Available online at www.sciencedirect.com

ScienceDirect

Procedia CIRP 66 (2017) 101 – 106

www.elsevier.com/locate/procedia

1st Cirp Conference on Composite Materials Parts Manufacturing, cirp-ccmpm2017

A Benchmark Study of Finite Element Codes for Forming Simulation of Thermoplastic UD-Tapes

D. Dörr[a,]*, W. Brymerski[b], S. Ropers[c], D. Leutz[d], T. Joppich[e], L. Kärger[a], F. Henning[a,e]

[a]*Karlsruhe Institute of Technology, Institute of Vehicle System Technology, Department of Lightweight Technology, 76131 Karlsruhe, Germany*
[b]*AUDI AG, N/EK-112, 74148 Neckarsulm, Germany*
[c]*VW AG, K-GERW/K, Wolfsburg, Germany*
[d]*BASF SE, E-PME/NN - F206, 67056 Ludwigshafen, Germany*
[e]*Fraunhofer Institute of Chemical Technology, Polymer Engineering Department, 76237 Pfinztal, Germany*

* Corresponding author Tel.: +49 721 608-45378; Fax: +49 721 608-945905; E-mail address: dominik.p.doerr@kit.edu

Abstract

For Finite Element (FE) simulation of composite thermoforming processes, either commercially available codes or multi-purpose FE solvers can be applied. In this work, the commercially available codes AniForm and PAM-FORM, as well as two approaches implemented in the multi-purpose FE solvers LS-Dyna and Abaqus are benchmarked in a comparative study. For this purpose, the final outer contour, as well as wrinkling behavior is analyzed in experimental forming studies and compared to the virtual predictions. It turns out, that FE forming simulation is capable to predict manufacturing defects. However, the capabilities and quality of the prediction of the considered tools differ.

Keywords: Composite forming, Process simulation, Finite element analysis (FEA), Thermoplastic resin, UD-tape

1. Introduction

Forming of two-dimensional pre-products into double-curved geometries is one of the most determining process steps in manufacturing of continuously fiber-reinforced plastics (CoFRP). Especially thermoforming of thermoplastic pre-impregnated, unidirectional-reinforced (UD) tapes is currently of great interest for the automotive industry due to low cycle times and recyclability [1,2]. However, thermoforming processes are influenced by several variables like temperature, blank holders or grippers, fiber orientation or material behavior. Dependent on these parameters, manufacturing defects like e.g. fiber fracture, gapping or wrinkling are possible. Beyond that, a change in fiber orientation is inevitable. By means of Finite Element (FE) forming simulation, manufacturing defects and the final fiber orientation are predictable, considering material behavior and process conditions. Hence, FE forming simulation facilitates the initial validation of a process and the determination of suitable process parameters. By usage of FE forming simulation as a powerful engineering tool, time and cost expensive "trial and error" process design is preventable.

In relation to the current interest in composite forming, several approaches for forming simulation are presented in literature [3-10]. In addition, some of the codes are also commercially available, which are in particular the PAM-FORM™ [3] and the AniForm™ [4] code. Furthermore, some so-called multi-purpose FE-solver offer material models for forming simulation. These are for instance LS-Dyna™ with its "*MAT_249" [8] or Abaqus™ with its "*Fabric" material model.

However, the strengths and weaknesses of the above mentioned FE codes for FE forming simulation of CoFRP are mostly unknown. Therefore, several FE codes for FE forming simulation are benchmarked in this work. Namely, these are the commercially available FE codes PAM-FORM™ [3] and AniForm™ [4], as well as two approaches implemented in LS-Dyna™ [8] and Abaqus™ [9,10]. The benchmark is based on

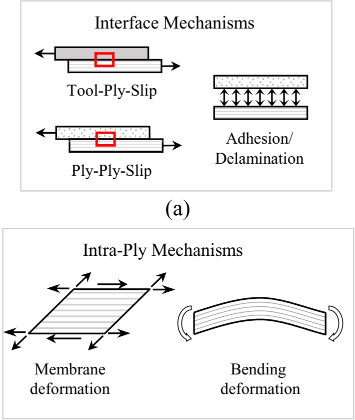
doi:10.1016/j.procir.2017.03.223

the comparison of available features and functionalities, as well as on the application of the FE codes to a generic geometry and the comparison of the simulation results to experimental tests. A comparison of the FE codes regarding CPU-times is unfortunately not possible due to the separate investigations of the considered FE codes by the authors (cf. Acknowledgments) and the execution of the simulations on different machines.

