Available online at www.sciencedirect.com
ScienceDirect
Procedia Materials Science 3 (2014) 1441 – 1446
Procedia Materials Science
www.elsevier.com/locate/procedia

20th European Conference on Fracture (ECF20)

Advancements in package opening simulations

Eskil Andreasson[a,b,*], Joel Jönsson[a]

[a]*Tetra Pak® Packaging Solutions, SE 221 86, Lund, Sweden*
[b]*Dept. of Mech. Eng., Blekinge Institute of Technology, SE 371 79, Karlskrona, Sweden*

Abstract

The fracture mechanical phenomenon occurring during the opening of a beverage package is rather complex to simulate. Reliable and calibrated numerical material models describing thin layers of packaging materials are needed. Selection of appropriate constitutive models for the continuum material models and how to address the progressive damage modeling in various loading scenarios is also of great importance. The inverse modeling technique combined with video recording of the involved deformation mechanisms is utilized for identification of the material parameters. Large deformation, anisotropic non-linear material behavior, adhesion and fracture mechanics are all identified effects that are needed to be included in the virtual opening model.

The results presented in this paper shows that it is possible to select material models in conjunction with continuum material damage models, adequately predicting the mechanical behavior of failure in thin laminated packaging materials. Already available techniques and functionalities in the commercial finite element software Abaqus are used. Furthermore, accurate descriptions of the included geometrical features are important. Advancements have therefore also been made within the experimental techniques utilizing a combination of μCT-scan, SEM and photoelasticity enabling extraction of geometries and additional information from ordinary experimental tests and broken specimens. Finally, comparison of the experimental opening and the virtual opening, showed a good correlation with the developed finite element modeling technique.

Selection and peer-review under responsibility of the Norwegian University of Science and Technology (NTNU), Department of Structural Engineering

Keywords: Abaqus; adhesion; constitutive model; opening simulation; progressive damage

* Corresponding author. Tel.: +046-46-363269.
E-mail address: eskil.andreasson@tetrapak.com

Selection and peer-review under responsibility of the Norwegian University of Science and Technology (NTNU), Department of Structural Engineering
doi:10.1016/j.mspro.2014.06.233

1. Introduction and Background

The macroscopic behavior of the packaging material is today often described by a homogenized material definition in the finite element simulation models. This is due to unavailable experimental results of the mechanical behavior of the individual layers. Furthermore, the adhesion in-between the layers are not included in the FE-model and hence neglected. A much more accurate representation of the reality is obtained if each individual material layer is modeled as a unique layer, represented both with in-plane geometry and an out-of-plane thickness. Therefore, representing the laminated packaging material as individual layers enables more flexible simulation models. This functionality is a pre-requisite when one of layers: thickness or geometry is changed or other load cases are investigated. The mechanical behavior of highly extensible, often denoted ductile polymer films, used in the packaging industry has recently been studied by Jönsson et al. (2013). The polymer materials, consisting of different variants of polyethylene grades, are used in the packaging material structure at Tetra Pak® today.

A significant re-orientation of the polymer chains and a substantial strain-hardening occurred during the deformation process in the experimental uniaxial tensile tests. The latter effect is very important and has to be accounted for in the numerical material modeling approach. The simulations were solved in the general finite element software Abaqus version 6.13 (2013). In this work a continuum damage modeling (CDM) approach was used for each individual material layer to represent the fracture mechanisms. CDM which is attractive in macro scale applications, thus solving the engineering problems, was chosen in this study due to the computational efficiency.

A damage criterion consisting of two functionalities: initiation of damage and evolution of damage was suitable for modeling the ductile fracture behavior, cf. Andreasson et al. (2012). During the numerical analysis it has been assumed that the polymer materials are anisotropic, homogenous through the thickness, independent of strain rate and independent of temperature to ease the material parameters identification. Similar material modeling approach was used for the less extensible aluminum foil, also present in the laminated packaging material structure. A package with a post applied opening device is included for illustration in Fig. 1. This is an example of an application that is simulated numerically in this paper to show the maturity of the simulation strategy.

