1}, P Craighero\textsuperscript{2} and W Volk\textsuperscript{3}

\textsuperscript{1}Product, Process Planning Special Technologies Cold Forming, BMW Group, 84122 Dingolfing, Germany
\textsuperscript{2}Technology Methods, Technique Standards, BMW Group, 80788 Munich, Germany
\textsuperscript{3}Chair of Metal Forming and Casting, Technical University of Munich, 85748 Garching, Germany

E-Mail: nikolas.ba.beulich@bmw.de

Abstract

High-strength steel and aluminum alloys are essential for developing innovative, lightly-weighted space frame concepts. The intended design is built from car body parts with high geometrical complexity and reduced material-thickness.

Over the past few years, many complex car body parts have been produced using hydroforming. To increase the accuracy of hydroforming in relation to prospective car concepts, the virtual manufacturing of forming becomes more important. As a part of process digitalization, it is necessary to develop a simulation model for the hydroforming process chain. The preforming of longitudinal welded tubes is therefore implemented by the use of three-dimensional free-bending. This technique is able to reproduce complex deflection curves in combination with innovative low-thickness material design for hydroforming processes.

As a first step to the complete process simulation, the content of this paper deals with the development of a finite element simulation model for the free-bending process with 6 degrees of freedom. A mandrel built from spherical segments connected by a steel rope is located inside of the tube to prevent geometrical instability. Critical parameters for the result of the bending process are therefore evaluated and optimized. The simulation model is verified by surface measurements of a two-dimensional bending test.

1. Introduction

The hydroforming process for hollow profiles allows the production of three-dimensional car body components designed with the highest complexity. To achieve the final part geometry by the use of hydroforming, a few pre and post processing steps are necessary. The hydroforming process chain is shown in Figure 1. To generate a realistic prediction for the producibility of hydroforming parts, the preforming steps are especially important for an accurate Finite Element analysis. This becomes more important when using high-strength steel and aluminum alloys with a low wall thickness. The first forming operation is the roll forming of sheet metals to profiles, followed by a bending and a swage
press operation. The preforming process fits the profile into the hydroforming tool geometry to avoid cutting areas with the tool sides. In addition, the profile ends are prepared for the punch of the hydroforming tool, which applies the internal pressure to the profile. By using the free-bending process as one of the preforming steps, it is possible to produce three-dimensional deflection curves with only one bending operation. This results in lower production costs, reduced cycle time and higher profile complexity.

This paper deals with the simulation of the 6 degrees of freedom (DOF) free-bending process. Therefore, the results and standards of existing research projects are used as a source for the development of the simulation model in LS-Dyna. A model for free-bending with 3 DOF was developed before by Gantner, Bauer et al. in 2005 [1]. Also, simulation methods for mandrels in standard rotary draw-bending operations are often described in literature (e.g. [2, 3]). Beulich [4] summarized and extended these approaches and developed a simplified simulation model for the 5-DOF free-bending. More complex contents, such as weld seam influences and tube wrinkling, are neglected for this paper.

![Figure 1. Overview of the forming steps in the hydroforming process chain](image)

### 2. 6-DOF Free-bending Technology

The free-bending technology for hollow profiles and tubes is a process with six degrees of freedom and therefore comes with a high sensitivity. The first chapter of this paper deals with a detailed description of this bending process to ensure reasonable process know-how for the subsequent chapters. The mechanical design of the bending machine and the mandrel are explained in sections 2.1 and 2.2, followed by the machine kinematics and control.

![Figure 2. Schematic draft of the free-bending machine with 6 DOF](image)

#### 2.1. Machine design

A schematic machine sketch is shown in Figure 2. The design is based on the fixed guide unit and the motor-driven bending die, which has two translational (y-, z-axis) and three rotational (x-, y-, z-axis)
degrees of freedom. The profiles for the bending operation are placed within these tools. Additionally, a mandrel is located inside the profiles with its stiff end located inside of the guide unit. A lubrication block is located behind the guide to reduce friction forces and heat development. During the bending process, the profile is pushed constantly in the positive x direction (translational DOF) and supported by the guide unit. Bending angle, length, radius and direction result from the deflection of the bending die. By adjusting the bending die position, it is possible to generate different uninterrupted bends for the production of complex, three-dimensional deflection curves. The six controlled degrees of freedom ensure high reproducibility as well as short cycle time.

2.2. Mandrel design
In general, the application of mandrels in bending processes is necessary to prevent the profiles from necking and failing during the realization of small bending radii. The mandrel is made up of ten segments and shown in Figure 3a. The end segment (fixed in the guide unit), a long segment (supports the profiles in the area of the guide end) and 8 spherical segments are all connected together by a steel rope that is fixed to the end segments on both sides through clamping and screw connections. The contact surfaces of the mandrel segments are equal for all segments and results of the geometrical intersection of spheres. The reduction of friction forces in the inside of the tube is performed by a lubrication block in the fixed mandrel segment. A more detailed view of the mandrel is shown in Figure 4.