In the following, FE forming simulation, as well as the applied FE codes for forming simulation are outlined shortly and their capabilities are wrapped up in some comparing criteria. Subsequently, the processing and experimental tests of the investigated generic geometry, which is manufactured with a BASF carbon-fiber (CF) UltraTape^{TM} with a PA6 matrix, are presented. Finally, the simulation results for the generic geometry are presented and compared to the experimental tests for their capability to predict wrinkling behavior as well as for the obtained outer contour after forming.

2. Finite Element forming simulation

Forming of a two-dimensional blank into a double-curved geometry invokes so-called deformation mechanisms in the laminate. The deformation mechanisms are usually categorized into interface and intra-ply mechanisms (cf. Figure 1).



(a)

(b)

Fig. 1. Deformation mechanisms during forming of continuously fiber reinforced composites [10]

Interface mechanisms include besides the slippage and adhesion between the plies, the interaction between the tool and the formed blank through normal pressure and surface traction.

On the other hand, intra-ply mechanisms are further divided into membrane and bending deformation. Membrane behavior includes tension in and perpendicular to fiber direction, as well as in-plane shear.

FE forming simulation is based on constitutive modeling of the deformation mechanisms of the laminate (cf. Figure 1), where each of the single plies of the stacked laminate is modeled by means of separate element layers. Accordingly, interface mechanisms are modeled by means of contact constitutive equations between the adjacent plies. Regarding material behavior of a single ply, material behavior is dominated by a very high rigidity in fiber direction and a very low bending rigidity [9,11], which is attributable to possible relative motion between fibers at process conditions. This property leads to one of the main aspects in FE forming simulation of CoFRP, as conventional plate theories are no longer applicable to describe bending behavior of a single ply. Therefore, usually membrane and bending behavior are modeled in a decoupled fashion, using 2.5D-elements [4-6,9,10].

Another main aspect in FE forming simulation of CoFRP is the high degree of anisotropy accompanied with a very low shear stiffness, inducing fiber rotation during forming. This aspect has to be considered in material modeling, making conventional material frames and related material models non-applicable to FE forming simulation [9,10]. Therefore, the material behavior is usually modeled using a hyperelastic approach or a hypoelastic approach within a co-variant material frame.

Beyond that, forming tools are modeled as rigid surfaces, since the rigidity of forming tools is several orders of magnitudes higher compared to the laminate stiffness. Hence, the deformation of the forming tools is negligible.

3. Investigated FE codes and its features

In the following, some general aspects for the benchmarked FE codes, as well as the modeling approaches applied to this study are described.

3.1. PAM-FORM

PAM-FORM is a commercially available FE code, based on an explicit time integration scheme and developed to predict forming behavior of textiles or pre-impregnated materials with a unidirectional or woven fiber architecture. The modeling approach is based on assigning characteristic curves for the specific deformation mechanisms. These characteristic curves are assignable in dependence on state variables, as for instance in-plane shear as a function of shear angle or shear rate or bending in relation to curvature.

In the scope of this study, the material modeling approach (Mat 140) for a unidirectional reinforcement is applied, assigning characteristic curves for the specific deformation mechanisms. In this context, in-plane shear stiffness is described as a function of shear angle, bending stiffness as a function of curvature and ply-ply friction is related to slip-velocity and transversal pressure.

3.2. AniForm

AniForm is a commercially available FE code, developed to predict forming behaviour of multi-ply, continuously reinforced composites and is based on an implicit time integration scheme. The FE code is capable to predict forming behavior of UD-tapes, fabrics, organosheets and non-crimp fabrics (NCFs), having a thermoplastic, thermoset or no matrix constituent. Forming behavior is modeled by combining various elastic and viscous material models, which are connected in parallel following a Voigt-Kelvin approach, where an arbitrary number of fiber families can be modeled.

