ADVANCED MECHANICAL SIMULATION MODELS FOR AUTOMATIC PANEL BENDERS

FRANZ HAMMELMÜLLER*, CHRISTIAN ZEHETNER*, ASTRID S. PECHSTEIN†, WOLFGANG KUNZE+, HELMUT J. HOLL† AND HANS IRSCHIK†

* Austrian Center of Competence in Mechatronics (ACCM)
Altenbergerstrasse 69, 4040 Linz, Austria
e-mail: franz.hammelmueller@accm.co.at, web page: www.accm.eu

† Institute for Technical Mechanics, Johannes Kepler University of Linz,
Altenbergerstrasse 69, 4040 Linz, Austria
e-mail: astrid.pechstein@jku.at, web page: www.jku.at/tmech

+ Salvagnini Maschinenbau GmbH,
Dr. Guido Salvagnini-Strasse 1, 4482 Ennsdorf, Austria
e-mail: wolfgang.kunze@salvagninigroup.com, web page: www.salvagninigroup.com

Key words: Panel Bender, Bending Process, Finite Element Simulation, Sheet Metal, Elasto-plasticity.

Abstract. With automatic panel benders complete products are manufactured from sheet metal. In order to achieve short cycle times with high flexibility, a deep insight into the non-linear bending process is required. For this reason, efficient mechanical simulation models have been implemented, combining Finite Element Method, multibody dynamics simulation tools, contact mechanics algorithms and substructuring. Scope of this work is the comparison of several simulation models with measurement results performed on a Salvagnini P4XeD automatic panel bender.

1 INTRODUCTION

In automatic panel benders typically sheet metals are processed which are very thin with respect to length and width direction, such that the bending process can be modelled by a plane state of strain with very good accuracy. Therefore a two-dimensional (2D) Finite Element model has been derived, which considers the essential parameters of the bending process.

Due to the construction of the machine there are also influences in longitudinal direction of the panels. These effects are modelled by means of three-dimensional (3D) Finite elements. With the three-dimensional model the whole machine can be simulated with high accuracy but with a large numerical effort. In the sense of sub-structuring techniques, computational results of the 3D model have been used in order to adapt the elastic stiffness of the 2D-model.

Analysing the 3D-simulation results it turned out that the cross-sections of the panel approximately remains plane even for bending of short sides compared to thickness. Thus, a shear deformable beam element has been applied in order to obtain an efficient simulation that considers the most important parameters.

Finally, the simulation results are compared to measurements performed on a Salvagnini P4XeD automatic panel bender. For further information see www.salvagninigroup.com and Kunze et al. [1].

2 GENERAL MODELING OF A PANEL BENDER

The fundamental functionality of a panel bender is schematically shown in Figure 1, cf. Zehetner et al. [2]. The sheet is clamped horizontally between two clamping tools (counterblade and blankholder tool), and bent by the movement of the bending tool. The figure shows the cross section. The dimension out of the plane is denoted as width.

The following nonlinear effects must be taken into account in the model of this forming process:

- Non-linear material behavior in the sheet metal (elasto-plasticity)
- Large (non-linear) strains and large rigid body motions
- Complex contact problems with friction (also bearings, hydraulic cylinders, etc. which are modeled by connector-elements)

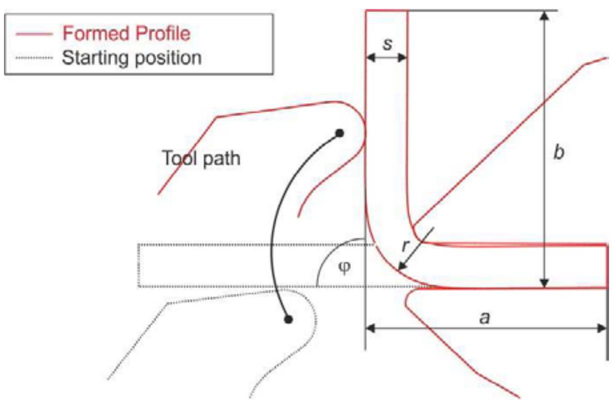

Figure 1: Functional principle of a panel bender

In chapter 3 and 4 a 2D and a 3D Finite Element model are presented, both have been implemented using the software package ABAQUS 6.12-1. In chapter 4, a beam element is presented which has been implemented in the software code HOTINT, cf. Gerstmayr et al. [5].