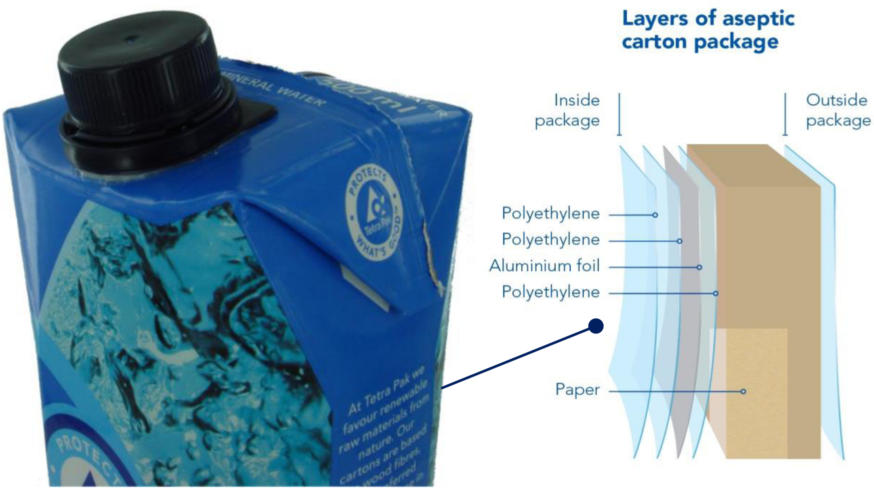

Fig. 1. Tetra Prisma Aseptic® package with a post applied screw cap opening to the left, the packaging material structure to the right.

During the opening process four topics/mechanisms are important to control, understand and accurately quantify:

- Mechanical material behavior - *stretching of the membrane*, all packaging material layers
- Progressive damage material behavior - *cutting of the membrane*, all packaging material layers
- Adhesion - *traction law between the individual packaging material layers*, all packaging material layers
- Contact/interaction - *friction between the cutter/membrane and between the frame/cutter/cap*, all included parts

The membrane, that is cut through, during the opening process consists of a packaging material structure that is shown in Fig. 2. The packaging material membrane consists of four different layers: decor polymer, laminate polymer, aluminum foil and inside polymer. Furthermore, in the finite element simulation model is the cutter part in the opening device together with the membrane included with a dense element mesh as shown in Fig. 2.

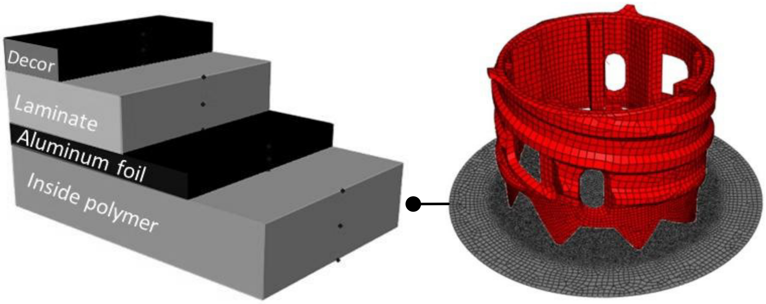

Fig. 2. Material structure of the membrane to the left and the finite element model of the membrane and the cutter to the right.

2. Identification of material parameters

An accurate continuum material model is fundamental when incorporating fracture mechanical behavior in the material model in the FE-simulations. The material properties of each material layer were determined by performing experimental uniaxial tensile tests. Individual thin films were tested, consisting of the same materials and produced with similar manufacturing process as the layers in the laminated packaging material, i.e. each polymer film represented a unique layer in the packaging material. Numerical material model parameters were identified with the inverse modeling approach complemented with the photoelastic effect, cf. Jönsson et al. (2013). This was easily adopted and possible to accomplish due to the thin transparent polymer film. Accounting for a significant strain hardening in the polymer layers is important in these highly extensible polymer films.

The results from the calibrated continuum material models used in the virtual tensile tests replicating the experimental tensile test are presented in Fig. 3. A very good fit was possible to obtain when strain-hardening was included in the two different polymer material models. Most often the material model is later on used beyond the validity of the calibration. The stress state can also be more complex and for instance include a cyclic behavior with a combined loading/un-loading scenario. In the presented simulation model the primarily focus is on the monotonic loaded mechanical behavior including progressive damage behavior.