In the scope of this work, forming behavior of the CF UltraTape is modeled by a Voigt-Kelvin approach for membrane behavior and a purely elastic approach is selected to model bending behavior. Inter-ply slippage is modeled using a slip velocity and normal pressure dependent friction model.

3.3. LS-Dyna

LS-DYNA is a so-called multi-purpose FE solver and provides the material model *MAT_249 [8] for forming simulations of thermoplastic CoFRP with an explicit time integration scheme. Up to three independent fiber families can be defined to model unidirectional reinforcements, as well as woven fabrics or NCFs. The material model is composed of an anisotropic hyperelastic material model with temperature dependent elastic properties. The shear properties between the fiber families can be either defined as a scalar value or as a function of the shear angle. The latter is used in this work. Beyond that, bending behavior is described via a stress-strain function in the compressive state of the fibers for this study. By decreasing the compressive stiffness, the composite keeps its tensile stiffness, while showing a weaker bending behavior. The inter-ply contact is modeled with Coulomb's frictional law and additional TIEBREAK definition to model the tackiness of the matrix.

3.4. Abaqus

Abaqus is a so-called multi-purpose FE-solver and is suitable for FE forming simulation only within limits. Abaqus offers one built-in material model ("*Fabric"), suitable for modeling of textiles with a woven fiber architecture (fabrics). However, two main limitations exist for the application of Abaqus to FE forming of CoFRP. For one thing, decoupling of bending behavior, where for the anisotropic bending behavior large shear deformation is accounted for, is not available. Beyond that, adhesion and friction is not available instantaneously. Hence, only single plies of fabrics with approximate bending properties can be modeled with built-in methods of Abaqus.

In the context of this paper, a user-defined approach for modeling the forming behavior of thermoplastic UD-tapes presented by Dörr et al. [9] is applied. This approach is implemented in several Abaqus user-subroutines (VUMAT, VUGENS, VUINTERACTION), including fully decoupled modeling of membrane and bending behavior. Membrane and bending behavior are both modeled according to a Voigt-Kelvin approach, accounting for the distinct rate-dependency of thermoplastic UD-tapes at process conditions. Beyond that, also rate-dependent frictional behavior, as well as adhesion are modeled between the plies.

3.5. Comparison of modeling capabilities and functionalities

The investigated FE codes described above are compared in Table 1 based on some comparative criteria regarding modeling capabilities and functionalities like tailoring determination or fiber orientation. The information given in Table 1 considers the commercially available FE codes PAM-FORM and AniForm, as well as the presented approaches implemented in LS-Dyna and Abaqus.

Table 1. Comparing features for the investigated FE codes.

Comparing feature	PAM-FORM	AniForm	LS-Dyna	Abaqus*
Fully decoupled bending behavior	Yes	Yes	No	Yes*
Assignment of characteristic curves	Yes	No	Yes	No*
Deformation described by constitutive models	No	Yes	No	Yes*
Rate dependent membrane behavior	Yes	Yes	No	Yes*
Rate-dependent bending behavior	No	Yes	No	Yes*
Rate-dependent interface mechanisms	Yes	Yes	Yes	Yes*
Thermo-mechanical modeling	Yes	Yes	Yes	No*
Modeling of grippers / clamp holders	Yes	Yes	Yes	Yes*
Tailoring determination	Yes	Yes*	No	Yes*
Export fiber orientation	Yes	Yes*	Yes*	Yes*

*Only available by customized developments.

Regarding the criteria listed in Table 1, it turns out that for the approach in Abaqus and LS-Dyna some general features for FE forming simulation are not available or only available by the implementation of own models, as for instance fully decoupled bending behavior. Furthermore, there is a difference in the strategies applied to material modeling. On the one hand, PAM-FORM and LS-Dyna model material behavior by parameterizing the specific deformation mechanisms (cf. Fig. 1) by means of characteristic curves. On the other hand, AniForm and the approach in Abaqus model material behavior based on constitutive models, which may be more amenable for the interaction between deformation mechanisms.

Another main aspect for modeling forming behavior of thermoplastic UD-tapes is the distinct rate-dependent material behavior, as shown by several authors [9,11,13,14]. Only PAM-FORM, AniForm and the approach in Abaqus are capable of considering this material characteristic.

