Contact Modelling of Large Radius Air Bending with Geometrically Exact Contact Algorithm

V Vorkov[1], A Konyukhov[2], D Vandepitte[1] and J R Duflou[1]

[1] KU Leuven, Department of Mechanical Engineering, Celestijnenlaan 300, 3001, Leuven, Belgium
[2] Karlsruhe Institute of Technology (KIT), Department of Civil Engineering, Geo and Evironmental Sciences, Otto-Ammann-Platz 9, 76131, Karlsruhe, Germany

E-mail: vitalii.vorkov@kuleuven.be

Abstract. Usage of high-strength steels in conventional air bending is restricted due to limited bendability of these metals. Large-radius punches provide a typical approach for decreasing deformations during the bending process. However, as deflection progresses the loading scheme changes gradually. Therefore, modelling of the contact interaction is essential for an accurate description of the loading scheme. In the current contribution, the authors implemented a plane frictional contact element based on the penalty method. The geometrically exact contact algorithm is used for the penetration determination. The implementation is done using the OOFEM – open source finite element solver. In order to verify the simulation results, experiments have been conducted on a bending press brake for 4 mm Weldox 1300 with a punch radius of 30 mm and a die opening of 80 mm. The maximum error for the springback calculation is 0.87° for the bending angle of 144°. The contact interaction is a crucial part of large radius bending simulation and the implementation leads to a reliable solution for the springback angle.

1. Introduction

Air bending remains one of the most popular production techniques in sheet metal forming. Recently, with the introduction of high-strength steels with limited bendability, the problem of cracking due to exceedance of deformation limits has become more pronounced. Therefore, the usage of punches with a large radius is a typical approach for the deformation reduction. The bending process with punches of big radii is dissimilar to conventional or small radius air bending. Conventional bending can be considered as a process where the loading scheme remains unchanged throughout the forming cycle and can be described as a traditional 3-point bending mechanism. In contrast with this observation, the loading scheme for large radius bending changes from 3-point bending at the onset to 4-point bending at the end of the forming cycle. Moreover, the loading scheme during the 4-point bending phase depends on the bending angle. The contact points between punch and plate move from the tip of the punch towards the die edges as the forming angle increases [1,2].

The constant change of the loading scheme poses a significant obstacle for the development of analytical models. Finite element analysis with contact interaction is a convenient way for process simulation that allows taking into account the interaction between the tool and the plate. Despite the computational power of modern computers, 3D models still require a significant amount of calculation time, particularly when contact interaction is taken into account. Thus, model reduction is advisable to keep the calculation time short, especially if the aim is to use the solution for adaptive control of forming.

Most of the commercial finite element codes use the master-slave approach for the implementation of the contact interaction [3]. In this approach one of the contacting bodies is represented as the master surface and the other contacting body is considered as the cloud of slave nodes. An efficient approach for determination of the closest distance between the master surface and slave nodes is the geometrically exact contact algorithm, which calculates the amount of contact interaction between two bodies, namely a penetration [4].

The current contribution focuses on the implementation of the frictional plane strain contact element with the geometrically exact contact algorithm as an approach for the penetration calculation.

2. Experimental procedure

Large-radius bending experiments have been conducted on a CNC press brake with a capacity of 50 tonnes. Tooling and plate dimensions are shown in **Figure 1a.** For all plates the width is 50 mm. Bending tests were performed on the high strength steel Weldox 1300. Material parameters and a detailed description of the experimental procedure can be found in reference [5].

The forming process has been recorded by a high-speed camera set-up mounted on the ram of the press brake. Videos of the bending experiments have been processed by WinTopo freeware to retrieve the formed plate profile that is subsequently measured by BricsCAD. Two bending parameters have been measured: (1) the springback angle, which is calculated as the difference between the product and the bend angle (see **Figure 1b**) and (2) the bend allowance, which is defined as the difference between the initial plate length and the sum of flange lengths for the bent part (see **Figure 1c**).

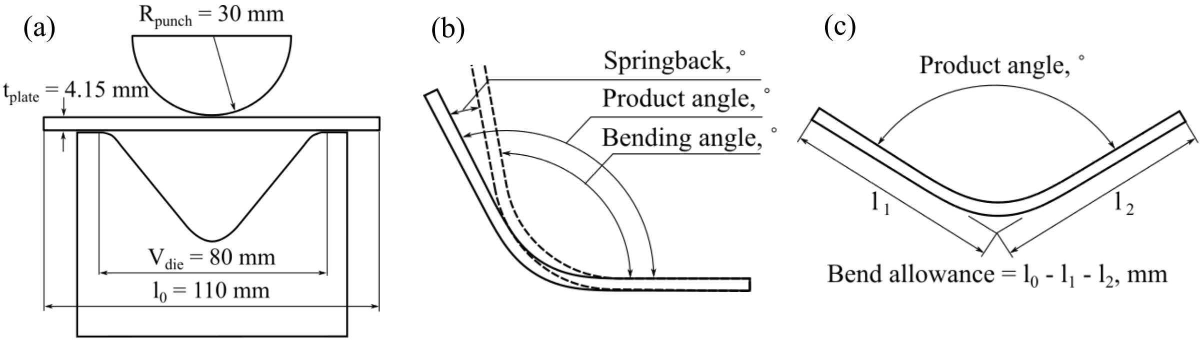

Figure 1. (a) Scheme of the experimental procedure; (b) angles and (c) bend allowance definitions.