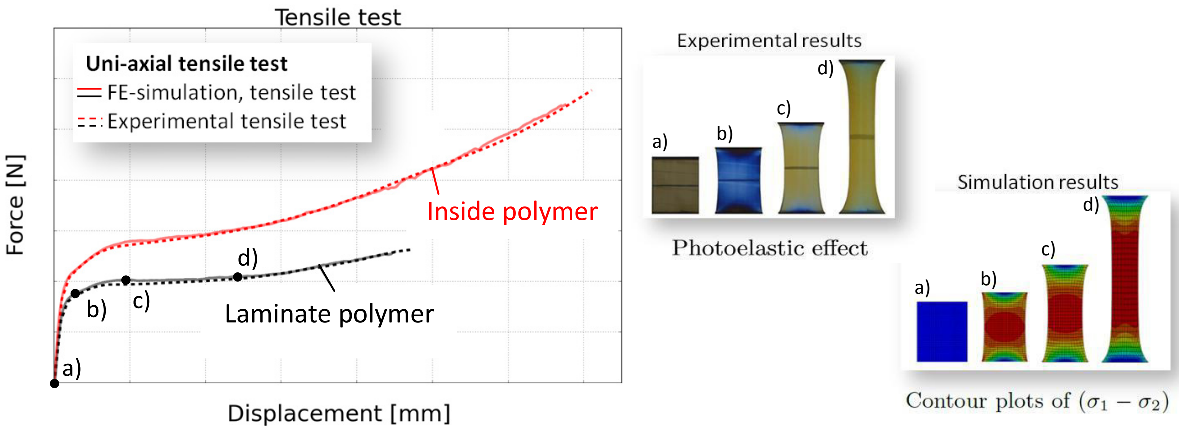

Fig. 3. Comparison of the virtual and the experimental tensile test response graphs with the corresponding deformation to the right.

A result graph from an experimental test, cf. Fig. 3., is a combination of geometrical effects, micro-mechanical mechanism and continuum material behavior. It is very important to be aware of this mixture of effects and hence try to extract the “real” and true material behavior from the specific experimental test-setup that has been performed. The material model used in the finite element software should typically not include the geometrical effects and geometrical shape effects. During the material parameter identification process, hence the inverse modeling phase should this be accounted for. A video recording or even better a Digital Image Correlation (DIC) could be used when solving the inverse problem. Otherwise the risk is large of finding a non-unique solution to the inverse problem that is not the most accurate one. The benefit of using a video that capture the deformation sequence correlated with the experimental data is also to be able to understand the involved mechanism during the experimental tensile test. Furthermore, visualization of the deformation sequence together with the data is possible afterwards.

3. Two virtual simulation models of the package opening

Solving opening simulations in an explicit framework has both advantages and disadvantages. Contact algorithms are much more mature and easily adopted with the general contact framework now available in commercial FE-codes. Progressive fracture modeling is also a conditionally unstable event and is most often impossible to solve in an implicit code today as the authors understanding. Explicit codes was originally developed and customized for rapid and dynamic events like a car crash or drop test. The opening process on the other hand can be done rapidly but the challenge is when it is done very slowly by the customer. Small elements used to resolve a high resolution have an additional cost in an explicit code. Thus decreasing the time increment, and extends the time to solve if the total time event is rather long in reality. Numerical tricks have to be utilized such as semi-automatic mass-scaling to find a good balance between the simulation time and the experimental quasi-static steady state results.

It is very important to account for each individual material layers thickness with their respective mechanical behavior both in respect of continuum behavior and fracture mechanical responses. The packaging material layers are all extrusion coated, the polymers are applied as molten layers, and laminated with aluminum foil at high temperature. This results in an adhesion value that also has to be accounted for and needs to be included in the simulation model. The advantages of modeling the layers individually are that a single layer can be changed and different levels of adhesion can be defined between the layers.