3 PLANE FINITE ELEMENT SIMULATION

3.1 Modeling

The width of the sheet metal is much larger than the thickness, therefore the bending process can be modeled by plane strain elements with high accuracy. The model is shown in Figure 2. Thickness variations in the geometry of the parts of the machine frame can be considered by adapting the plane thickness of the particular section.

The contact interactions of machine parts and the sheet metal are represented by different contact models like surface to surface contact or node to surface contact with the respective contact properties. If the parts are connected, constrained conditions like the so called tie contact are used. Hydraulic cylinders and bearings are approximated with connector elements. The stiffness of the bearings are considered in these connector-elements. For modelling see also Holl et.al.[3].

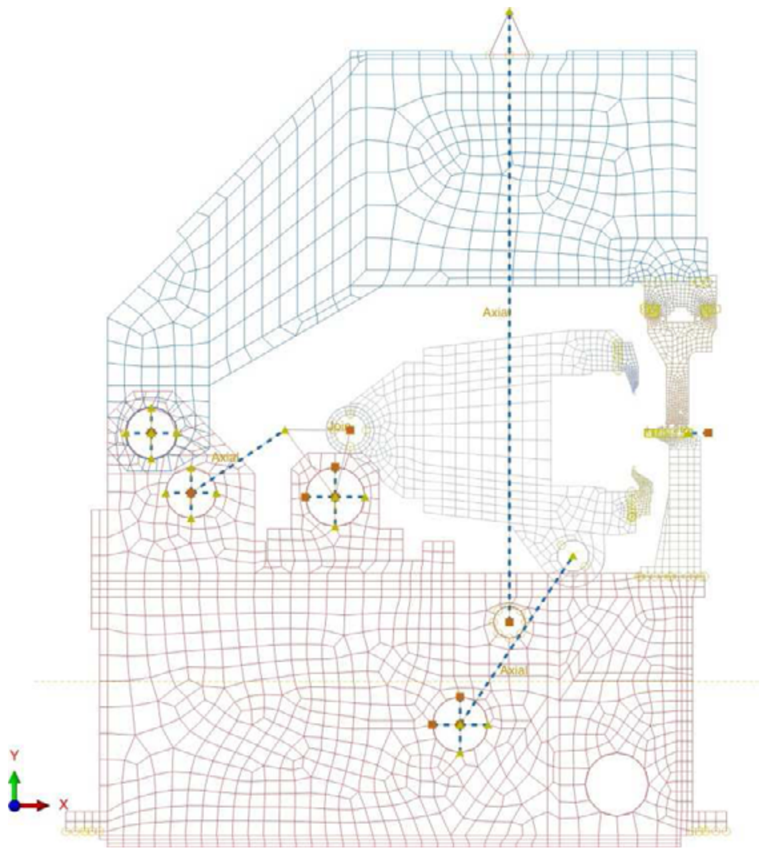

Figure 2: 2D FEM model

The mesh is generated with linear four node elements. Finally this two - dimensional FEM model has a total number of only 13684 variables, such that the computation time for a whole bending process with the software package ABAQUS is t_C = 305s (about 5min) using only one central processing unit (CPU) on a Standard Dual Core PC with 8 GB RAM and a clock rate of 3.16 GHz.

3.2 Simulation Results

Exemplary simulation results are shown in Figure 3 and Figure 4. Figure 3 shows the v. Mises stress for two bending angles $\varphi = 60°$ and $\varphi = 135°$. In the upper clamping tool (blankholder) the stress distribution is discontinuous because of differences in the element thickness in the respective regions of the part. This modelling strategy has been chosen in order to consider the specific three-dimensional shape of that part in the 2D model.