3. Finite element approach

For finite element analysis two solutions were used: a solution based on the OOFEM open source finite element code [6]; and a model implemented in the commercial finite element code Abaqus. For the sake of comparison, the mesh and material parameters, and the friction coefficient are the same for both models. Linear elements with plane strain formulation were utilized since this situation is dominant during the bending process [7].

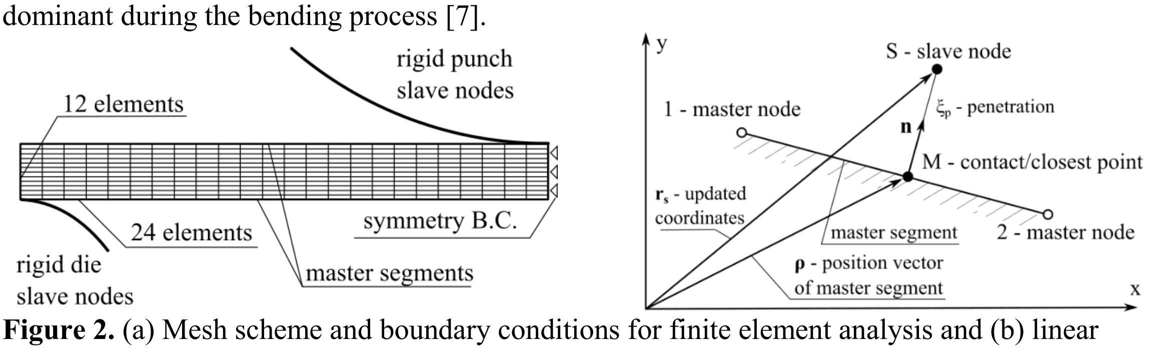

Figure 2. (a) Mesh scheme and boundary conditions for finite element analysis and (b) linear Node-To-Segment contact element geometry: 2 master nodes and 1 slave node.

The interaction between plate and tooling was modelled by means of the contact interaction with a friction coefficient of 0.3. This friction coefficient value is identified from measurement results obtained by Aerens [7] on coarse and fine materials. The contact surfaces of the forming tools (punch and die) in the plane model are represented by absolutely rigid lines. The mesh and boundary conditions are presented in **Figure 2a**. Due to the symmetry of the forming process, only half of the model has been simulated. Isotropic hardening with piecewise approximation of the stress strain curve [5] has been used for modelling the material behaviour.

3.1. OOFEM model

Plane strain elements Quad1PlaneStrain have been taken from the OOFEM element library – an element with the total Lagrangian formulation using the first Piola-Kirchoff stress and the deformation gradient. The solution has been divided into two parts along the deformation cycle: the forming phase and the springback. Both solution steps have been solved using an implicit Static Structural solver.

Despite the fact that the OOFEM library contains a large number of element types, contact elements are not available. Therefore a penalty Node-To-Segment contact element has been implemented according to the description by Konyukhov and Izi [3] with the penetration calculation implemented through the geometrically exact contact algorithm. **Figure 2b** shows the scheme of the Node-To-Segment linear contact element which is defined by two master nodes 1 and 2, and a slave node S.

3.2. Geometrically exact contact algorithm

Closest point projection is a differential geometry procedure which is formulated as the extremal problem for the determination of the shortest distance between a point and a line segment (see **Equation 1**). More information about the closest point projection procedure in a covariant form can be found in [3].

$$F(\xi) = \frac{1}{2}\|r - \rho(\xi)\| \tag{1}$$

According to the geometrically exact contact algorithm for the implemented contact element the value of the penetration is calculated in the direction of the normal vector of the element using **Equation 2** (see **Figure 2b**), where: ξ_p – penetration, $\mathbf{r}_s$ – updated coordinates, $\boldsymbol{\rho}$ – position vector of master segment, and $\mathbf{n}$ – normal vector. For the penetration the following notation is used: $\xi_p > 0$ – no contact; $\xi_p = 0$ – slave point S on contact boundary; $\xi_p < 0$ – contact with penetration.

$$\xi_p = (\boldsymbol{r_s} - \boldsymbol{\rho}) \cdot \boldsymbol{n} \tag{2}$$

3.3. Abaqus model

For the Abaqus model 4-node plane strain CPE4R plane elements were selected. The forming process in Abaqus is again split into two solution steps: the implicit solver has been used in both the forming phase and the springback phase. The mesh and material parameters as well as the friction coefficient are kept the same as in the OOFEM solution.