![Figure 3a. Mandrel made up of spherical segments belted by a steel rope.](image)

![Figure 3b. Automatic calculation of translation distance and rotation angle for a specified deflection curve.](image)

2.3. Kinematics
The control program for each deflection curve consists of the pushing velocity, start position of the pusher, position of the mandrel and the time-dependent bending die orientation. This position is characterized by movement curves for all five DOFs. Additionally, it has to be mentioned that each translational movement is paired with an appropriate rotation (e.g. a translation about +y is paired with a rotation about -z). The calculation of translation and rotation is based on the bending data input on the machine control (radius, angle, length, orientation). For each CNC bending program, these control curves are returned in respect to the pusher position and used directly for the Finite-Element analysis.

The generation of control curves for tube profiles without the use of machine control data was developed in Matlab. The approach is based on a geometrical solution of the bending kinematic and uses the pairing of translational and rotational DOF of the bending die. Figure 3b shows a detailed plot of the geometrical calculation principal for a two-dimensional bending profile. The base of the calculation is the output of the guide unit and the movement of the guide in respect to the deflection curve. The input for the calculation is a cloud of points on the deflection curve. The Matlab algorithm calculates the following geometrical situation for each of the points (assuming a perpendicular orientation of the deflection curve by the intersection with guide and bending die):

---

IOP Conf. Series: Journal of Physics: Conf. Series 896 (2017) 012063  doi:10.1088/1742-6596/896/1/012063
1. Position of the guide. 2. Tangent on deflection curve in guide position. 3. Constraint offset in “x-direction” on the tangent for the x-position of the bending die. 4. Perpendicular on tangent in “offset point”. 5. Intersection of perpendicular and deflection curve for the bending die position. 6. Translational orientation is the distance from bending die position to the “offset point”. 7. Tangent on deflection curve in bending die. 8. Rotational orientation results from the intersection of both tangents.

The results calculated with Matlab are compared with the values taken out of the machine control (Table 1). As Gantner, Bauer et al. [1, 6] described in their papers in more detail for the 3-DOF and 5-DOF machines, deviation parameters for translational and rotational values have to be calculated to acquire a relation between the theoretic and the real bending programs. The parameters are affected by tube radius, wall thickness, material properties and lubrication, which especially results in springback behavior. By using these specific correction values, bending programs calculated by an inverse engineering approach can be scaled and used for the simulation of free-bending operations. A second deflection curve was tested with the inverse method with Matlab to verify the calculation. Therefore, the deviation parameter for translation deviates about 17%. For further inverse kinematic calculation, average deviation parameters have to be determined by several different bending tests.

Table 1. Bending die movement (step function) for one bend by a two-dimensional s-shaped deflection curve.

| Simulation time [s] | Translation y-axis [mm] | Rotation z-axis [deg] |
|---------------------|-------------------------|-----------------------|
|                     | Calculated              | Machine control       | Deviation parameter \( P_{ty} \) | Calculated | Machine control | Deviation parameter \( P_{tz} \) |
| 0.000               | 0.000                   | 0.000                 | 0.000                   | 0.000     | 0.000          |
| 0.030               | 0.000                   | 0.000                 | 0.000                   | 0.000     | 0.000          |
| 1.053               | 20.804                  | 30.654                | 1.47                    | 33.289    | 27.742         | 0.834 |
| 1.774               | 20.804                  | 30.654                |                         | 33.289    | 27.742         |       |
| 2.541               | 0.000                   | 0.000                 |                         | 0.000     | 0.000          |       |

3. FEM simulation model

The simulation model is built in LS-Dyna and parameterized to simplify value changes in the following analysis. In chapter 3.1, the general model setup is explained, followed by the mandrel setup and the springback analysis.

3.1. General Model Setup

An overview of the simulation model is shown in Figure 4. Basic settings and parameters are taken from the forming guidelines by Maker and Zhu [7]. LS-Dyna-specific Keywords and theory basics used in this paper are described and explained in [8–10]. The tools (mandrel, guide, bending die) are meshed as shell parts and defined as elements of type 2 – \( Belytschko-Tsay \) without thickness consideration and
MAT_RIGID (MAT_20). The DOFs described above are locked by BOUNDARY_SPC as needed. The profile is meshed with shell elements of type 16 – Fully integrated shell element, 7 thickness integration points and is defined with the desired thickness. The pusher is implemented as a moving node set at the end of the profile by BOUNDARY_PRESCRIBED_MOTION. The bending die orientation is defined by load curves for each DOF by BOUNDARY_PRESCRIBED_MOTION. Contacts between the tools and the tube are assigned as CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE. Therefore, the penalty method (SOFT=0) is used with default values for the contact stiffness and assumed friction values of 0.08 between guide/tube and bending die/tube. The calculation for the bending is performed by the use of the explicit LS-Dyna code, which includes a mass-scaling option with a maximum value of -2e07 to ensure reasonable simulation results. The optimization of simulation time is performed with a sensitivity analysis. Therefore, two-dimensional deflection curves are used to allow for a reflection of the tube in the x-z-plane by BOUNDARY_SPC.