Two different FE-simulation models were developed, one FE-model that described the membrane by using the internal composite layup within one shell element definition. The second FE-model, where the different layers were described by individual shell elements connected with cohesive contact. The membrane was modeled with a circular geometry build up with a dense mesh composed by first order shell elements with reduced integration, S3R. The FE-model can be seen in Fig. 2. The reason for using three node elements, instead of four node elements, was because it was possible to create a more stochastic mesh with equally sized three node elements for the geometry. This method with stochastically distributed elements enabled an arbitrary fracture path in the progressive damage behavior, instead of a predetermined fracture path. The cutter was controlled by a kinematic coupling to a reference point with the aim to mimic the experimental test movement. A vertical displacement and rotation around the central axis of the cutter was assigned. The edge of the membrane was locked in all degrees of freedom and thus no consideration was taken to the flexibility of the paperboard edge connected to the membrane at the outer circumference.

Both simulation models used the same contact definition, general contact with a penalty friction definition. Furthermore, both models used double precision in the submission command to the solver, because of the large amount of increments needed. Five integration points was used through the thickness direction of the shell element.

3.1. One shell element model – full adhesion level

The finite simulation model where the membrane was described using the composite layup definition was very stable, i.e. it was insensitive to changes of e.g. the material properties and the boundary conditions. Furthermore, no damping or stabilization was needed in order for the model to converge and it was possible to use high level of mass scaling and still obtain an accurate and reliable numerical solution. The disadvantage of the composite layup model is that it is only possible to simulate full adhesion between the packaging material layers. Furthermore, it is not possible for layers that are not initially adjacent to each other to interact, i.e. it is not possible for the inside polymer layer to interact with the laminate polymer layer. But still it is possible to have different continuum material and damage modeling approach assigned to each material definition.

3.2. Four shell elements model – different adhesion levels

In this simulation model all material layers in the membrane, in total four layers, were represented by four separate shell elements with different cohesive contact in-between. Numerical stabilization had to be introduced when solving the model with four individual shell elements in order to get a converged solution. The linear bulk viscosity parameter was changed. The FE-model was sensitive to the level of mass scaling. However, the model was dependant on the total time of the simulation, i.e. the deformation speed. It is important to emphasize that the effect from the deformation speed, mass scaling and stabilization on the simulation results are not independent.

The advantages of the model with four shell elements are that it is possible to include different levels of adhesion between the layers as well as controlling the level of adhesion at different sections between two layers. It is also possible to model geometry of the layers in a more accurate way compared with the composite layup model. Furthermore, it is possible for all layers to interact with each other, i.e. when the aluminum foil breaks it is possible for the inside polymer layer to come into contact with the laminate polymer layer.

4. Findings and Conclusions

In this work and by Pagani et al. (2012) it has been shown that it is now possible in an opening finite element simulation, both in respect of hardware and software, to numerically model each packaging material layer in the membrane as individual layers connected with a cohesive contact. The advantage with this approach is that it is easy to change single layers mechanical properties, thickness or geometrical shape. The FE-simulations accurately describe and are able to accurately predict the opening procedure when the four shell elements model is used.

The final results from the experimental and virtual opening process are shown in Fig. 4. The simulation results mimic the experimental behavior satisfactory for both models. It is definitely possible to predict the opening force level and the overall behavior. The four shell elements FE-model has a cohesive behavior implemented between the shell elements, defined as a contact interaction. The simulation model with full adhesion, one shell element, overestimates the cutting force which is reasonable due to full interaction between the packaging material layers. Including adhesion in the virtual model is a better representation of the reality and hence is the force response curve behavior and peak force value predicting the experimental results very good.