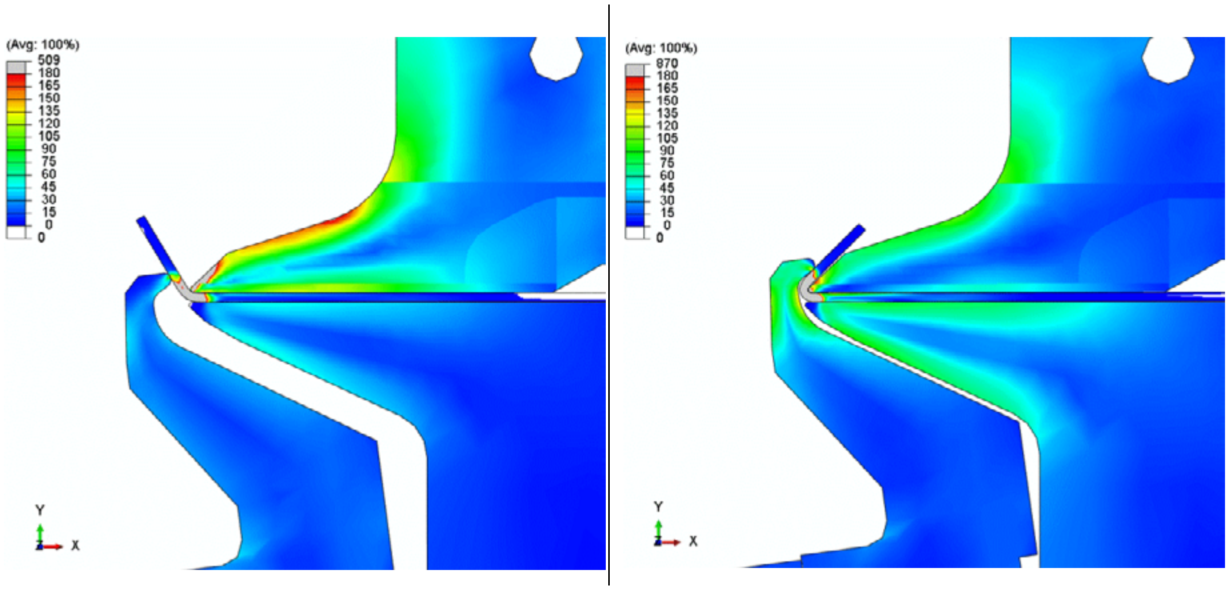

Figure 3: Comparison stress in sheet and tools, 2D FEM model

Figure 4 shows the bending force per sheet width, which is the resulting contact force between sheet metal and bending tool, in relation to the bending angle of the sheet. The increase of the force for bending angles larger than $\varphi = 100°$ results from a change in the contact situation: the sheet additionally gets into contact with a contact surface on the upper clamping tool.

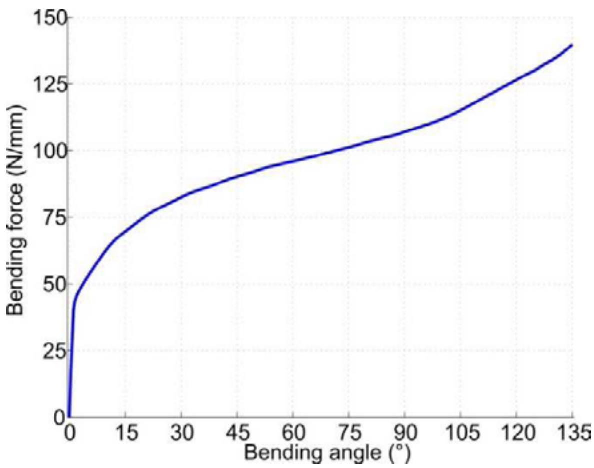

Figure 4: Bending force, 2D FEM model

A comparison with measurement results shows that the most important process parameters are included with very good accuracy in the plane finite element model.

4 THREE – DIMENSIONAL FINITE ELEMENT SIMULATION

For further refinement, three–dimensional simulation models have been developed, with which effects in the longitudinal direction of the machine are taken into account. This includes: Detailed models of the machine parts, longitudinal warping of the sheet, crowned bending and clamping tools, detailed modeling of the components of the machine frame as welded construction, etc. see Zehetner et al. [3].

4.1 Modeling

The machine frame parts are modeled with linear four node shell elements in order to reduce the number of elements, and furthermore for reducing the computation time. The mesh in all other parts, like the clamping tools, the bending tools and so on, is generated with linear 3D solid elements, mainly hexahedron and (at geometrically more complex regions) tetrahedron elements.

Also in the three–dimensional model the different parts are interacting by using different contact models, which have been optimized in order to achieve good convergency.

If two different parts are connected, constrained conditions like tie contact are used. For considering hydraulic cylinders and bearings connector elements have been used. The connection between the geometry and the connector is realised with so called reference points.