3.1.1. Evaluation of simulation parameters
To ensure an accurate calculation combined with a reduced simulation time, the model is analyzed to receive information about the influence of simulation velocity and element edge length on the bending result. The profile (Figure 9a) is composed of two sharp radii with a bending angle of 90 degrees. The results for the effective plastic strain are plotted for each element along a path on the bending plane. Therefore, the first deflection represents the inside curve followed by the outside curve. The simulation velocity is varied between 200 mm/s (real) and 20000 mm/s (100x). The bending result (Figure 5) shows an equivalence for the first arc and a strong deviation (caused by dynamic effects) for the second arc for a velocity of 20000 mm/s. Because of this, the simulation velocity for future calculations is restricted to a maximum value of 10000 mm/s. The bending analysis for the element edge length comparison was performed by 2000 mm/s (Figure 6). Therefore, the strain by a length of 5 mm diverges significantly from the average strain curve on the lower side and the strain by a length of 2 mm equally on the upper side. For future simulations, a maximum length of 4 mm is recommended and should be refined to 2 mm for more precise results.

![Figure 5](image1.png)

**Figure 5.** Deviation of the effective plastic strain in relation to the simulation velocity and element edge length for a 2D 90-degree arc.

![Figure 6](image2.png)

**Figure 6.** Deviation of the effective plastic strain in relation to the simulation velocity and element edge length for a 2D 90 degree arc.
3.1.2. Material Modelling

Material models for sheet metal forming operations are based on an elastoplastic approach with many additional effects, such as sheet anisotropy, isotropic or kinematic hardening effects, yield loci description or strain rate dependence. The material model MAT PIECEWISE_LINEAR_PLACTICITY (MAT_24) is the most common in LS-Dyna, which describes a plastic material behavior with hardening effects. Approaches that are more complex can be defined by the use of MAT_3-PARAMETER_BARLAT (MAT_36) or MAT_133_YLD2000 (MAT_133), which describes the anisotropy and yield loci description. The basic values used for all material models are the density and the elastic properties, such as Young’s modulus and Poisson’s ratio. The plastic behavior is defined by measured and extrapolated yield curves (tensile test). For the definition of anisotropy or hardening, more extensive material tests are necessary. By using tubes instead of sheets in forming operations, the pre-damage of the tube material caused by the tube production process has to be considered. As summarized by Sorine M. in [11], a reasonable approach to mapping the material properties of tubes in the simulation model is the determination of material properties with a tensile test in the length direction of the tube. Figure 6a shows the yield curves for the aluminum alloy ENAW 5182 evaluated from the sheet in the rolling direction 0° and the laser-welded tube in the length direction 90° moved from the weld seam. Based on the yield curve of the tube, a model for MAT_24 is constructed. Furthermore, the material models of MAT_24, MAT_36 and MAT_133 are derived from sheet tests. The result of the used material model is plotted in Figure 6b for a simple two-dimensional deflection curve. The bending test shows an equality of strain values for the sheet material models with anisotropy (MAT_36, Mat_133) and a small deviation on the outside curve for MAT_24. Compared to sheet materials, the tube material (MAT_24) shows a larger divergence from the calculated strain values.

![Figure 7. Comparison of yield curves for aluminum tubes](image)

![Figure 8. Effect of tube material model on bending result](image)
3.2. Mandrel Model Setup
As described in chapter 2.2., the mandrel is made up of spherical segments. These segments are defined in the simulation as rigid bodies and connected by spherical joints (CONSTRANGED_JIONT_SPHERICAL). The rotation points of the segments are added to the parts by CONSTRANGED_EXTRA_NODES. Information about the joint movement and forces are output by CONSTRANGED_JIONT_STIFFNESS_GENERALIZED. The necessary coordinate systems are therefore defined by DEFINE_COORDINATE_NODES. The steel rope located within the mandrel is modeled as Stolle C & Reid J recommended in their publication Modeling Wire Rope Used in Cable Barrier Systems [12]. Beam elements with a length of 2 mm were created between both ends of the mandrel and fixed to the end segments by CONSTRANGED_EXTRA_NODES. The beams are assigned to the element type Belytschko-Schwer full cross-section integrated and defined with the material MAT_MOMENT_CURVATURE_BEAM (MAT_166) (Parameters see [12]). The contact interaction between beam and mandrel is defined by CONTACT_AUTOMATIC_GENERAL. As recommended in [12], a contact surface for increased contact behavior was created inside the mandrel by copying elements and assigning them to a new part with the material definition MAT_NULL (MAT_009). The contact definition between the mandrel and the tube is defined by CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE, similar to in chapter 3.1. For this contact, a friction coefficient of 0.10 is assumed because of high contact forces and less lubrication.