Regarding the engineering functionalities of tailoring determination and fiber orientation export, only PAM-FORM offers these capabilities as commercially available FE code for forming simulation. AniForm and the approach in Abaqus offer these functionalities only based on customized methods. The same applies for the fiber orientation export in LS-Dyna.

Besides modeling capabilities also operability is a criteria to be compared. For AniForm and PAM-FORM a forming simulation model is obtained after several minutes based on appropriate input-data (e.g. meshed tools or laminate), whereas more manual work has to be done for LS-Dyna and Abaqus.

4. Comparison and validation of FE forming simulation

In the scope of this study, a BASF UltraTape^{TM} (B3WC12 UD02) with a PA6 matrix and a unidirectional carbon-fiber reinforcement is applied. For manufacturing of pre-consolidated blanks, which are the pre-product for thermoforming processes, spot-welded blanks are heated in an oven well above the melting temperature of the thermoplastic and are consolidated by cooling the blanks under constant transversal pressure in a hydraulic press. The appropriate spot-welded blanks are prepared utilizing the fully automated tape-laying machine FiberForge Relay 2000 at Fraunhofer IGCV in Augsburg, Germany. Thermoforming of the pre-consolidated blanks is conducted on a Dieffenbacher 630 t hydraulic press, using industrial scaled equipment, comprising a linear transfer system including needle grippers and infrared heaters. In the scope of this study, the blank is formed freely without inducing membrane forces by e.g. grippers, where the melted blanks are dropped onto the lower female forming tool by the handling system and the tool stroke is subsequently conducted by the upper male tool.

The investigated generic geometry is a complexly shaped geometry comprising corner blendings and beads (cf. Figure 2). The tool measures a basal area of 300 mm x 416 mm and a depth of 150 mm. The initially flat blanks measure 400 mm x 600 mm and comprise 8 layers of thermoplastic UD-tape, to conform to the tool, which is optimized for a laminate thickness of 1.28 mm. In the scope of this study, two different layups are considered: A biaxial $[0;90]_{2s}$ and a quasi-isotropic $[0;45;90;-45]_s$ layup. For the biaxial layup, only the forming state of a fully closed mold is presented, whereas additionally a remaining tool stroke of 5 mm is investigated for the quasi-isotropic layup, as a more distinct wrinkling behavior is observed for this layup and the maximum of wrinkles is expected for the nearly closed mold.

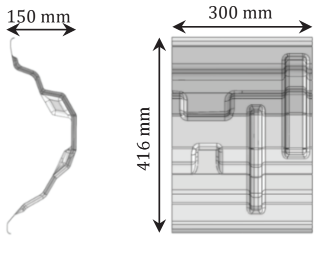
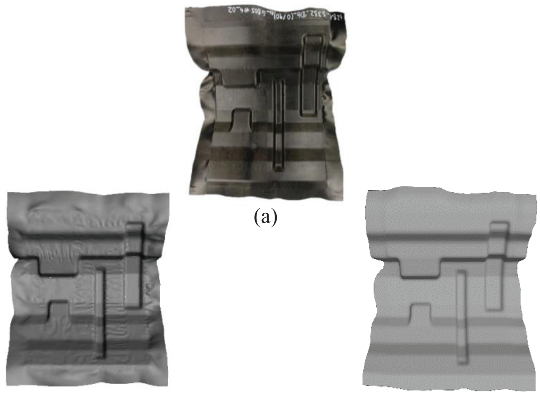


Fig. 2. Generic geometry applied to experimental tests and forming simulation.

In the following, the simulation results of the investigated FE codes for the generic geometry and the comparison of the experimental tests for the deformed surface and outer contour are presented. The results are obtained by parameterized material models according to the characterization tests presented by Haanappel et al. [12] for in-plane shear and by Sachs et al. [13] for bending behavior, which are conducted with the investigated CF UltraTape. The FE codes are parameterized by an inverse approach, modeling the characterization tests in a FE analysis.

