Numerical simulation of wood filled impact limiter with LS-DYNA

Karl Klein*, Johannes Will**, Thomas Seider**

* Gesellschaft für Nuklear-Service mbH, 45127 Essen, Germany
** dynardo GmbH, 99423 Weimar, Germany

Summary

CASTOR® Casks for the transport and storage of radioactive material are equipped with impact limiter reducing the loads of hypothetical accident conditions. The material wood is used to absorb the kinetic energy. Because of the specific compression behaviour of wood, a special material model has to be applied for the numerical simulation. This model has to be capable to describe the non-linear deformation behaviour taking into account the direction of fibre as well as the decrease of the volume. The program LS-DYNA provides several material models designed for foams or honeycomb structures that are able to describe compressible behaviour. For the application of these models to wood, suitable material parameters have been determined on the basis of simulations of experiments with cylindrical specimens. The determined material parameters have been applied to the simulation of a drop tests of a transport and storage cask.

Keywords

CASTOR®, impact limiter, wood, material models, drop test

---

0. Introduction

CASTOR® Casks for the transport and storage of radioactive material are equipped with impact limiters reducing the loads of hypothetical accident conditions. For the licensing as a transport cask the cask has to fulfil the requirements after the drop of the height of 9 m onto an unyielding target. These drop tests are performed as a part of the development process of the casks. For the prediction of the deformation behaviour of the impact limiter and for the optimisation of the design of the impact limiter the finite element code LS-DYNA is used. It is of central interest to model the behaviour of the energy absorbing materials, the wood, the impact limiter metal housing as well as the attachment at the cask quite precise. For the energy absorbing wood material different wood material laws are investigated. This paper focuses on the numerical modelling of the wood under impact loading and the verification with tests.

1. Deformation behaviour of wood under dynamic impact loading

Wood shows significant anisotropic deformation behaviour at elastic and plastic compression. This is resulting from its cellular microstructure where the aspect ratio of the cells is about 25. The cells are aligned with their long axis in the direction of the trunk. Therefore the deformation behaviour along the long axis of the cells (longitudinal L) is different from the behaviour of the deformation of the cells perpendicular to this direction. Whereby the difference behaviour between radial (R) and tangential (T) deformation is relative small. The Material could be described as transversal-isotrop. Fig. 1 shows the fibre orientation.

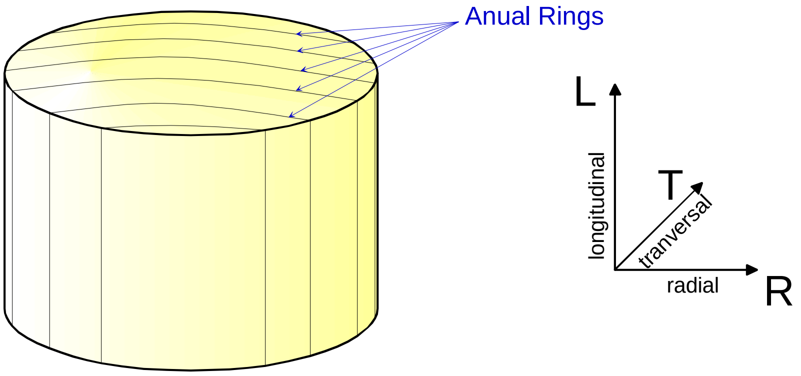

Fig. 1: Orientation of the wood fibres

For a basic characterisation of the deformation behaviour under compressive load it is essential to gain the stress strain relations for the compaction in L and R or T direction. This characterisation has been performed with cylindrical specimen for static and dynamic compression.

3. Plastic material models for the simulation of the compression of wood material

Plastic material models for finite element simulation need to have a yield criterion a flow rule and strain softening/hardening law. The yield criterion defines a surface in the multi-axial stress domain, which separates stress states leading to elastic and non-elastic deformation. The flow rule defines the direction of the inelastic deformation and the strain softening/hardening law defines the movement of the yield surface as a consequence of plastic deformation or densification. For wood densification no specific material model was found which is able to describe the non-linear transversal deformation process with all details; however there are some approaches to model the deformation behaviour of cellular material, which may be applicable to wood.

Honeycomb type material models basing on the honeycomb approach (Type 26/Type 142) are in general suitable for the description of orthotropic material, which shows extensive compressive behaviour. The elastic behaviour of the model before compaction is orthotropic where the components of the stress tensor are coupled. For honeycomb models the non-linear plastic behaviour is modelled with load curves giving the stress strain relationships for normal and shear stress directions related to the material axis. Plastic behaviour is uncoupled and a consequence of this no poisons ratio is considered. Compared to test results of wood under impact loading the transversal stress and strains are not reflected adequately. Furthermore the influence of loadings, which are not aligned with the main axis, may lead to inaccurate results. However the honeycomb models are easy to use and adequately precise for stress states which are dominated by mono-axial compressing along the main material axis (in-axis loadings).