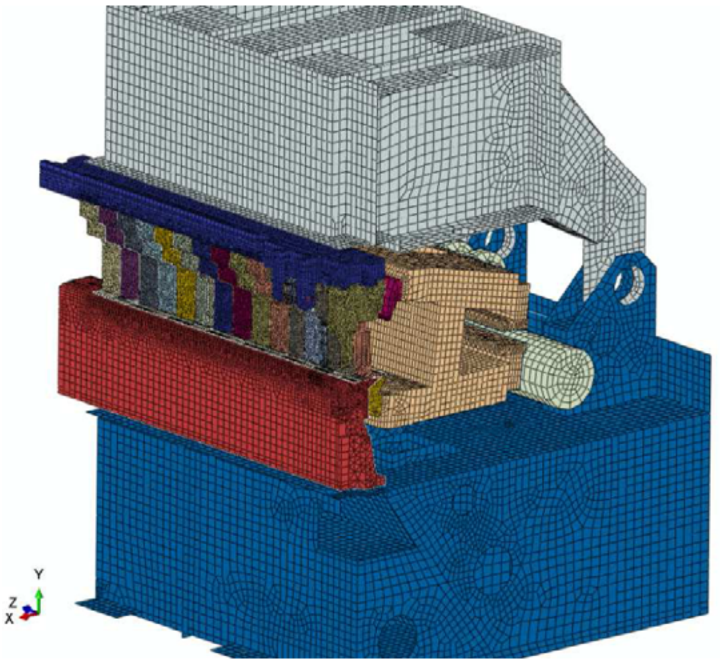

Figure 5: 3D FEM model

This complete three-dimensional FEM model has a total number of 5117118 variables, and finally we got a computation time with ABAQUS of t_C = 410153s (about 114h) using eight CPUs on a Workstation with 8 CPUs, 96 GB RAM and a clock rate of 2.8 GHz.

4.2 Simulation Results

The stresses in bending and clamping tools which occur in the bending procedure are shown in Figure 6 for an exemplary selected bend for two bending angles $\varphi = 60°$ and $\varphi = 135°$. The bending parameters like sheet metal thickness s, tool path and clamping force etc. are the same as in the two-dimensional ABAQUS model in chapter 3.

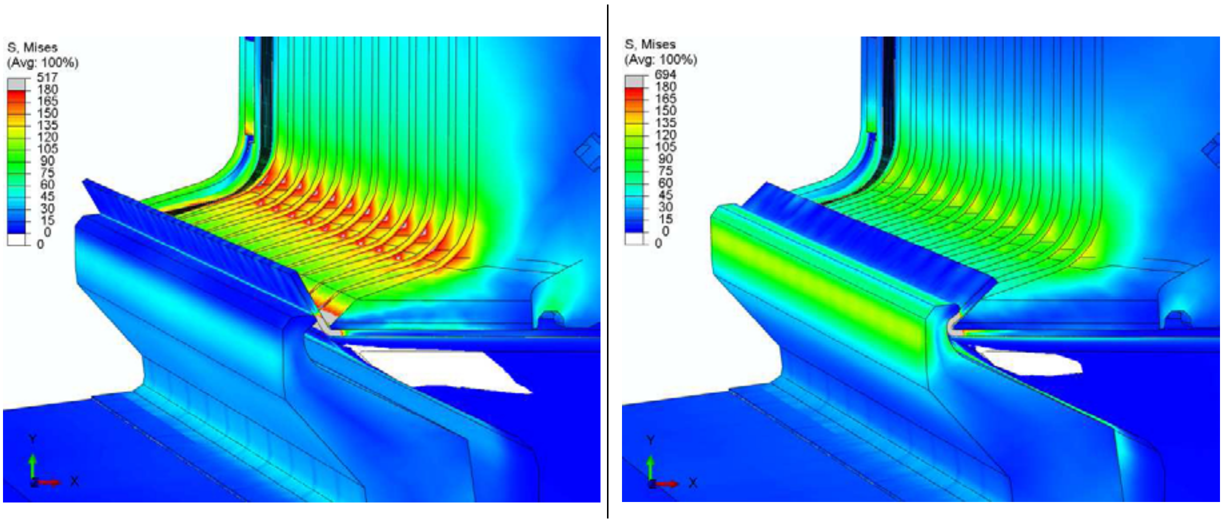

Figure 6: Comparison stress in sheet and tools, 3D FEM model

Figure 7 shows the bending force per sheet width for the three-dimensional (3D) model compared to the 2D results from Figure 4. It can be seen that the results of both simulation models show a very good coincidence.