4.1. Comparison of the deformed surfaces

The results of the experimental tests and the forming simulations are given in Figure 3 for the biaxial layup and fully closed mold. It turns out that in accordance to the experimental tests, no wrinkles are predicted by LS-Dyna, AniForm and the approach in Abaqus. On the contrary, PAM-FORM predicts wrinkles, which possibly may vanish if a further closing of the mold would be enforced. However, less distinctive wrinkling behavior is observed in experimental tests.

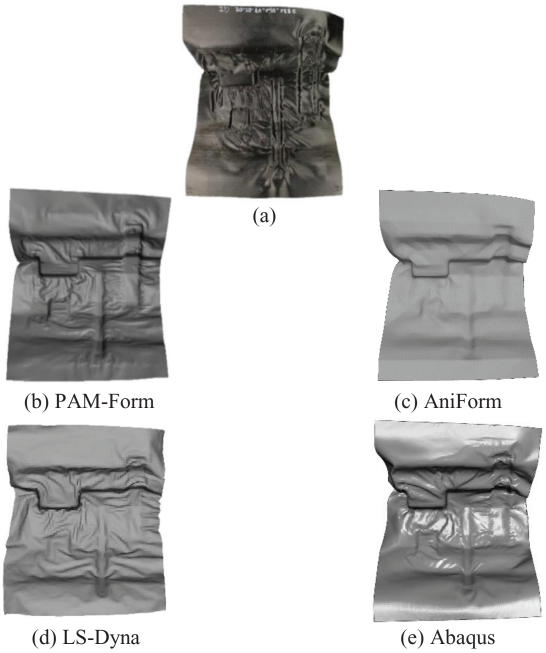

(a)

(b) PAM-Form (c) AniForm


(d) LS-Dyna (e) Abaqus

Fig. 3. Comparison forming simulation results (b-e) to experimental test (a) for the biaxial layup and fully closed mold.

Much more pronounced wrinkling behavior is observed for the quasi-isotropic layup, which is obvious for a remaining tool stroke of 5 mm (cf. Figure 4). This behavior is clearly predicted by PAM-FORM, LS-Dyna and the approach in Abaqus, where position and direction of wrinkles are predicted in good agreement to the experimental tests. Also for a fully closed mold, some defects remain visible, in accordance to the experimental tests (cf. Figure 5). For the AniForm code, wrinkles are predicted much less pronounced as observed in the experimental test, but position and direction of the defects remain slightly visible.


(a)

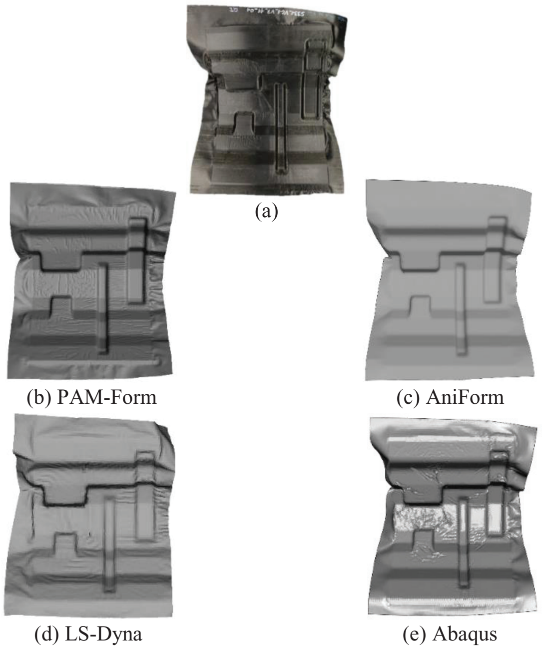
(b) PAM-Form

(c) AniForm

(d) LS-Dyna

(e) Abaqus

Fig. 4. Comparison of forming simulation results (b-e) to experimental test (a) for the biaxial layup and 5 mm remaining tool stroke.


(a)

(b) PAM-Form

(c) AniForm

(d) LS-Dyna

(e) Abaqus

Fig. 5. Comparison of forming simulation results (b-e) to experimental test (a) for the quasi-isotropic layup and fully closed mold.