Foam type material models (Type 57/Typ 63) are basing on material approach with defined yield surface and hardening/softening low in isotropic stress state and are able to model extensive compressive material behaviour. Because of the limitations of honeycomb type models here the suitability of foam type materials to off-axis loadings and transverse strain effects was investigated.

4. Verification of LS-DYNA material modelling approaches

For the determination of the stress deformation behaviour of the wood, cylindrical wood specimen with thin metal housing are compressed by dynamic loading. The diameter of the wood specimen is 100 mm, the height is 50 mm, the metal housing is 0.5 mm thick. Different orientations of the wood have been considered. The dynamic compression was performed by dropping an impactor onto the specimen. The stress strain curve in axial direction has been measured. To verify the basic material approach for modelling dynamic wood compaction in LS-DYNA, the impact of the test specimen has been simulated for the different wood orientations. The materials
Type 26 – MAT HONEYCOMB,
Type 63 – MAT CRUSHABLE FOAM
Type 142 - *MAT TRANSVERSEL ANISOTROPIC CRUSHABLE FOAM
Type 57 - *MAT LOW DENSITY FOAM
have been considered. Fig. 2 shows the encapsulated specimen and the finite element model.

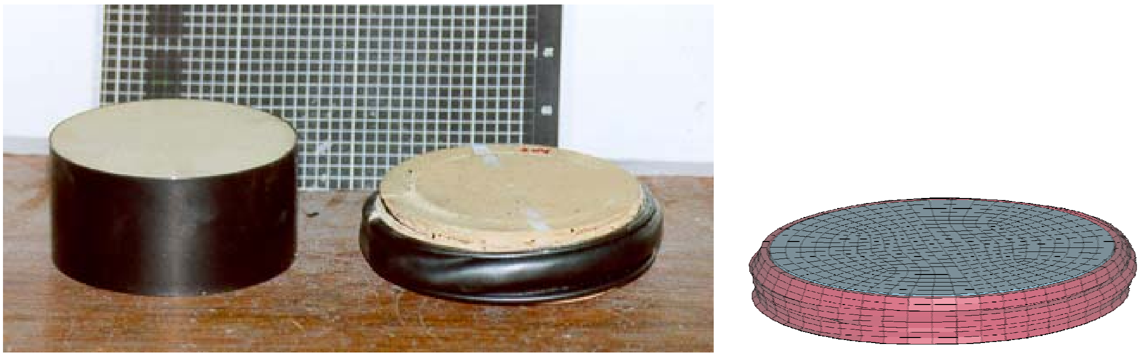

Fig. 2: Encapsulated test specimen (undeformed left; deformed right) and deformed finite element model

Simulations using honeycomb models with in-axis loadings for the longitudinal and transversal wood orientations show good agreement with the measurements in the case of the maximum deformation, maximum force and the impact time. For the drop tests where the fibre orientation of the wood was turned so that the fibre orientation shows an angle of 20°, 45° or 70° to the impact direction (off-axis loadings) the honeycomb model tends to overestimate the stiffness, so that the deformation is significant smaller and the maximum force is much to high. To avoid this effect with of – axis loading, the axis of the material model were aligned to the impact axis and measured stress strain curves for loadings with drop angle 20°, 45° or 70° was used. This in – axis configuration then showed again good agreement with the measurements. But of course for practical applications this material data has to be available for each drop orientation.

Note: Due to the possibility of introducing a stress strain curve for load at an angle, material type 142 is referred to give better results for off-axis loadings. In our case introducing these curve frequently lead to numerical instabilities. Parameter studies show great influence of different meshes on the material behaviour, but no reasonable trend was found.

The measurements show that there is a significant increase of the diameter of the specimen if the compression ratio increases. Due to the conventions of the honeycomb material approach, the simulations using material Type 26/142 showed no significant strain perpendicular to the loading direction. Due to the isotropic flow rules the foam type material models Type 57/63showed significant transversal strain, which however is not conform to the measured transversal strain of the experiment.

The wood material shows a significant unloading deformation after impact (example: initial high: 50 mm, maximum deformation 12 mm unloading deformation 25 mm) All material approaches have much to less unloading deformation and therefore showed significant differences to the experiments at the unloading phase of the impact. This results from insufficient unloading procedures and from insufficient material data about the elastic behaviour of the compressed wood.

The verification of the material models at the cylindrical test specimen shows, that the major impact behaviour of the first impact could be reflected by all material models. Secondary effects like the transversal strain or the spring back need further experimental investigation and extensions of the honeycomb type material models.

For in-axis loadings the honeycomb material model Type 26 shows the best correlation to the experimental results. Therefore this material model was used for the impact limiter simulation following next chapter. Because of the problems with off-axis loadings the wood material axis there always directed in a way that one material axis is in loading direction.

The foam type material models did not show reasonably better consideration for transverse strain or unloading behaviour..