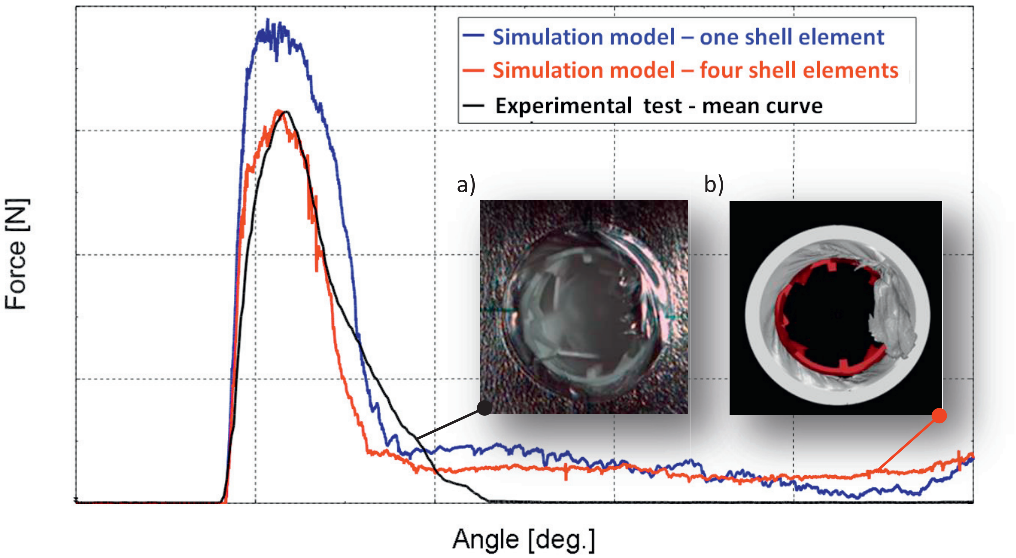

Fig. 4. Comparison of the reaction force vs. angle from the experimental test performed in Modena (Italy) and the two virtual opening models.

Acknowledgements

We would like to thank the people involved in the evolution of the FE-simulation strategy work. A team effort has made this FE-simulation model accurate, reliable and possible today. We have received valuable feedback and support during the course of the work by several people: Roberto Borsari, Alberto Mameli, Mattias Henriksson, Viktor Petersson, Johan Nordgren, Paul Håkansson and Andreas Åberg. Especially the master thesis conducted by Martin Sandgren and Joel Jönsson is worth mentioning and the further development of the opening simulation model conducted by Joel Jönsson. Manufacturing of packaging materials and experimental tests performed by Gabor Benkö, Marcus Pettersson, Nasir Mehmood and Lorenzo Marini are greatly appreciated. The work has been done with financial support from Tetra Pak®, Sweden, the KKS-profile - Model Driven Development and Decision Support and Blekinge Institute of Technology, Sweden.

Appendix A. Abaqus keywords used in the two finite element simulation models

A short description and summary of the important and specific utilized keywords extracted from the Abaqus *.inp-file that was solved in the numerical opening simulation.

```
*
** MATERIAL DEFINITIONS
**
*Material, name=Aluminium_foil
*Density
*Elastic
*Plastic
*Damage Initiation, criterion=DUCTILE
**
*Material, name=Polymer_film
*Density
*Elastic
*Plastic
*Damage Initiation, criterion=DUCTILE
*
** COHESIVE BEHAVIOR
**
*Damage Initiation, criterion=MAXS
*Damage Evolution, type=ENERGY, mixed mode behavior=POWER LAW, power=1.
*
** STEP: Virtual package opening
**
*Step, name=Cutting_through_membrane, nlgeom=YES
*Dynamic, Explicit
*Bulk Viscosity
*
** Mass Scaling: Semi-Automatic
**               Whole Model
*Variable Mass Scaling, dt=4e-08, type=below min, frequency=100
*
** INTERACTIONS – general contact
**
*Contact, op=NEW
*Contact Inclusions, ALL EXTERIOR
```

References

Abaqus Users Manual, 2013. Version 6.13-2, Dassault Systémes Simulia Corp., Providence, Rhode Island, USA.

Andreasson, E., Jemal, A., Reddy Katangoori, R., 2012. Is it possible to open beverage packages virtually? Physical tests in combination with virtual tests in Abaqus. Proceedings of the SIMULIA Community Conference. Providence, Rhode Island, USA.

Jönsson, J., Sandgren, M., 2013. Deformation and Damage Mechanisms in thin ductile polymer films - Experimental tests in combination with numerical simulations. Master Thesis, Division of Solid Mechanics, Lund University, ISRN LUTFD2/TFHF-13/5174-SE, Lund, Sweden.

Pagani, M., Perego, U., 2012. An explicit dynamics approach to the simulation of crack propagation in thin shells using reduced integration solid-shell elements, 10th WORLD Congress on Computational Mechanics - WCCM 2012, São Paulo, Brazil.