4.2. Comparison of the outer contours

Besides the comparison based on the deformed surfaces, the FE codes are also compared regarding their prediction of the outer contour, which is an important target value for process design of end-contour near production. The results are based on 3D-measurements and the method presented by Dörr et al. [10]. The simulation results and experimental test results are compared for the fully closed mold and for the biaxial (Figure 6) and quasi-isotropic layup (Figure 7).

It turns out, that for the biaxial layup, the outer contour is predicted with a good agreement to the experimental tests by PAM-FORM, AniForm and the approach in Abaqus (cf. Figure 6). For the quasi-isotropic layup a good agreement is achieved by LS-Dyna, except for the upper right folding edge, and an even better agreement is observed for AniForm and the approach in Abaqus.

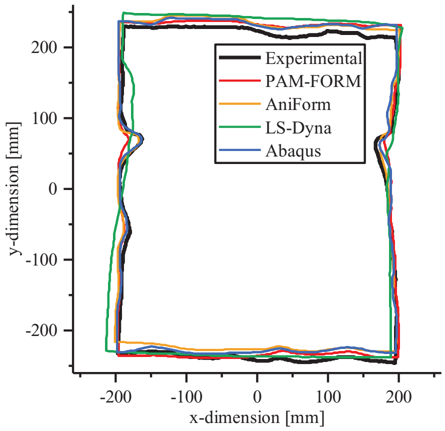


Fig. 6. Comparison of the outer contour obtained with the different simulation approaches for the biaxial layup and the fully closed mold.

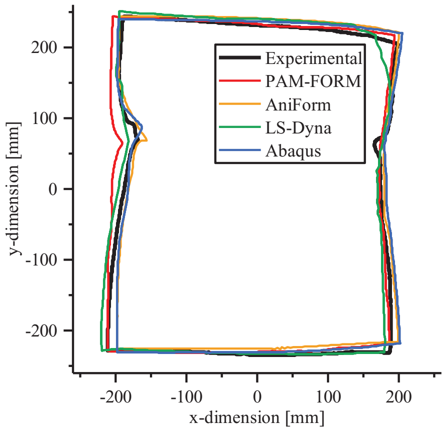


Fig. 7. Comparison of the outer contour obtained with the different simulation approaches for the quasi-isotropic layup and the fully closed mold.

5. Conclusion

Several FE codes for forming simulation of thermoplastic UD-tapes are presented and benchmarked by available features and functionalities, as well as by their quality of forming predictions. The investigated FE codes are the commercially available FE codes PAM-FORM^{TM} and AniForm^{TM}, as well as two approaches implemented in the multi-purpose FE solver LS-Dyna^{TM} and Abaqus^{TM}. The FE codes are applied to a generic, complexly shaped generic geometry and compared to experimental tests by means of the obtained deformed surface and outer contour.

Regarding the modeling capabilities and functionalities of the investigated FE codes, it is shown that the investigated FE codes have their strengths and weaknesses (cf. Table 1). The main aspect to be noted is, that as commercially available FE codes, customized for FE forming simulation of CoFRP, only PAM-FORM and AniForm are directly applicable to forming simulation. LS-Dyna and especially Abaqus require some additional work to make the software feasible for forming simulation. However, the development of own material modeling approaches is only possible for the multi-purpose FE solvers, as the FE codes customized for forming simulation do no offer appropriate user-interfaces.

The application of the investigated FE codes to the generic geometry has shown, that in summary all of the investigated FE codes offer the capability to be used as engineering tools for process design, as among others critical deformation behavior regarding wrinkles is predicted by each of the investigated codes. However, there is a difference in the quality of the prediction for the different FE codes. As it is shown, that very good results are obtained with user-defined modeling approaches, which are only available in literature, these modeling approaches should successively be incorporated into commercially available FE codes to gain the prediction quality.

In future work, the presented FE codes are applied to the process design of the demonstrator part of the project SMiLE, based on the experiences made for the presented generic geometry. The demonstrator part is a vehicle underbody structure comprising thermoplastic UD-tapes, manufactured by a new approach for handling and sequential forming.