5. Impact limiter modelling

To predict the decelerations and the impact limiter deformation at a cask drop of 9 m height the cask and the impact limiter are modelled with the LS-DYNA. The finite element model is shown in Fig. 3. The model consist of the cask which is modelled by solid elements (LS-DYNA Brick Element Type 1), the wood impact limiter modelled with solid elements (LS-DYNA Brick Element Type 1) and the metal housing modelled with shell elements (LS-DYNA Shell Element Type 10) as well as the bolts for the attachment of the impact limiter at the cask body are modelled Hughes Liu elements (LS-DYNA-Bar Element Type 1).

For the mesh 352 000 elements and 350 000 nodes are used.

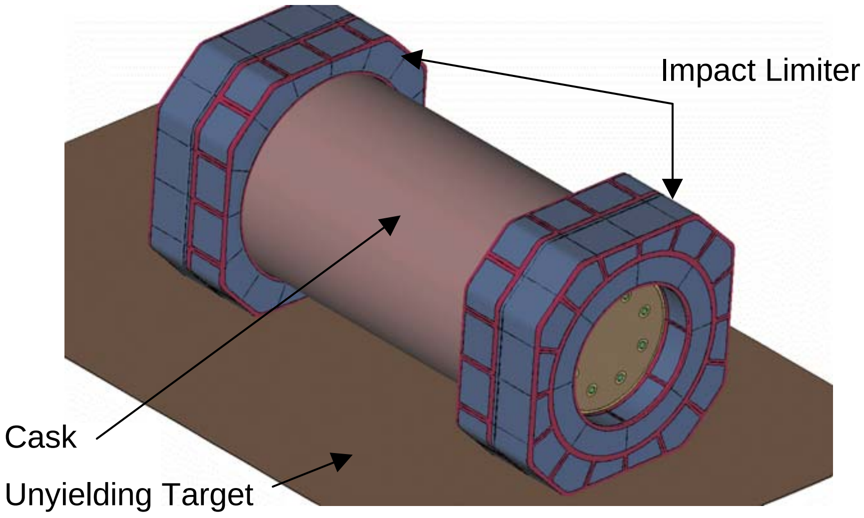

Fig. 3: finite element model of the cask and the impact limiters

For the material of the impact limiters, which are filled with the spruce, the material type 26 is used. Several drop orientation have been investigated. Fig. 4 shows the deformed structure of the impact limiter after the drop test with horizontal axis of the cask. The measured and simulated decelerations and deformations are in good accordance to the simulated values.

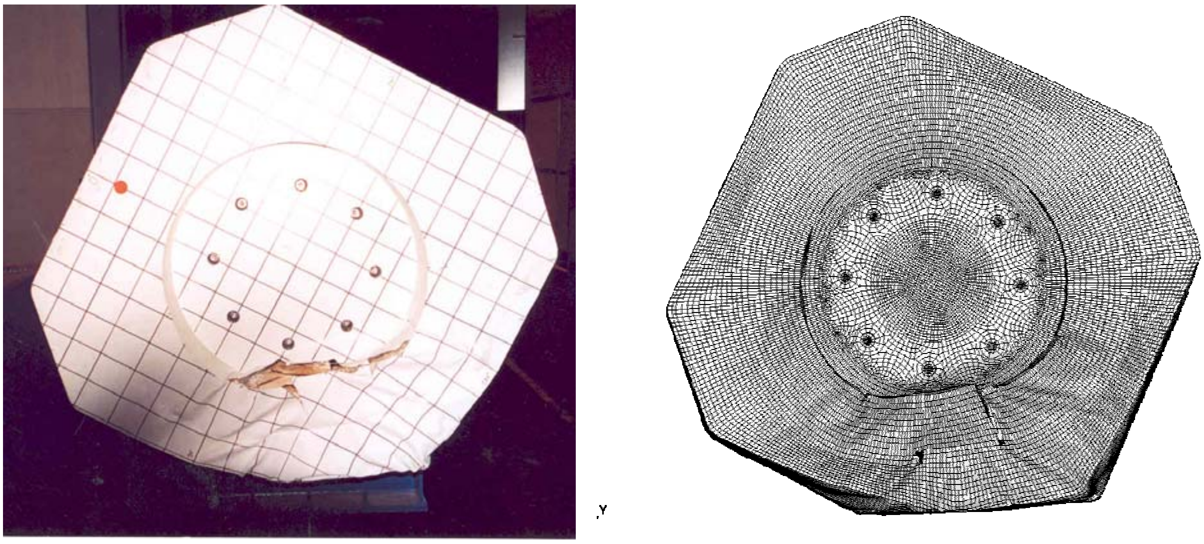

Fig. 4: deformed shape of the impact limiter after a drop of 9 m height onto a unyielding target (left: drop test; right: finite element model)

6. Conclusions

The behaviour of the impact limiters at drop test could be predicted with reasonably accuracy by the finite element code LS-DYNA. It is of great importance to qualify a suitable material model for the specific energy absorbing material. For the wood types spruce and beech this has been performed by cylindrical specimen which have been dynamically compressed. With this characterisation good agreement between the simulation results and cask drop tests has been achieved.