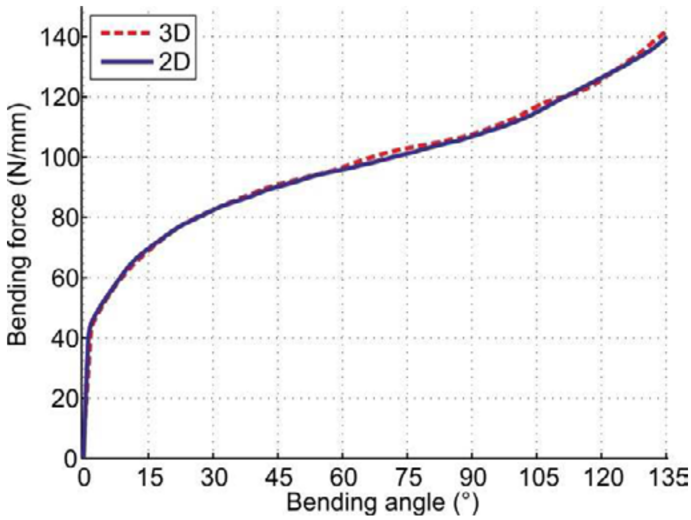

Figure 7: Bending force, 2D and 3D FEM model

5 BEAM THEORY

Analysing the results of 2D and 3D Finite Element computations, it turns out that the cross-sections approximately remain plane even for large thickness to width ratio. It also turns out that shear deformation and thickness contraction play an important role. Hence, a shear-deformable beam element of the Timoshenko-type, with transverse deformability appears to be a good candidate for further model reduction, consequently the bending process has been modelled by appropriate beam elements considering these effects. The goal of this study is the derivation of a simulation model considering the most important process parameters.

5.1 Modeling

Figure 8 shows the used model, which has been implemented in the software package HOTINT, see Gerstmayr et al. [5]. The sheet metal is clamped between the clamping tools, which are assumed rigid and rigidly fixed. The tool, modelled as a rigid circle follows a prescribed tool path which coincides with the tool path in the 2D and 3D Finite Element of Sections 3 and 4. The sheet is modelled by a two-dimensional element (ANCF) based on the work of Nachbagauer et al. [4], extended for plastic material behaviour. The latter is defined by a bilinear flow-curve. For small strains the material behaves linear elastic. After reaching the yield stress, linear hardening has been implemented.

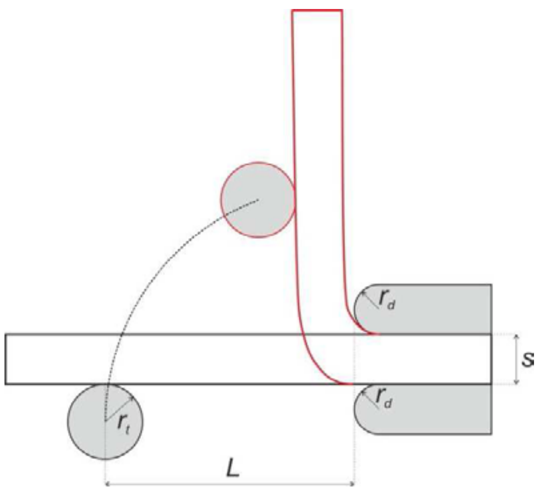

Figure 8: Sketch of the implemented model of the panel bender

The ANCF beam model has a total number of 404 variables and the computation time with the software package HOTINT is t_C = 238s (about 4min) using one CPU at a Dual Core Laptop with 4 GB RAM and a clock rate of 2.5 GHz.

5.2 Simulation Results

Figure 9 shows the comparison of the bending force per sheet width for the 2D models. The results of the ANCF beam model (computed with HOTINT) and the 2D-ABAQUS model are compared to measurement results. Exemplarily, two bending processes with sheet thickness s = 1 mm and two values for L = (5, 10) mm are investigated. Note that these process parameters differ from the investigations in chapters 3 and 4.

Figure 9 shows an acceptable accordance between measurements and the two simulation models is obtained. As expected, the coincidence is better for the smaller ratio of s and L. Note that the maximum bending angle in this case is $\varphi = 90°$, so there is no increase of the force like it can be seen in Figure 4 and in Figure 7.