Acknowledgements

The authors would like to thank the German Federal Ministry of Education and Research for the funding of the project SMiLE (03X3041P), for which the presented work is carried out. The presented results were generated by the participated partners of the work package “Development of methods for forming simulation of thermoplastic pre-impregnated, continuously reinforced tape structures”, where at AUDI AG AniForm, at VW AG PAM-FORM, at BASF LS-Dyna and at KIT-FAST Abaqus is investigated. The experimental tests were conducted at Fraunhofer ICT in Pfinztal, Germany and the material characterization results were supplied by TPRC in Enschede, the Netherlands, based on the commissioning by the AUDI AG and the VW AG for in-plane shear and bending characterization, respectively.

References

[1] E. Fuchs, F. Field, R. Roth, R. Kirchain, Strategic materials selection 365 in the automobile body: Economic opportunities for polymer composite design, Composites Science and Technology 68 (9) (2008) 1989–2002.

[2] J. Verrey, M. D. Wakeman, V. Michaud, J.-A. Månson, Manufacturing cost comparison of thermoplastic and thermoset rtm for an automotive 21 370 floor pan, Composites Part A: Applied Science and Manufacturing 37 (1) (2006) 9–22.

[3] P. de Luca, P. Lefébure, A. K. Pickett, Numerical and experimental investigation of some press forming parameters of two fibre reinforced thermo- 390 plastics: Apc2-as4 and pei-cetex, Composites Part A: Applied Science and Manufacturing 29 (1-2) (1998) 101–110.

[4] S. P. Haanappel, R. ten Thije, U. Sachs, B. Rietman, R. Akkerman, Formability analyses of uni-directional and textile reinforced thermoplas- 395 tics, Composites Part A: Applied Science and Manufacturing 56 (2014) 80–92.

[5] E. Guzman-Maldonado, N. Hamila, N. Naouar, G. Moulin, P. Boisse, Simulation of thermoplastic prepreg thermoforming based on a viscohyperelastic model and a thermal homogenization, Materials & Design 93 435 (2016) 431–442.

[6] P. Boisse, N. Hamila, E. Vidal-Sallé, F. Dumont, Simulation of wrinkling during textile composite reinforcement forming. influence of tensile, inplane shear and bending stiffnesses, Composites Science and Technology 71 (5) (2011) 683–692.

[7] K. Vanclooster, S. Lomov, I. Verpoest, Experimental validation of forming simulations of fabric reinforced polymers using an unsymmetrical mould configuration, Compoistes Part A 40(4) (2009) 530-539.

[8] T. Klöppel, A. Haufe, C. Liebold (DYNAMmore GmbH – Stuttgart) – Drapiersimulationen mit LS-DYNA – Sitzung der AG Composite Simulation, Denkendorf, 6. of July 2016.

[9] D. Dörr, F. J. Schirmaier, F. Henning, L. Kärger, A viscoelastic approach for modeling bending behavior in Finite Element forming simulation of continuously fiber reinforced composites, Composites: Part A 94 (2017) 113–123, doi: 10.1016/j.compositesa.2016.11.027.

[10] D. Dörr, T. Joppich, F. Schirmaier, T. Mosthaf, L. Kärger, F. Henninng, A method for validation of finite element forming simulation on basis of a pointwise comparison of distance and curvature, ESAFORM 2016 (Nantes, France), AIP Conf. Proc. 1769, 170011 (2016).

[11] S. Ropers, M. Kardos, T.A. Osswald, A thermo-viscoelastic approach for the characterization and modeling of the bending behavior of thermoplastic composites, Composites: Part A 90 (2016) 22-32, doi: 10.1016/j.compositesa.2016.11.027.

[12] S. P. Haanappel, R. Akkerman, Shear characterisation of uni-directional fibre reinforced thermoplastic melts by means of torsion, Composites Part A: Applied Science and Manufacturing 56 (2014) 8-26.

[13] U. Sachs, R. Akkerman, S.P. Haanappel, Bending characterization of UD composites, Key Engineering Materials, ISSN: 1662-9795, Vols. 611-612, pp 399-406, 2014.

[14] E. Guzman-Maldonado, N. Hamila, P. Boisse, J. Bikard, Thermomechanical analysis, modelling and simulation of the forming of pre-impregnated thermoplastics composites, Composites Part A 78 (2016) 211-222.