4. Results and discussion

For the comparison of results the springback and the bend allowance have been selected as the representative parameters for the final shape of the bent plate. **Figure 1b** and **c** show the definition of these parameters. **Figure 3** compares experimental results with OOFEM and Abaqus finite element models. Since the springback depends on the bending angle (see **Figure 1b**) and the bend allowance corresponds to the product angle (**Figure 1c),** appropriate angles are depicted in the graph.

Both finite element models predict the springback angles, with an expected trend: springback increases with increasing bending angle. When compared to experiments, the maximum error for the springback is 0.87° for OOFEM, and 1.45° for Abaqus, for the bending angle of 144°.

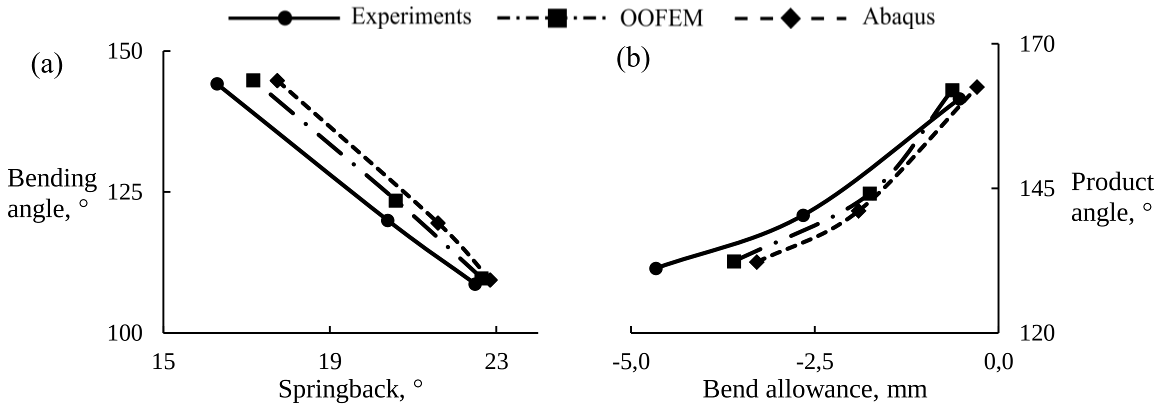

Figure 3. Comparison of experiments and finite element analyses for (a) the springback angle and (b) the bend allowance.

The bend allowance is characteristic of the final shape of the bent part, which allows to calculate the initial length of a blank before the bending process (see **Figure 1c**). This parameter primarily depends on the final or product angle and on the accuracy of the springback prediction, hence the initial error in the springback calculation leads to a significant error in the bend allowance. The maximum error for the bend allowance (the product angle of 131°) is 1.06mm for OOFEM and 1.37mm for Abaqus. The significant errors for the bend allowance mean that the prediction of the bent curvature is not accurate enough for the plane model, since this model cannot handle 3D effects, such as the anticlastic effect.

5. Conclusion

In the current contribution the 2D penalty Node-To-Segment contact element with friction has been implemented within the OOFEM environment. The penetration between the master segment and slave nodes has been calculated by means of the geometrically exact contact algorithm. The obtained solution for large-radius bending was is compared to the solution obtained by the commercial finite element code Abaqus and also to experimental data. The plane finite element model provides an appropriate estimation for the springback angle.

6. Acknowledgements

The authors would like to thank Mikael Öhman from KTH Royal Institute of Technology, Sweden for his help in the implementation of the contact algorithm using OOFEM environment.

References

[1] Vorkov V, Aerens R, Vandepitte D, Duflou JR. On the Identification of a Loading Scheme in Large Radius Air Bending. Key Engineering Materials, vol. 639, Trans Tech Publ; 2015, p. 155–162.
[2] Vorkov V, Aerens R, Vandepitte D, Duflou JR. The Multi-Breakage Phenomenon in Air Bending Process. Key Engineering Materials, vol. 611, Trans Tech Publ; 2014, p. 1047–1053.
[3] Konyukhov A, Izi R. Introduction to computational contact mechanics: a geometrical approach. John Wiley & Sons; 2015.
[4] Schweizerhof K, Konyukhov A. Covariant description for frictional contact problems. Computational mechanics 2005;35:190–213.
[5] Vorkov V, Aerens R, Vandepitte D, Duflou JR. Springback prediction of high-strength steels in large radius air bending using finite element modeling approach. Procedia Engineering 2014;81:1005–1010.
[6] Patzák B. OOFEM—an object-oriented simulation tool for advanced modeling of materials and structures. Acta Polytechnica 2012;52.
[7] Aerens R. Le pliage en l'air. CRIF, Section Construction Mécanique, MC110; 2000.