3.3. Springback
Springback simulation which is often used in sheet metal forming operations, is also necessary for geometry prediction in bending processes [13]. It is used to erase the influence of dynamic effects after a forming simulation caused by elastic material properties and dynamic effects. The dynamic relaxation is performed with the implicit solver of LS-Dyna. To ensure an accurate simulation result, the definition of a static positioning is required (DEFINE_COORDINATE_NODES + BOUNDARY_SPC). The implicit calculation is modeled as recommended in the guidelines for forming springback operations [14, 15].

4. Bending tests
The analysis performed earlier in this paper is based on a two-dimensional profile (Figure 9a), which is built as a 64-mm tube from ENAW 5182 with a wall thickness of 2 mm. Bending tests are performed to verify the simulation model. Therefore, an optical forming analysis of the bending test is performed with the GOM software ARGUS. A dot pattern around the tube is lasered on three positions of the tube and measured before and after the bending process. The deviation of these patterns results in the calculation of e.g. major principal strain. The result of the comparison of measured data with the simulated bending profile is shown in (Figure 9b).

Figure 9a. 2D bending geometry and pattern areas plotted with Major principal strain.

Figure 9b. Comparison of measured bending test and simulation result.
5. Summary and further research
This paper describes the functionality and development of a simulation model for the free-bending process with 6 degrees of freedom as a first step to a complete and accurate hydroforming process simulation. Therefore, a new mandrel design with an inside steel rope was implemented and tested with regard to functionality. After the determination of efficient simulation parameters, the bending test for the verification of the simulation model returns accurate results received by optical measurements.

An important part of the calculation is also the producibility prediction without bending tests or the determination of deviation parameters. The necessary control algorithm was developed in Matlab and outputted single control curves for each DOF. For future applications, this algorithm has to be extended and compared with other inverse engineering approaches to ensure the optimal prediction result. Sensitivity and robustness analysis with different deflection curves and tube materials are recommended to evaluate the control performance.

The effects of different material models on free-bending simulation results were also shown in this paper. For the bending as a stand-alone process, it is recommended to use a material model developed directly out of the tube. Especially for aluminum alloys, anisotropy becomes more important in subsequent forming operations. Therefore, the development of an approach for the creation of anisotropy considering tube material models is considered.

Besides the enhancement of the free-bending control algorithm, it is planned to expand the simulation model to more dimensional friction models and weld seam considerations for bending profiles. To guarantee accurate stress and strain distribution about the pipe circumference, the development of a realistic model for the roll forming process is considered.

References
[1] Gantner P, Bauer H, Harrison D K and Silva A K de 2005 Journal of Materials Processing Technology 167 302–8
[2] Gantner P, Bauer H, Harrison D K and Silva A K de 2004 8th International LS-DYNA® Users Conference
[3] Mentella A, Strano M and Gemignani R 2008 Int J Mater Form 1 165–8
[4] Beulich N 2016 Simulation des Freiformbiegens von Profilrohren im Automobilbau (english translation: Simulation of the Free-bending of tube profiles in the automotive industry) master thesis Chair of Mechanics, University of Leoben
[5] Strasser S 2007 Freiformbiegen von Hohlprofilen (english translation: Free-bending of hollow profiles) master thesis FH Deggendorf
[6] Gantner P, Harrison D K, Silva A K de and Bauer H 2006 Journal of Engineering Manufacture
[7] Maker B and Zhu X 2000 Input Parameters for Metal Forming Simulation using LS-Dyna
[8] Livermore Software Technology Corporation 2016 LS-DYNA KEYWORD USER'S MANUAL VOLUME II R9.0
[9] Livermore Software Technology Corporation 2014 LS-DYNA Theory Manual
[10] Livermore Software Technology Corporation 2016 LS-DYNA KEYWORD USER'S MANUAL VOLUME I R9.0
[11] Sorine M 2007 Formability of advanced high strength shell tubes in tube bending and hydroforming dissertation Mechanical Engineering, University of Waterloo
[12] Stolle C and Reid J 2010 11th International LS-DYNA® Users Conference
[13] Soezen L, Guler M, Goerguelucarslan R and Kaplan E 2010 11th International LS-DYNA® Users Conference
[14] Maker B and Zhu X 2001 Input Parameters for Springback Simulation using LS-DYNA
[15] Maker B and Zhu X 2005 A Procedure For Springback Analysis Using LS-DYNA