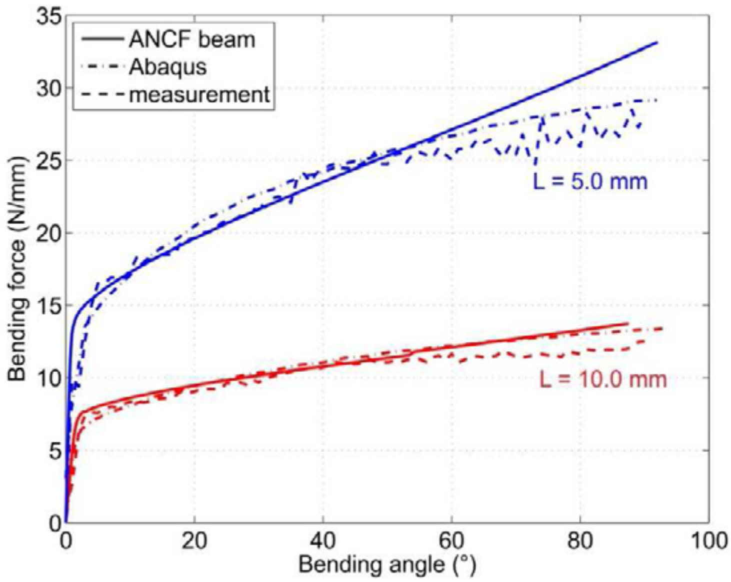

Figure 9: Bending force for ANCF Beam (HOTINT), 2D FEM model (ABAQUS) and measurements

Note that in the ABAQUS model an exponential flow curve has been implemented based on results of tension tests, wherever in the ANCF beam model a bilinear flow curve is implemented as mentioned above. A refinement of the ANCF beam element for an exponential flow curve should improve these results**.** This is the next step planned for further investigations.

The von Mises stresses are shown for the ANCF Beam (HOTINT) and 2D FEM model (ABAQUS) in Figure 10 and 11 and an acceptable coincidence can be seen in the results for the same position of the bending tool.

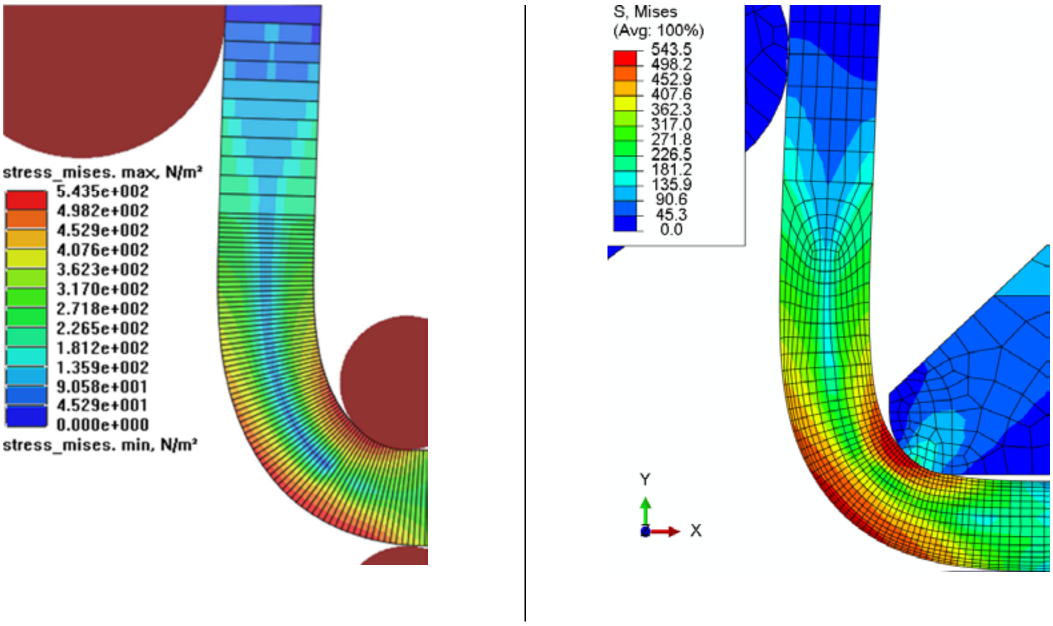

Figure 10: v. Mises stress in sheet for L=5.0mm, left figure: ANCF Beam, right figure: 2D ABAQUS model

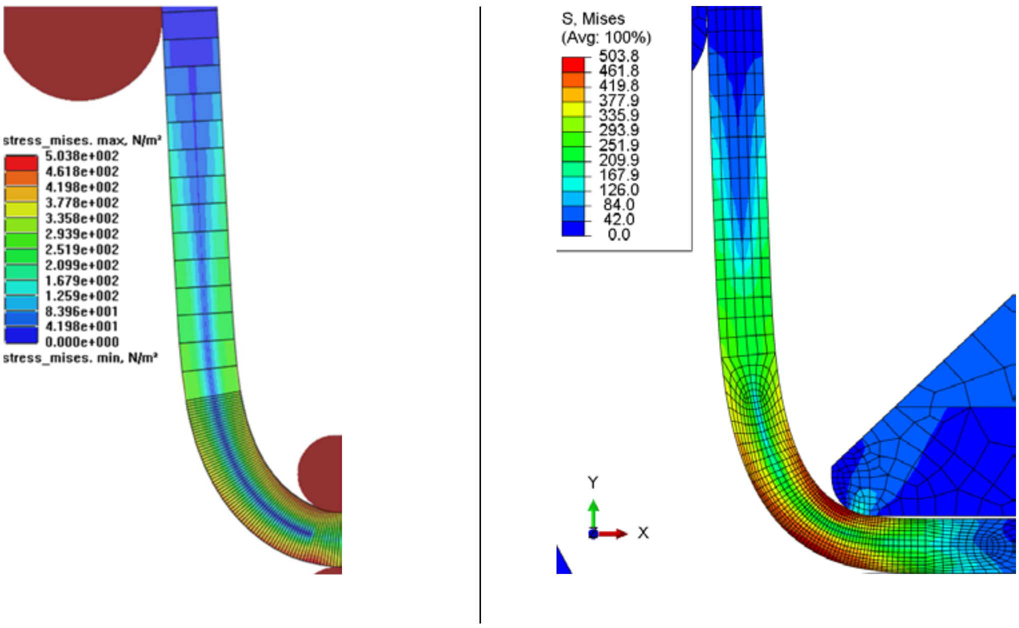

Figure 11: v. Mises stress in sheet for L=10.0mm, left figure: ANCF Beam, right figure: 2D ABAQUS model

6 CONCLUSIONS

In this paper three modelling strategies for a sheet bending process have been presented. With the 3D-model, a very detailed simulation tool is available, however at the cost of large computation times. It has been shown that the most important process parameters can be considered with a plane (2D) simulation model which has been optimized with respect to computation time. This efficient model enables detailed parameter studies. The latter show that it is also possible to model the bending process with shear deformable beam elements considering thickness contraction. Thus an ANCF beam element has been implemented yielding a third simulation model. A comparison of the three models and measurement results show a good coincidence. Hence, the presented models can be considered as efficient tools for an optimisation of the bending process.

REFERENCES

[1] Kunze, W., Zehetner, C., Cojocaru, E., Hammelmüller, F., Holl, H.J., Irschik, H., *Advanced mechanical modeling of automatic panel benders with highest accuracy demands: Process control for a one-piece flow production*, Proceedings of the 4th European Conference on Structural Control (4ECSC, St. Petersburg, Russia, 2008, ed. A.K. Belyaev and D.A. Indeitsev), Volume 2, S. 445 – 452, Russian Academy of Sciences, St. Petersburg, 2008.

[2] Zehetner, Ch., Cojocaru, E., Hammelmüller, F., Holl, H. J., Irschik, H., *Modellbasiertes Engineering zur Steigerung der Flexibilität und Effizienz bei Abkantautomaten*, Tagungsband Industrielles Symposium Mechatronik (ISM – Smart Production, 18.11.2010), Linz, Austria, 2010.

[3] Holl, H.J., Zehetner, C., Hammelmüller, F., Irschik, H., *Rechnergestützete nichtlineare Modellierung von Abkantautomaten*, e&i, Elektrotechnik und Informationstechnik, Heft 9/2004, S. 319-324.

[4] Nachbagauer, K., Pechstein, A.S., Irschik, H. and Gerstmayr J., *A new locking-free formulation for planar, shear deformable, linear and quadratic beam finite elements based on the absolute nodal coordinate formulation*. Multibody System Dynamics, 26:245-263, 2011.

[5] Gerstmayr, J., et al. *HOTINT-A Script Language Based Framework for the Simulation of Multibody Dynamics Systems*, In: Proceedings of the ASME 2013 International Design Engineering Technical Conferences & Computers and Information in Engineering Conference IDETC/CIE 2013, Portland, OR, USA, 2013. Paper No. DETC2013/MSNDC-12299.